Finite element analysis of a vehicle diesel engine crankshaft

Fengliang Liu, Zhongcai Zheng*, Wei Luo, Dongyue Han, Yifan Zhang

Shandong Jianzhu University, School of Mechanical and Electronic Engineering, Jinan, China

* Corresponding author: zhengzx@sdjzu.edu.cn

Abstract — The crankshaft is the largest moving part in the engine and one of the most important parts in the engine. It bears the force transmitted from the connecting rod and converts this force into torque. The working conditions of the crankshaft are harsh, and its stress conditions are quite complicated. The stress concentration phenomenon is quite serious under the effect of bending and torsion loads, especially in the rounded transition area from the crank to the journal. In addition, the crankshaft has a significant impact on the overall size, weight, reliability, and manufacturing cost of the engine Therefore, the fatigue strength, stiffness, and wear resistance of the journal must be strictly checked and analyzed during the design of the crankshaft. In this paper, the finite element method is used as the main research method to analyze the static strength of a diesel engine crankshaft. The pre-processing software is used to optimize the topology of the crankshaft single turn and the tetrahedral meshing. Then the finite element analysis software is imported to set the boundary Condition and apply the load, and finally submit the calculation. According to the working conditions, the crank pin load and the main journal support reaction force are calculated. Through the analysis of the results of crankshaft statics simulation, the weak links of the stress distribution of the crankshaft under dangerous conditions are found, which provides a theoretical basis for the structural design of the crankshaft.

1. Introduction

The crankshaft is an important connection between the piston and the crankshaft in the diesel engine. It converts the reciprocating linear motion of the piston into the rotary motion of the crankshaft, and transmits the pressure acting on the top of the piston to the crankshaft to output power. The crankshaft works under the combined action of the constantly changing gas pressure, the inertial force of the reciprocating and rotating masses, and their moments (torque and bending moment). The crankshaft undergoes torsional deformation and bending deformation during work. Working conditions Very demanding. Especially in the transition area of the fillet connection between the crank and the journal, the probability of bending fatigue failure and torsion failure is quite high, which in turn leads to irreversible fracture of the crankshaft. If the design of the crankshaft is unreasonable, stress concentration or insufficient local strength or rigidity may easily occur during use, resulting in failure of the crankshaft. Crankshaft failure causes engine failure, causing extremely serious consequences.

Wang et al. [1] checked the fatigue strength of the crankshaft under alternating loads, and verified the analysis and calculation results through the fatigue test of the crankshaft.

Hu et al. [2] analyzed and compared different crankshaft materials and processes by means of probability statistics on the basis of finite element analysis, and obtained economical and reliable results.
Zhang et al. [3] calculated the fatigue life of the crankshaft based on the dynamic simulation of the crankshaft system of a six-cylinder automobile engine and the static finite element analysis of the crankshaft.

Li et al. [4] studied the comparison and analysis of calculation results and test results at the transition fillet of the journal when the crankshaft structure changes.

Liu et al. [5] used three-bit software to establish a solid model of a six-cylinder diesel engine crankshaft, and analyzed the modal response of the overall crankshaft to obtain its natural frequency and natural vibration shape.

Wang et al. [6] use finite element analysis software to perform finite element analysis on the crankshaft of internal combustion engine to determine the weak parts of its strength, and provide theoretical basis for the improvement and design of crankshaft structure.

The subject uses the finite element method as the main research method. Aiming at the crankshaft of a vehicle diesel engine, the finite element preprocessing software is used to extract the 1/2 single turn of the crankshaft from the three-dimensional model of the crankshaft, and then cut out the 1/4 single turn of the crankshaft for geometric cleaning and Topology optimization, and then 2D, 3D meshing, the 60° loading surface and constraint node set are established at the crankshaft 1/4 single turn. Import the model into the finite element analysis software, set the boundary conditions and apply the load to the crankshaft according to the actual working conditions. Through the analysis of the results of crankshaft statics simulation, the weak links of the stress distribution of the crankshaft under dangerous conditions are found, which provides a theoretical basis for the structural design of the crankshaft.

