OPTIMIZATION OF CRANKSHAFT SECTION UNDER BENDING WITH CONSIDERATION OF THE MAIN STRUCTURE PARAMETERS OF THE CRANKSHAFT

Zong Yucheng, Liu Gongmin*, Liu Yushi

College of Power and Engineering, Harbin Engineering University, Harbin, China, 150001
email: liugongmin@hrbeu.edu.cn

In this paper, a V-type 16 cylinder diesel engine crankshaft is used. First, the load is applied on the finite element model of the single crank and boundary conditions are applied according to the engine mounting conditions, then obtaining the maximum displacement and Von Mises stress and analysing the position. Second, a 2-D mesh model of the vertical interface of the crank centre is established, and then the maximum stress and displacement are calculated by using ANSYS under equivalent loading conditions. The results are compared with aforementioned analysis to verify that the 2-D mesh model is reliable. Finally, the 2-D mesh model is optimized by using ANSYS PDS module with consideration of the radius of main journal, crank pin and fillet, the aim is to reduce the crankshaft quality at reasonable displacement and stress.

Keywords: crankshaft, breakage, reliability, optimization

1. Introduction

With the acceleration of globalization, the demand for large power machinery increases. As the main part of diesel engine, the reliability and economy of crankshaft are more important. One of the main failure modes of crankshaft is that the crankshaft fillet cracks or even ruptures, which can cause diesel engine to lose power, thus resulting in unpredictable consequences. Therefore, it is necessary to analyse the strength of the crankshaft to prevent failure damage because it is of great significance to ensure the performance and reliability of the diesel engine, and to guide the further structural optimization.

The Finite Element Method (FEM) is an old but effective numerical method, which has been used to solve a variety of diesel crankshaft vibration problems. Many researchers have successfully applied FEM to crankshaft vibration response calculation, and made a series of achievements. Taking a single crank as the research object, Stahl G found the influence curve by measuring the influence of the main parameters of the single crank on the fillet [1]. By testing bending failure strength of crankshafts and analysing the data, Cyrus Kano summarized the relationship between single geometrical parameter and the crankshaft strength [2]. Combining the FEM method with classical analysis techniques, Heath and Mc Namara analysed crankshaft stress and provided a reference for the optimization design of crankshaft [3]. Nallicheri et al. performed an extensive study on material alternatives for the automotive crankshaft based on manufacturing economics [4]. W Y Chien obtained the stress intensity factor of a single crank by 2-D finite mesh [5]. Gupta designed and analysed the crankshaft. Using different materials to conduct dynamic simulation of the crankshaft, Gupta optimized the crankshaft in regard to geometry and materials [6].

In this paper, in order to simplify the calculation process, a single crank model of the V-type 16 cylinder diesel engine crankshaft is set up, and then it is statically analysed by using ANSYS method.
The force and boundary conditions are loaded according to the engine’s actual working procedure. A 2-D finite element model of the vertical interface of the crank centre is set up to optimize the single crank by ANSYS PDMS module, the main variables being the diameters of main journal, crank pin and fillet radius.

2. Theoretical model

In the working process of the diesel engine, the main forces on the crank-link mechanism are the gas pressure in the cylinder, the inertial force of the reciprocating motion quality and other loads on the crankshaft. Because the effect of torsion on the stress range is relatively small, only the bending moment on the crank pin at the maximum explosion pressure moment is considered [7]. The parameters of 16V170 diesel engine are as below.

