The Structure, Stress and Modal Analysis of 1.6-Liter Gasoline Engine Connecting Rod Based on Finite Element Analysis

Maoqun Zhang\textsuperscript{1, a}, Chao Zhao\textsuperscript{1, b}, Zhenqi Yan\textsuperscript{1, c}, Lijian Yao\textsuperscript{1, d}, Zidong Yang\textsuperscript{1, e}, Muhammad Hassa\textsuperscript{2, f}

\textsuperscript{1}School of Engineering, Zhejiang A&F University, Linan, Zhejiang 311300, China
\textsuperscript{2}US-Pakistan Centre for Advanced Studies in Energy, National University of Science and Technology, Islamabad 44000, Pakistan

*Corresponding author e-mail: ljyao@zafu.edu.cn, \textsuperscript{a}2465144243@qq.com, \textsuperscript{b}zhaochao@zafu.edu.cn, \textsuperscript{c}354276710@qq.com, \textsuperscript{d}yzd66@126.com, \textsuperscript{e}engrhasan74@yahoo.com

Abstract. The three-dimension model of engine connecting rod based on 1.6-liter gasoline Engine was built using Solid works 2016. The stress and modal analysis of the built model were evaluated by Ansys Workbench. The results showed that the maximum tensile and compressive stress of engine connecting rod when performing of pulling and pressing force were 42.59 MPa and pressing 74.21 MPa respectively. The small end of connecting rod was an area which stress concentration. Last, a modal analysis was carried out, and the previous six order natural frequency and vibration mode were obtained. The effects of the stress and losing circle shape of the rod middle area on the working properties of the connecting rod were analyzed. After checking and calculation, this 1.6-liter gasoline engine connecting rod met the strength requirement. This study could provide theoretical evidence for the further optimization and design of engine connecting rod.

1. Introduction

The connecting rod connects the piston and the crankshaft, passing the force on the piston to the crankshaft, changing the reciprocating motion of the piston to the rotating motion of the crankshaft\cite{1}. It is considered one of the key components for an internal combustion engine\cite{2}. The structure, stress and modal analysis of the connecting rod become a hot topic in engine field\cite{3}. So far, a large number of engineers and scholars have analyzed the design and simulation of all kinds of connecting rod using different software such as Ansys, Catia, Msc/Nastran. Generally speaking, there are four steps when connecting rod doing finite element analysis. (1) Establishing three-dimensional model, (2) defining the material and grid, (3) defining the restraints and loads, then applying, (4) obtaining the stress diagram and modal analysis results and analyzing\cite{1}.

At present, the application of finite element analysis in automobile engines mainly focuses on three aspects: temperature field analysis, stress field analysis and vibration mode analysis. Huang et al. established the three-dimensional model of G170 agricultural diesel engine connecting rod, and the static strength of this connecting rod was analyzed using finite element analysis. The distribution
Diagram of stress and deformation of the connecting rod under the maximum load condition were obtained[4]. Adnan et al. addressed the computation of strength and deformation characteristics of a connecting rod. They only analyzed for the axial compressive load in view of the axial compressive load is much greater than that of tensile load. Adnan et al. concluded that the most important variants of the connecting rod stress is materials. Factor of Safety and the design of connecting rod is checked and analyzed[5]. Strozzi et al. reported several typical and uncommon failure modes in connecting rod for internal combustion engines. With reference to the con-rod shank, the fatigue cracks occurring at the transition zone between the small end and the shank have been considered, and the corresponding stress concentrations have been illustrated[6]. The finite element analysis of connecting rod pays more attention to the engine used in special occasions, such as heavy duty diesel engine[7] and low speed high torque hydraulic motors[8]. Mirsadegh et al. design of high duty marine diesel engine connecting rod under maximum combustion pressure of 190 bar based on finite element analysis. The critical stress spots and magnitude of deformation of different sections are analyzed[7]. Investigation by Li et al. concluded that the connecting rod of LSHT (lower speed and high torque) hydraulic motors would deform much in the small end. The deformation could be compensated by the normal wear of the seal with babbitt alloy to ensure the static pressure[8]. As a component in reciprocating type of internal combustion engine, connecting rod has higher chance of fatigue failure when it is subjected to alternative compressive, tensile and buckling stresses[9]. Based on Kirloskar diesel engine with a compression ratio of 16.5, the study by Rao et al. determined the fatigue life of the existing connecting rod materials including Forged Steel, Aluminum Alloy and Titanium alloy. Further, the analysis is performed by using Ansys workbench for determining the vonmises stresses, deformations, life and factor of safety[10]. Now, the finite element analysis is constantly approaching the actual working condition. Su et al. studied the effects of contact affection on the finite element analysis of connection rod, taking a high-power diesel engine as an example. The importance of restrict condition is showed[11]. Li et al. calculated the strength of the connecting rod with and without considering the lubrication of oil film using finite element analysis. The results show that the calculation considering oil film lubrication is more suitable for actual working conditions[12].