2. Crankshaft solid modeling and mechanism load analysis

2.1. Crankshaft solid modeling

According to the crankshaft size parameters of a certain vehicle diesel engine, the three-dimensional solid modeling of the diesel engine crankshaft is shown in Figure 1.

![Figure 1. Three-dimensional solid model of diesel engine crankshaft](image)

2.2. Motoring Pressure Calculations

Peak firing pressure is to be known to apply the load on the piston surface. Motoring pressure will be maximum at TDC position whereas peak firing pressure will happen at an angle of 15-20 after the TDC position.

\[
P_1 V_1^Y = P_2 V_2^Y
\]

Where \( P_1 \) is atmospheric pressure (1.1 bars) and \( P_2 \) is motoring pressure.

Therefore,

\[
P_2 = P_1 \left( \frac{V_1}{V_2} \right)^Y
\]

Where \( V_1/V_2 \) is the compression ratio and the value of the ratio of specific heats is 1.4.

Peak firing pressure for the engine is used to calculate the equivalent force acting on the piston surface. Equivalent force on the piston surface is given by the following relation. Equivalent force on the piston surface due to gas load:

\[
F_g = \text{peak firing pressure} \times \frac{2}{4} \times D^2
\]
2.3 Force Analysis of Slider Crank Mechanism

In reciprocating engines, the reciprocating masses are subjected to acceleration during the first half of the stroke. The inertia forces of the reciprocating masses oppose the motion resulting in the reduced net force on the piston. In the next half of the stroke, the reciprocating masses are subjected to retardation resulting in an increase in the net force on the piston. Since the pressure of gaseous fluid does not remain constant during the working cycle, the gasload on the piston is subject to variation throughout the working cycle.

The net force on the piston considering the gas loads FG and inertia force FI due to reciprocating parts is given by the following relation:

\[ F_P = F_G - F_I \] (4)

Where

\[ F_I = (\text{Mass of the rotating parts}) \times (\text{Acceleration/retardation of reciprocating parts}) \]

\[ \text{Mass of the rotating parts} = \frac{R}{g} \] (5)

\[ \text{Acceleration of the reciprocating parts} = A_p \] (6)

\[ (\text{Crank length}) \times (\text{Angular velocity})^2 \times \left[ \cos \theta + \cos \left( \frac{2\theta}{n} \right) \right] = r \omega^2 \left[ \cos \theta + \cos \left( \frac{2\theta}{n} \right) \right] \] (7)

Where \( n \) is the ratio of connecting rod length to crank radius \( (n = \frac{L}{R}) \) (8)

Therefore, inertia force due to reciprocating parts

\[ F_I = \frac{R}{g} \times \frac{R}{g} \times r \omega^2 \left[ \cos \theta + \cos \left( \frac{2\theta}{n} \right) \right] \] (9)

Since it is a reciprocating engine, the weight of reciprocating parts adds to the gas load when piston goes down but subtracts from gas load when piston moves up. Thus net force on the piston is given by the following relation:

\[ F_P = F_G \pm \frac{R}{g} A_p \pm W \] (10)

2.4 Crankshaft load analysis

Before loading the load, a coordinate system needs to be established. This time the coordinate system is established on the center of the main journal, that is, the main journal center coordinates are \((0, 0, 0)\), and global (global coordinate system) is used.

The reaction force acting on the main journal is distributed on the main bearing bush of the crankshaft in an axial parabola and a radial cosine of 120°, and the range is exactly the width of the bearing bush. Its distribution is shown in Figure 2. The load at the main journal of the crankshaft is distributed on the crankshaft connecting rod journal in an axial parabola and a radial cosine of 120°. The width is the width of the connecting rod head bearing bush, as shown in Figure 3.