<table>
<thead>
<tr>
<th>Cylinder diameter(D)</th>
<th>170 mm</th>
<th>Rated speed(n)</th>
<th>1500 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Travel of piston(S)</td>
<td>194 mm</td>
<td>Rated power(Ne)</td>
<td>1470 kW</td>
</tr>
<tr>
<td>Compression ratio(ε)</td>
<td>13.5</td>
<td>Crank radius-connecting rod length ratio(λ)</td>
<td>0.28</td>
</tr>
<tr>
<td>Connecting rod length(L)</td>
<td>352 mm</td>
<td>Piston mass(m_p)</td>
<td>9.24 kg</td>
</tr>
<tr>
<td>The crank radius(r)</td>
<td>97 mm</td>
<td>Connecting rod mass(m_L)</td>
<td>14.10 kg</td>
</tr>
<tr>
<td>Firing order(one side)</td>
<td>1-5-2-3-8-4-7-6</td>
<td>Crankshaft mass(m_c)</td>
<td>427.43 kg</td>
</tr>
</tbody>
</table>

Table 1: The parameters of the 16V170 diesel engine

The gas pressure on the piston is as follows:

\[ P_s = \frac{\pi D^2}{4} p_e. \tag{1} \]

In the diesel engine expansion stroke, the manufacturer gives the maximum explosion pressure of 14 MPa, which happened at 360° CA, and then the gas pressure of crankshaft can be calculated by Eq. (1).

When calculating the inertial force of the crank-connecting rod mechanism, because of the different motion forms while connecting with rod tip and with rod end, the connecting rod mass is generally divided into two parts. Generally speaking, the mass distribution is shown as follows:

\[ m_1 = 0.3m_j, \quad m_2 = 0.7m_j. \tag{2},\tag{3} \]

Combined Eq. (2) with Eq. (3), the masses of reciprocating straight-line motion and rotation motion are as follows:

\[ m_j = m_p + m_1, \quad m_r = m_c + m_2. \tag{4},\tag{5} \]

Through the above analysis, the inertia force of the reciprocating straight-line motion mass and rotation motion mass can be derived:

\[ P_j = -m_j \dot{j} = -m_j r \omega^2 (\cos \varphi + \lambda \cos 2\varphi). \tag{6} \]

\[ K_{r,2} = -m_2 r \omega^2, \quad K_{r,c} = -m_1 r \omega^2. \tag{7},\tag{8} \]
Figure 1: The acting force of crankshaft-connecting rod mechanism.

Figure 1-(a) shows that the $P_g$ and $P_j$ act on the same line, there is the resultant force of them as below:

$$ P = P_g + P_j. \quad (9) $$

Figure 1-(b) shows the relationship between $K$, $N$, $S$ and $T$, which can be calculated by the following equations:

$$ K = P \cos(\varphi + \beta) / \cos \beta, \quad N = P \tan \beta. \quad (10),(11) $$

$$ S = P(1 / \cos \beta), \quad T = P \sin(\varphi + \beta) / \cos \beta. \quad (12),(13) $$

Then the force acting on the crank pin can be expressed as follows:

$$ P_c = \sqrt{T^2 + P_c^2}, \quad P_c = K + K_{r,2}. \quad (14),(15) $$

Substituting data in Table 1 into Eq. (1)-Eq. (15), and $P_c=303222$ N.

3. Finite element analysis

3.1 Single crank static analysis

In this section, 3-D mapping software Pro/E is used to establish the simplified single crank model, as shown in Fig. 2. The single crank consists of two main journals, two crank arms and one crank pin. Table 2 lists the geometrical and material properties of the single crank, the $\sigma_{\text{max}}$ and $\sigma_{\text{avg}}$ in Table 2 can be calculated by reference [8].

Figure 2: Model of a single crank
Table 2: The geometrical and material properties of the single crank.