To sum up, the finite element analysis of connecting rod for internal combustion engine mainly focuses on the diesel engine, especially under some special working conditions. In addition, few literatures studied the modal analysis of the connecting rod. However, 1.6-liter gasoline engine is the most common internal combustion engine using in family car. It is of great significance to perform finite element analysis for this kind of engine connecting rod. In this paper, the three-dimension model of engine connecting rod based on 1.6-liter gasoline engine was built. The structural stresses of the connecting rod after inducing gas pressure were checked, and the maximum stress of connecting rod were calculated. Finally, the modal analysis of the built model was evaluated by Ansys Workbench.

2. Materials and methods
2.1. Connecting rod parameters determination
The connecting rod we used was obtained from a 1.6-liter gasoline engine (Chery A13A, Anhui chery automobile co. LTD, Wuhu, China). The characteristic parameters of the connecting rod were listed in Table 1. C70S6 was selected as the material of the connecting rod, which was the first developed as connecting rod material in Germany. It contains 0.7% of carbon and 0.06% of sulfur and its composition is characterized by low silicon, low manganese, addition of trace alloy element vanadium and easy cutting element sulfur. The metallographic structure of C70S6 is composed of pearlite and discontinuous ferrite. The mechanical properties of C70S6 materials are shown in Table 2.
Table 1. Main parameters of the connecting rod (mm)

| Small end outside diameter | Small head hole diameter | Length | Big end hole diameter | Big end thickness | Thickness of shaft |
|----------------------------|-------------------------|--------|-----------------------|------------------|-------------------|
| 58.1                       | 40.0                    | 205.0  | 77.0                  | 40.0             | 33.0              |

Table 2. The mechanical properties of C70S6

| Material name | Modulus /GPa | Poisson’s ratio/μ | Ultimate tensile strength /MPa | Yield limit /MPa | Compression yield limit /MPa | Shear limit /MPa |
|---------------|--------------|-------------------|-------------------------------|-----------------|-----------------------------|-----------------|
| C70S6         | 200          | 0.3               | 990                           | 580             | 600                         | 665             |

2.2. The three-dimensional model establishment of connecting rod

Get the dimension of connecting rod from the CAD and make characteristic modeling in Solidworks 2016. In the modeling process, the engine connecting rod was divided into several parts to be drawn and assembled finally, which could more accurately reflect the constraints of each part. The three-dimensional diagram of the connecting rod which have been assembled was shown in Figure 1. Generally speaking, it would increase the difficulty of modeling and finite element analysis due to the errors and machining residues in the manufacture process. Therefore, it was necessary to simplify the rounding of the actual part models while drawing a three-dimensional diagram. we removed the leftover material on the solid part and some convex and concave materials that affected roughness. After simplification, make the three-dimensional model exported from Solidworks 2016 and added to Ansys Workbench software for finite element analysis.

![Figure 1. Three-dimensional model of the connecting rod](image1)

![Figure 2. The Meshing of the connecting rod](image2)

2.3. Calculation of the connecting rod load

The connecting rod is subjected to complex forces when working, such as gas-explosive force, motion inertia force, and complex friction force. The main failure of connecting rod is the fatigue caused by tensile and/or compressive stress. Therefore, this study mainly considers the force of periodic variation. One is the explosive force Fp which is produced by gas combustion. Fp causes the compression of connecting rod. the other is the reciprocating inertia force Fj which is caused by the high speed movement of the connecting rod and its own certain mass. Fj causes the tensile of connecting rod. The gas explosion force and the reciprocating inertia force have a great impact on the finite element analysis result, so the two forces should be calculated according to the specific parameters of the engine connecting rod.

The reciprocating inertia force

\[ F_j = m(1 + \lambda)r\omega^2 \]  

(1)
Where $m$ is the unbalanced rotational mass (0.468 Kg), $r$ is crank radius (40.23 mm), $\omega$ is crank to connecting rod ratio (0.27), and $\omega_i$ is the crank angular velocity (607 rad/s, calculated based on Chery automobile engine rated speed $n = 5800$ r/min). Taken into these mentioned above parameters into equation (1) and get the reciprocating inertia force $F_j = 8.8$ KN.