Figure 2. Schematic diagram of main journal support reaction force
Figure 3. Schematic diagram of crank pin load

Main journal support reaction force:

\[
q(x, \theta) = 5 \times \frac{F_j}{D_{js}} \left( 1 - 4 \times \frac{x^2}{L_{js}} \right) \cos^3 \frac{\theta}{2} \tag{11}
\]

Connecting rod journal load:

\[
q(x, \theta) = \frac{5}{2} \times \frac{F_p}{D_{pl-ps}} \left( 1 - 4 \times \frac{x^2}{L_{ps}} \right) \cos^3 \frac{\theta}{2} \tag{12}
\]

Among them: \(F_p\) represents the load of the crank pin; \(F_j\) represents the supporting reaction force of the two main journals (the sign rule is: pulling positive and negative pressure can be calculated according to the simply supported beam formula); \(D_j\) represents the diameter of the main shaft diameter; \(L_{js}\) represents the width of the crankshaft main bush.

3. Geometry cleaning and topology optimization

The finite element analysis method of crankshaft mainly adopts crankshaft models including integral crankshaft method, single crank method, 1/2 crankshaft method and 1/4 crankshaft method. Among them, the integral crankshaft method is the best method for calculating results, but its calculation amount is its large size and long calculation time are generally not adopted. The model analysis method of the single knee method ignores the interaction between adjacent cranks, and simulates the constraint effect of the bearing bush on the main journal with spring constraints. The 1/2 tortuous method and the 1/4 tortuous method are simpler in comparison. The 1/2 tortuous model is symmetrical. In order to facilitate the operation and simplify the calculation, the 1/4 tortuous method and the 1/4 tortuous method are used. Obtained by 1/2 tortuous cutting.

In order to facilitate mesh division and subsequent finite element analysis and calculation, it is necessary to clean up the characteristic lines or structural lines on the geometric model that have little effect on the analysis results. Import the built 615 diesel engine crankshaft solid model and remove the solid.

3.1. Model geometry cleanup

The surface structure of the crankshaft is relatively complex. In the finite element analysis, some small bolt holes, oil holes or step surfaces will not participate in the analysis in order to reduce the difficulty of meshing and improve the quality of the mesh elements. For this reason, the model should be topologically optimized before analysis. Topology optimization work for the 1/4 crankshaft model includes: removal of free edges, filling of bolt holes, and treatment of narrow step surfaces. Prepare for the simplification and meshing of the geometric model through geometric cleaning and topology optimization of the crankshaft. The model after geometry cleaning is shown in the figure 4.
3.2. Partition of the geometric model
Crankshaft stress is mainly distributed at the junction of the main journal and connecting rod journal and crank pin, where fatigue and fracture are more likely to occur, while the working conditions of other positions are relatively good. In order to simplify the operation and improve the calculation efficiency, the model area should be reasonably divided. In order to improve the quality of the grid, the calculation accuracy is ensured while minimizing the amount of calculation. The partition of the 1/2 crankshaft geometry model is shown in the figure 5 and figure 6.

3.3. Divide the load boundary
In order to facilitate the application of a 120° cosine load on the main journal and connecting rod journal, it is necessary to generate a 120° load action boundary on the main journal and connecting rod journal, and the model here is a 1/4 crank, and a 60° load action needs to be established boundary. The load boundary of the main journal and connecting rod journal is shown in the figure 7 and figure 8.
Figure 8. Connecting rod journal load boundary

4. Pre-processing of finite element analysis

4.1. Meshing

In the finite element analysis of the crankshaft, the grid division is very important. The quality of the grid division is directly related to the calculation efficiency and the accuracy of the analysis results. This time, the tetrahedral mesh is selected for the crankshaft.

Because the crankshaft stress is mainly distributed at the junction of the main journal and connecting rod journal and crank pin, fatigue and fracture are more likely to occur here, and the quality requirements for meshing are high. The arc transition at the junction of the crank pin is meshed, and the arc transition area of the main journal, connecting rod journal and crank pin is meshed as shown in the figure 9 and figure 10.

Figure 9. Meshing of the transition zone of the main journal

Figure 10. Meshing of connecting rod neck transition zone

In this section, first perform 2D meshing on the quarter tortuous model generated in the previous chapter, then perform 3D meshing, and then use the mapping command to generate the 1/2 tortuous 3D tetrahedral mesh model as the picture shows. This operation simplifies the operation steps, improves the efficiency, and also improves the quality of the tetrahedral mesh. The 1/2 crankshaft meshing is shown in the figure 11.