<table>
<thead>
<tr>
<th></th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Geometry</strong></td>
<td></td>
</tr>
<tr>
<td>Diameter of main journal ((D_1))</td>
<td>160 mm</td>
</tr>
<tr>
<td>Length of main journal ((L_1))</td>
<td>67 mm</td>
</tr>
<tr>
<td>Diameter of crank pin ((D_2))</td>
<td>135 mm</td>
</tr>
<tr>
<td>Length of crank pin ((L_2))</td>
<td>132 mm</td>
</tr>
<tr>
<td>Thickness of crank arm</td>
<td>38 mm</td>
</tr>
<tr>
<td>Fillet radius ((R))</td>
<td>3 mm</td>
</tr>
<tr>
<td><strong>Material (40Cr)</strong></td>
<td></td>
</tr>
<tr>
<td>Young’s modulus ((E))</td>
<td>200 GPa</td>
</tr>
<tr>
<td>Poisson’s ratio ((v))</td>
<td>0.3</td>
</tr>
<tr>
<td>Mass density ((\rho))</td>
<td>7800 kg/m³</td>
</tr>
<tr>
<td>Tensile strength ((\sigma_b))</td>
<td>980 MPa</td>
</tr>
<tr>
<td>The failure strength of crank pin fillet ((\sigma_{DFP}))</td>
<td>303 MPa</td>
</tr>
<tr>
<td>The failure strength of main journal fillet ((\sigma_{DFW}))</td>
<td>297 MPa</td>
</tr>
</tbody>
</table>

First import the model which established in Pro/E into Hypermesh and modify the model units as millimetre. Then mesh the model and define the element type as solid 185. After that input the mesh model into ANSYS and load the boundary conditions according to the actual working environment. Next, constrain the both end faces’ displacements around X and Y axes and the main journal surface’s rotation around X axis. Figure 3 is the model that has been meshed, and oil film pressure based model of constraints and load is set as shown in Fig. 4.

After the boundary conditions and loads being applied, the next step is solving.

![Figure 3: Finite element model](image1)

![Figure 4: Model constraint and load](image2)

![Figure 5: Stress nephogram](image3)

![Figure 6: Displacement nephogram](image4)

Observing the stress nephogram and displacement nephogram shown in Fig. 5 and Fig. 6, we can find that the trend of stress and displacement is more obvious in the middle vertical plane of the crank. The maximal amount of deformation displacement reaches 0.02 mm, and the maximal von Mises stress occurs on the fillet near the main journal, which is 104.7 MPa, meeting the strength \(\sigma_{\text{von Mises}}\) of 297
MPa. There is a large strength residual from the model, and space is left to carry out lightweight optimization design.

### 3.2 2-D finite element analysis

In order to optimize the crankshaft, it is necessary to parameterize the crank. The parameterization of 3-D model is so complicated that an elastic two-dimensional plane strain finite element analysis according to reference 5 is first conducted to understand the nature of stress concentration near the fillet when the crankshaft is under bending, with the geometry of it being the middle vertical plane of the single crank and the element type being shell 181. The equivalent load and constraint are the same as Section 3.1, as shown in Fig. 7.

![Figure 7: Constraint and load of 2-D model](image)

Importing the 2-D mesh model to ANSYS, the results are as follows:

![Figure 8: Stress nephogram](image)  ![Figure 9: Displacement nephogram](image)

As shown in Fig. 8, the maximum von Mises stress is 103.8 MPa, only 1% error compared with the stress of 3-D model. The location where the maximum stress occurs is the same as that in the 3-D model. From that we can see the 2-D element model is reliable and applicable to the crankshaft optimization design.

### 4. Geometry optimization procedure

The main purpose of the research is to reduce the weight while improving or maintaining the stress performance of the original crank. The investigation of the stress nephogram shows that such locations as main journal and crank pin are subject to low stresses, so their diameters can be changed to reduce weight. In addition, in order to avoid increasing the stress level at the fillet area because of the change of D₁ and D₂, the fillet radius R has to be increased to maintain the stress level.
4.1 The effect of the $D_1$ and $D_2$

In this section, ANSYS PDS module, a sampling analysis method based on the available finite element analysis results, is applied to optimize the single crank structure. Firstly, compile analysis files with parameterized languages (APDL) of ANSYS program, then define the numerical variation range of $D_1$ and $D_2$, as shown in Table 3. Because the space of connecting rod tip is 13 mm, the change of $D_1$ and $D_2$ will not affect the overall structure of the diesel engine. Finally, the PDS module is used for sampling analysis of 500 groups of 2-D finite element models.

