Simulation Analysis in Gear Meshing Process Based on ANSYS Software

Qiang ZHANG¹,a, Xiao-Jie WU¹,a, Qian ZHU¹,a and Yu-Lin LUO²,b

¹Department of Air-material and Aerospace Ground, Air Force Logistics College, Xuzhou, Jiangsu 221000, China
²The Army of No.95133, Wuhan, Hubei430000, China
aemail: lsc20111213@163.com(corresponding author), bemail: 215561271@qq.com

Abstract: Based on finite element analysis theory, the finite element model of gear dynamic meshing process is established by ANSYS finite element software. Through the loading analysis, results obtained the stress nephogram of gear meshing process. The results show that the maximum stress in gear meshing process is concentrated in the meshing area, and the maximum stress increases with loading time. The stress fatigue zone mainly concentrates on the pinion gear, and the stress-time curve of stress dangerous point is obtained.

1 Introduction

Because of its high transmission efficiency and compact structure, the gear system is widely used in the machinery industry. The dynamic behavior and working performance have a very important impact on the mechanical system [1]. The contact stress and transmission performance parameters in gear meshing process are very concerned by gear researchers. Many scholars have done lots of research on gear meshing analysis. LITIVIN et al. [2] uses finite element method to study the contact stress and bending stress of gears. GUINGAND et al. [3-4] used analytical method and finite element analysis method to calculate the gear stress and carried out experimental verification; BARONE et al. [5] carries out loading analysis for three-dimensional gear model in ANSYS and obtained tooth contact stress, root bending stress and other performance parameters. By present, gear research mainly focuses on the principle of gear meshing, geometric characteristics, and tooth contact analysis [6]. There is not much research on gear loading contact analysis in-depth and contact stress and stress maximum position identification and determination in gear meshing process are less.

In this paper, using the pinion gear to drive bigger齿轮，simulation analysis on the finite element model of the gear meshing in ANSYS, the contact stress and strain in gear meshing process is obtained, given the gear contact stress limit value, and the stress-time curve of the meshing point is studied. It is of certain guiding significance in the actual working condition analysis of gears.

2 The Contact Stress of Gear Calculation

In order to obtain the contact stress of gear tooth surface, the problem of gear meshing can be transformed into two cylinder contact problem with equivalent time-varying radius by means of Hertz contact theory. Suppose that the radius of curvature of the main gear and the driven gear tooth profile of the gear pair are respectively R_1 and R_2. The normal meshing force of a single pair of gear teeth is calculated by reference[7], and the normal engaging force between two gears are respectively F_{n1} and F_{n2}, then

\[ F = F_{n1} + F_{n2} \quad (\text{Eq.1}) \]

In the formula, F is the total transmission load of the gear pair, and \( F = 2T_\rho / B d_{b1} \), in which \( T_\rho \) is the gear torque, \( d_{b1} \) is the base diameter of gear, and B is the gear width.

\[ F_{n1} = \frac{F}{2} - \frac{C_i}{2} \left[ h_p(y) + h_g(y) \right] \quad (\text{Eq.2}) \]

\[ F_{n2} = \frac{F}{2} + \frac{C_i}{2} \left[ h_p(y) + h_g(y) - h_p(y + p_w) - h_g(y + p_w) \right] \quad (\text{Eq.3}) \]

In the formula, \( C_i \) is the gear deflection, \( h_i(y)(i = p, g) \) is wear depth of gear contact point, \( p_w \)
is base gear pitch, y is the distance from contact point to
the node.

3 Finite Element Simulation of Gear
Loading Meshing Process

3.1 Pre-treatment of Gear Finite Element
Analysis

The gear finite element model in meshing processes
established by using ANSYS parametric language, the
basic parameters of the big and pinion gears are
respectively shown in Table 1 and Table 2. Gear model
simulated by using 8-node solid 185 unit in ANSYS,
 meshing through body scanning, local grid refinement
for the area of gear meshing, finite element meshing
model of gear is shown in Figure 1, and the number of
meshes is 27130, the number of nodes is 31197.

The gear meshing process is essential for the rotation
of the gear according to roll ratio rotation the gear axis.
In the finite element software, the three-dimensional
solid unit generally does not have rotational freedom.
Therefore, a kinematic coupling relationship can be
established between the inner ring and the two flanks of
the small gear, coupling constraints established between
the gear tooth bottom and two tooth flanks and their own
axis reference point. In the process of gear meshing
simulation, the contact relation between the big gear
and the pinion gear is established by the contact pair in
ANSYS. The rotational freedom around the axis of the
gear is released, and the twisting is applied at 1200 N.M.
The pinion gear, as the driving wheel, the angular
displacement constraint is applied on its own axis. The
freedom of other direction of complete constrained, in
order to drive big gear rotation.

3.2 Gear Meshing Simulation Results

The stress cloud images at different stages of the gear
meshing process are obtained by ANSYS software. As
Figure 2 shown, in the process of gear rotation, the stress
of gear tooth top is the largest because it occurred
edge contact, and maximum contact stress at meshing
region is up to 179MPa. The stress extreme value mainly
concentrates on the meshing area, the value of which
increases with loading time, which accelerates the gear
wear with the work rotation of the gear. The contact
stress of extreme value area is mainly located in the
addendum of the pinion gear, where occurred easily
bending fatigue fracture pinion.

<table>
<thead>
<tr>
<th>material</th>
<th>Steel 45</th>
<th>modulus</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>poisson ratio</td>
<td>0.3</td>
<td>tooth number</td>
<td>45</td>
</tr>
<tr>
<td>elastic modulus</td>
<td>206GPa</td>
<td>Pressure Angle of pitch circle</td>
<td>20°</td>
</tr>
<tr>
<td>density</td>
<td>7850</td>
<td>addendum coefficient</td>
<td>1</td>
</tr>
<tr>
<td>tip clearance coefficient</td>
<td>0.25</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 1. The basic parameters of big gear.

<table>
<thead>
<tr>
<th>material</th>
<th>Steel 45</th>
<th>modulus</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>poisson ratio</td>
<td>0.3</td>
<td>tooth number</td>
<td>36</td>
</tr>
<tr>
<td>elastic modulus</td>
<td>206GPa</td>
<td>Pressure Angle of pitch circle</td>
<td>20°</td>
</tr>
<tr>
<td>density</td>
<td>7850</td>
<td>addendum coefficient</td>
<td>1</td>
</tr>
<tr>
<td>tip clearance coefficient</td>
<td>0.25</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2. The basic parameters of pinion gear.

In order to study the change of stress during the gear
meshing process, Figure 3 shown the stress-time curve of
the meshing node 8410 during the entire loading process.
When the gear is not engaged, the stress value is very
small and almost zero. When entering the meshing region,
stress value begins to significantly increase, which
approximately meet the Hooke's law. The stress reaches
the maximum value, which begins to maintain into the
fatigue phase, which fully satisfied the stress change law.

Figure 1. Finite element model of gear meshing process.
Conclusion

The gear meshing finite element model is established by ANSYS software. The stress map and stress-time curves of fatigue hazard points under different load stages are obtained by loading calculation. The conclusions of the study are as follows:

1. The extreme stress mainly concentrates on the meshing area, the value of which increases with loading time, the contact stress of extreme value area is mainly located in the addendum of the pinion gear, where occurred easily bending fatigue fracture pinion, which is coincident with the result of literature[2,4].

2. The stress-time curve of the dangerous point and the fatigue limit stress of meshing point are obtained, which provided the basis for gear design to prevent the fatigue failure.

References