Figure 11. 1/2 tortuous three-dimensional tetrahedral mesh model

4.2. Boundary conditions

Before performing finite element analysis and calculation in finite element analysis software, the model needs to be pre-processed, including: creating materials, creating section attributes, assigning
model interface attributes, setting analysis steps, setting spring constraints, setting boundary conditions, and loading loads.

In order to import the meshed model into the finite element analysis software for calculation, it is necessary to establish the loading surface, contact surface, and constraint point set of the tetrahedral mesh. According to the actual working conditions of the crankshaft, the processing of the 1/2 crankshaft three-dimensional tetrahedral mesh model mainly includes the following contents: the establishment of the loading surface, the main journal symmetry constraint node set, and the connecting rod journal displacement constraint point set establishment, The establishment of the spring constraint point set. Displacement constraint node set and spring constraint point set are shown in the figure 12 and figure 13.

4.3. Apply load

After converting the polar coordinate calculation formula of the main journal support reaction force and Connecting rod journal load into the rectangular coordinate calculation formula, it is formulas (13) and (14).

$q(X, Y, Z) = \text{Amplitude} \times \left(1 - 4 \times Y \times \left(\frac{Y}{35}\right)^2 \cos\left(1.5 \times \tan^{-1}\left(\frac{Z}{X}\right)\right)\right)$

$q(X, Y, Z) = \text{Amplitude} \times \left(1 - 4 \times (Y + 75) \times \left(\frac{(Y + 75)}{35}\right)^2 \cos\left(1.5 \times \tan^{-1}\left(\frac{Z}{(X - 65)}\right)\right)\right)$

Calculate according to the formula and apply cosine load to the main journal and connecting rod journal as shown in the figure 14 and figure 15.
5. Finite element analysis of crankshaft

5.1. Finite element statics analysis theory

The finite element method, or FEM for short, is a method for solving engineering and mathematical physics problems, also known as the finite element method. It is a method of numerical discretization, which solves its numerical solution according to the principle of variation.

The basic idea of the finite element method is to discretize a continuous object into a finite number of elements of a certain size in the mechanical model. These element elements are only connected at a finite number of nodes, and equal effects are introduced at the nodes to replace the external force acting on the elements. For each element, according to the idea of block approximation, an appropriate function is selected to express the distribution law of displacement in the element, and the relationship between element nodal force and nodal displacement is established according to the energy principle in elastic theory. Finally, the relational expressions of all the elements are assembled to obtain a set of algebraic equations with nodal displacements as unknowns.

The basic idea of finite element can be summarized as "divide first and then recombine" or "divide into zero and integrate zero into whole". Specifically: First, the continuous solution domain is discretized into a finite number of element bodies, so that they are connected to each other only at a finite number of designated nodes; then a relatively simple function is selected for each element to approximate the physical quantity of the element, and based on the problem description, the basic equations of, establish the equilibrium equations of the element nodes; then the equations of all the elements form the overall algebraic equations representing the mechanical properties of the entire structure; finally, the boundary conditions are introduced to solve the algebraic equations to obtain the numerical solution.

Static analysis is a kind of structural analysis. Its main purpose is to solve the stress and displacement of objects under static load. Static analysis mainly includes two types of analysis, one is linear analysis. According to the classical mechanics theory, the general equation of object dynamics can be obtained, which is formula (15).

\[ Mx'' + Cx' + Kx = F(t) \]  

In formula (15), \( M \) is the mass matrix, \( C \) is the damping matrix, \( K \) is the stiffness matrix, \( x'' \) is the acceleration vector, \( x' \) is the velocity vector, \( x \) is the displacement vector, and \( F(t) \) is the force vector. If the structure needs linear static analysis, then any physical quantity involved cannot be related to time, so the linear statics equation of formula (16) is obtained.

\[ Kx = F \]  

Where \( F \) is the static load.

To apply linear static analysis in structural analysis, three conditions need to be met, namely that the material meets the requirements of linear materials, the load cannot be a dynamic load and needs to be a static load, and the last one is that the deformation needs to comply with the small deformation theory.
Another type of static analysis is nonlinear analysis. There are many reasons for structural nonlinearity. It can be divided into the following situations. The first one is a series of changes in the state of the research object, and the other two are nonlinear situations, namely geometry and material.