The gas explosive force

$$F_p = \frac{\pi}{4} D^2 (P_z - P') - F_j$$

Where $D$ is the cylinder diameter (81.0 mm), $P_z$ is the maximum absolute pressure in cylinder (4.5 Mpa), $P'$ is the standard atmospheric pressure was (0.1 Mpa). Taken into these mentioned above parameters into equation (2) and get the gas explosive force $F_p = 13.8$ KN.

### 3. Results and Discussion

#### 3.1. Meshing of the connecting rod

The three-dimensional model of engine connecting rod which building previously by Solidworks 2016 was imported into Ansys Workbench for the finite element analysis. Due to the quadratic displacement mode for more accurately simulating the connecting rod model, The Solid 186 ball element type with 20 nodes was used for mesh division. Added the C70S6 as material to the Geometry and start the grid partitioning. Free mesh was selected, and the grid was modified by the sizing command to 1 mm. Mesh is used to divide the model and the size of the grid is modified by the sizing command to 1mm. Finite element mesh is used to divide the model. Then, a total of 476,395 nodes and 279,193 units were divided. The model after meshing is shown in Figure 2.

#### 3.2. Stress analysis of the connecting rod

When the connecting rod is working, the big end is matched with the supporting bearing, while the small end is continuously affected by the gas explosive force and the inertial impact force in the range of 90°[2]. Therefore, the big end of connecting rod in this study was applied as the fixed constraint, while the corresponding calculation load was just applied to the inner surface area of the small end. According to the relevant parameters of the Chery engine and the calculation in Section 2.3, the compressive force to small end in this study was caused by the explosive force $F_p$ (13.8 KN), and the tensile force to small end in this study was caused by the reciprocating inertia force $F_j$ (8.8 kN). The principal stresses of the connecting rod under the tensile and compressive forces are shown in Figure 3. As shown in Figure 3, the maximum tensile stress of the connecting rod was 42.58 Mpa, and the maximum compressive stress was 74.21 Mpa after calculating by Ansys workbench. From the principal stress diagram of the connecting rod under tensile and compressive stress, it was clearly that both maximum principal stresses were less than the allowance stress of the material (C70S6). Therefore, the connecting rod was satisfied with the safety factor requirement. Enlarging the stress diagram and it was found that micro-deformation occurred in the small end area under either tension or compression. It could be seen that the maximum stress concentrated on the load bearing area of the small end. This is consistent with previous similar studies[5, 10]. The small end of the connecting rod was relatively weaker than the other parts of the connecting rod, and the stress concentration occurred. The maximum stress of the whole connecting rod was mainly concentrated in the bearing area of the small end (the arc transition zone), while the force on other parts of the connecting rod was more uniform under all conditions. This is in good agreement with the actual working condition[7]. Therefore, the finite element analysis could well reflect the force situation of the components.
3.3. Modal analysis of the connecting rod

During the modal analysis stage, Remote Replacement was performed. We defined the constraint that the small end of the connecting rod was constrained along the Y axis, while it could rotate along the Z axis (as shown in Figure 4). The same constraint defined to the big end of the connecting rod. The specific constraints of connecting rod in modal analysis are shown in Figure 4. After loading the constraint conditions, the previous six order vibration modes of the connecting rod were obtained, as shown in Figure 5. According to the results of previous six order vibration modes, the first order vibration mode of the connecting rod was bending vibration along the Z axis (Figure 5a), the second order vibration mode was bending vibration along the X axis (Figure 5b), the third order vibration mode was torsional vibration along the X axis (Figure 5c), the fourth order vibration mode was the bending deformation coming from the big end of the connecting rod (Figure 5d), the fifth order vibration mode was the bending deformation coming from shaft and big end (Figure 5e), and the sixth order vibration mode was the torsional deformation coming from shaft and big end (Figure 5f). From the analysis of the previous six order vibration modes, the center part of the engine connecting rod was prone to occur bending and deformation. Therefore, the design of the center part of connecting rod should be taken into consideration under the design stage, either choosing higher strength materials or properly increasing the cross-section area of the middle rod. As shown in Figure 5, all the vibration modes emerged circle lost phenomenon. The occurrence of circle lost would lead to errors in the fitting of big end and crank pin, the fitting of small end and piston pin, etc., which would cause unnecessary wear even failure such as fatigue fracture in the working process. Therefore, how to avoid or reduce the circle lost phenomenon should be taken into consideration in the design stage.
Figure 5. The modal analysis of the connecting rod. (a) The first order vibration mode, (b) The second order vibration mode, (c) The third order vibration mode, (d) The fourth order vibration mode, (e) The fifth order vibration mode and (e) The sixth order vibration mode.