From Fig. 10 and Fig. 11, it can be seen that the range of the maximum stress is 80 MPa-120 MPa, it is much lower than $\sigma_{\text{WCOV}}$. The range of the maximum displacement is 0.044 mm-0.057 mm, being reasonable. Compare Fig. 12-(a) with Fig. 12-(b), we find that the linear correlation coefficient of $D_1$ and $D_2$ on the maximum stress is -0.9033 and 0.4221. In other words, the maximum stress decreases with the increase of $D_1$ and it increases with the increase of $D_2$. $D_1$ has greater influence on the maximum stress than $D_2$. After the analysis, we can get the optimized size: $D_1=150$ mm, $D_2=125$ mm, and the maximum stress referring to the new size model without the fillet is 107 MPa, a little higher than the previous unoptimized result.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Initial value</th>
<th>Low limit</th>
<th>Upper limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_1$</td>
<td>160 mm</td>
<td>150 mm</td>
<td>170 mm</td>
</tr>
<tr>
<td>$D_2$</td>
<td>135 mm</td>
<td>125 mm</td>
<td>145 mm</td>
</tr>
</tbody>
</table>

![Figure 10](image1.png)  
![Figure 11](image2.png)  
![Figure 12a](image3.png) ![Figure 12b](image4.png)

(a) The relationship between maximum stress and $D_1$.  
(b) The relationship between maximum stress and $D_2$.  

Figure 12: The influence of $D_1$ and $D_2$ on the maximum stress.
4.2 The effect of the R

In this section, the optimized size of D₁ and D₂ is applied to analyze the influence of R on the maximum stress. The PDS module is used again. As shown in Figure 13, the numerical range of R is from 0.1 mm to 6.5 mm, and the range of the maximum stress is from 96.3 MPa to 174.6 MPa. The maximum stress decreases fast with the increase of R between R is 0.1 mm to 4.5 mm, and it decreases slowly between R is 4.5 mm to 6.5 mm. Therefore, the location of R=4.5 mm is the turning point, and the maximum stress of this point is within the allowable range. Therefore the optimized size of R is 4.5 mm.

![Figure 13: The influence of R on the maximum stress.](image)

In order to ensure the reliability of the optimized results, the results should be checked. The 3-D model will be created by Pro/E with these parameters: D₁=150 mm, D₂=125 mm and R=4.5 mm, and then mesh and analyse the model with reference to Section 3.1. The optimized results are shown as below. The maximum stress of the optimized single crank is 76.0 MPa, which decreases by 28.7 MPa than the unoptimized crank. Less than σₑₓₚₓ, it meets the strength requirement. The maximum displacement is 0.0289 mm, which increases by 0.007 mm than the unoptimized crank. Less than the maximum allowable displacement value of 0.05 mm, it meets stiffness requirement. After the geometry optimization procedure, the single crank mass reduces by 3.37 kg and the whole crankshaft mass reduces by 26.99 kg, thus the purpose of reducing crankshaft quality being achieved in the case of decreasing or maintaining the stress.

![Figure 14: The stress nephogram of optimized crank.](image)

![Figure 15: The displacement nephogram of optimized crank](image)
5. Conclusions

Pro/E and Hypermesh is used to establish a 3-D finite element model of a single crank, and maximum stress and displacement under actual loading conditions are gained. Besides, the elastic two-dimensional plane strain finite element model is established to verify the rationality of the 2-D model and replace the 3-D model for calculation. In the end, geometrical parameters (D₁, D₂ and R) related maximum stress is discussed and the influence of these parameters on the maximum stress is analysed. The study finds that the D₁ is negatively correlated with the maximum stress and the D₂ is positively related to the maximum stress. D₁ has greater influence on the maximum stress than D₂, and R has a great effect on reducing the maximum stress. The optimized size is shown to provide a reference for future crankshaft optimization.

REFERENCES

1 Stahl, G., Der Einfluss der Form auf die Spannungen in Kurbelwellen, Konstruktion, 10, 61-67, (1958).