The main solution methods of both are to discretize the physical model, that is, to divide the grid. The quality of the grid has an important impact on the accuracy of the final result. The grid can be checked by checking the quality of the element and the Jacobian. Then according to the actual environment of the structure, the model is abstracted, loads and boundary conditions are applied, and the required results are finally obtained. The difference between the two is that nonlinear static analysis needs to consider the nonlinearity of geometric materials.

5.2. Crankshaft stress analysis

5.2.1. Stress cloud

After the calculation, you can view the stress cloud diagram formed by the cosine load in the result file of the finite element analysis. The crankshaft stress cloud diagram is shown in the figure 16 and figure 17.

![Figure 16. Main journal stress cloud diagram](image)

![Figure 17. Connecting rod journal stress cloud diagram](image)

5.2.2. Maximum stress point

Through the stress cloud diagram, it can be clearly seen that the maximum stress point appears in the arc transition area between the main journal and the crank pin. The maximum stress point is shown in the figure 18.
6. Conclusions and prospects
Through the finite element analysis by intercepting the 1/2 crankshaft model of the vehicle model, the stress cloud diagram analysis obtained:

During the operation of the crankshaft, due to its special structural shape, stress concentration occurs. The stress concentration occurs in the transition area between the main journal and connecting rod journal and the crank, and the maximum external pressure is greater than that of the internal, and the maximum pressure is 278.564Mpa, as the picture shows. For 1/2 bend, the stress state is distributed symmetrically.

The analysis results obtained by using tetrahedral meshing on the crankshaft model are not much different from experience, and the accuracy is high, the calculation amount is small, and the calculation efficiency is high.

In view of the conclusion of this finite element analysis, we can make the following summary:
Crank pin fillets and main journal fillets are dangerous areas where stress concentration occurs. Strict design and check analysis should be carried out to avoid dangerous conditions such as fractures. In the design, materials with higher strength and rigidity can be selected, special heat treatment procedures can be carried out in the fillet transition area, or the crankshaft structure can be improved to reduce stress concentration from the structural shape.

Since the shape of the 1/2 crank is symmetrical, and its stress state is also symmetrically distributed, we can use the 1/4 crank in the analysis and calculation, compare and analyze the analysis results of the two models, and find more efficient finite element analysis method.

REFERENCES
[1] L. Wang, D. Hu, “Finite Element Analysis of Fatigue Strength of 368 Q Engine Crankshaft,” Journal of Internal Combustion Engine, vol. 1, no. 3, pp. 270-274, 2000.
[2] W. Hu, X. Zhai, “495B turbocharged diesel engine crankshaft fatigue reliability prediction,” Automotive Engine, no. 4, pp. 59-61, 2005.
[3] G. Zhang, B. Huang, G. Pu, C. Wang, “Crankshaft fatigue life calculation based on dynamic simulation and finite element analysis,” Internal Combustion Engine Engineering, no. 1, pp. 41-44, 2006.
[4] H. Li, S. Yang, “Study on the Law of Crankshaft Bending Stress Variation,” Automobile Technology, no. 1, pp. 13-18, 2015.
[5] D. Liu, W. Li, R. Zhang, J. Yang, and W. Shi, “Finite element analysis of crankshaft strength of automotive diesel engine based on ANSYS Workbench,” Auto Parts, no. 10, pp. 20-24, 2017.
[6] H. Wang, H. Xue, H. Wang, Y. Shang, and R. Liu, “Finite Element Analysis of 12V240 Internal Combustion Engine Crankshaft,” Mechanical Research and Application, vol. 23, no. 06, pp. 46-47, 2010.
[7] J. Sun, C. Gui, Z. Li. “Current status, discussion and prospects of the research on crankshaft strength of internal combustion engines,” Journal of Internal Combustion Engines, no. 2, pp. 179-184, 2002.
[8] Y Shi, Y Zhou. “ABAQUS finite element analysis example detailed,” Beijing: Machinery Industry Press, 2006.6