4. Conclusion
The maximum tensile stress of 1.6-Liter Gasoline Engine Connecting Rod was 42.59 Mpa, and the maximum compressive stress was 74.21 Mpa after calculating by finite element analysis. Both maximum principal stresses were less than the allowance stress of the material (C70S6). Therefore, the connecting rod was satisfied with the safety factor requirement. The maximum stress of the whole connecting rod was mainly concentrated in the bearing area of the small end (the arc transition zone), while the force on other parts of the connecting rod was more uniform, and micro-deformation occurred in the small end area under either tension or compression. The center part of the engine connecting rod was prone to occur bending and deformation according to modal analysis. All the vibration modes emerged circle lost phenomenon. The occurrence of circle lost would lead to fitting errors, which would cause unnecessary wear even failure fracture. Therefore, how to avoid or reduce the circle lost phenomenon should be taken into consideration in the design stage. In all, the finite element analysis of the connecting rod was in good agreement with the actual working condition.

Acknowledgments
This work was financially supported by by Commonwealth Project of Science and Technology Agency of Zhejiang Province, China (No. 2017C32068, LGN18B06000), China Scholarship Council Project (No. 201708330483), Zhejiang Science and Technology Department Basic Public Welfare Research Project (LGN18F030001) and the Major Project of Zhejiang Science and Technology Department (2016C02G2100540).

References
[1] Gopal G, Suresh Kumar L, Vijaya Bahskar Reddy K, Uma Maheshwara Rao M, Srinivasulu G. Analysis of Piston, Connecting rod and Crank shaft assembly. Materials Today: Proceedings. 2017;4 (8):7810-7819.
[2] Ali HMM, Haneef M. Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads At The Big End. Materials Today: Proceedings. 2015;2 (4):2094-2103.
[3] Rakic S, Bugaric U, Radisavljevic I, Bulatovic Z. Failure analysis of a special vehicle engine connecting rod. Eng Fail Anal. 2017; 79: 98-109.

[4] Huang S, Huang J. Static Strength of Connecting Rod of G170 Agricultural Diesel Engine Based on Finite Element Analysis. Transactions of the Chinese Society for Agricultural Machinery. 2004;35(01):173-175.

[5] Adnan AH, Akash K, Ajinkya C, Moin K, P S. Design and Structural Analysis of Connecting Rod. International Research Journal of Engineering and Technology. 2018;5(5):282-285.

[6] Strozzi A, Baldini A, Giacopini M, Bertocchi E, Mantovani S. A repertoire of failures in connecting rods for internal combustion engines, and indications on traditional and advanced design methods. Eng Fail Anal. 2016; 60: 20-39.

[7] Seyedzavvar M, Seyedzavvar M. Design of high duty diesel engine connecting rod based on finite element analysis. J Braz Soc Mech Sci. 2018;40 (2).

[8] Li Y, Shi G, Chen Z. FEM Analysis of Connecting-rod in Low Speed High Torque Hydraulic Motors with ANSYS. Transactions of the Chinese Society for Agricultural Machinery. 2007;38 (11):137-139.

[9] Juarez C, Rumiche F, Rozas A, Cuisano J, Lean P. Failure analysis of a diesel generator connecting rod. Case Studies in Engineering Failure Analysis. 2016; 7: 24-31.

[10] Uma Maheswara Rao SV, Hanumanta Rao TV, Satyanarayana K, Nagaraju B. Fatigue analysis of sundry i.c engine connecting rods. Materials Today: Proceedings. 2018;5 (2, Part 1):4958-64.

[11] Su T, Lv C, Zhang Y, Yang S, Zhang Z, Di J. The Contact Affection to Fea of Connection Rod. Transactions of Chinese Society for Internal Combustion Engines. 2002;20 (01):79-83.

[12] Li Y, Hao Z, Zeng X. Finite Element Analysis for Connecting Rod Considering Oil Film Lubrication. Journal of Zhejiang University (Engineering Science). 2012;46 (07):1233-1237.