Structure Analysis and Optimization Design of Driving Foot
Based on ANSYS Workbench

Ming Li\textsuperscript{1,2}, Xi-fu CHEN\textsuperscript{1,*} and Qian LU\textsuperscript{1}

\textsuperscript{1}Mechanical Engineer School, Yancheng Institute of Technology, China
\textsuperscript{2}Mechanical Engineer School, Jiangsu University, China

*Corresponding author

Keywords: Linear piezoelectric motor, Driving foot, ANSYS Workbench, Optimizing design.

Abstract. Driving foot as the energy transfer mechanism in linear piezoelectric motor and the mechanical output characteristics has an important role in linear piezoelectric motor’s output. In order to improve the driving foot structure design method, the equivalent stress, deformation and modal of the driving foot structure are simulated and analyzed based on the ANSYS Workbench, and the amplification factor of driving foot is optimized. After optimizing the magnification of driving foot is 7.6, the maximum equivalent stress is 86MPa and the first-order natural frequency is 7119Hz.

Introduction
Linear piezoelectric motor is widely used in biomedicine, ultra-precision processing and positioning, aerospace and other precision positioning fields because of its high positioning accuracy, fast response, good stability and no electromagnetic interference, which has gradually become a hot topic in expert research\cite{1,2}. The non-resonant linear piezoelectric motor converts the linear reciprocating motion generated by the piezoelectric element into the elliptical motion, and output a stable linear motion through friction drive principle\cite{3,4}.

ANSYS software is a simulation analysis software developed by the world's largest finite element analysis software company, ANSYS Workbench using ANSYS computing kernel, with automatic meshing, fast parameter optimization tools, and provides a great convenience. According to the analysis and comparison of the calculation results of the scheme, according to the requirements of strength, stiffness and stability, the original scheme is modified and supplemented by the successive finite element analysis and calculation, according to a certain search method, and the reasonable stress and deformation distribution can be obtained, so as to seek the optimal solution and obtain a better structural design scheme\cite{5,6}.

The non-resonant friction-driven linear piezoelectric motor, is a typical displacement-magnified linear piezoelectric motor\cite{7}. In the non-resonant state, the output force and output displacement of the piezoelectric actuator are first transmitted to the stator driving foot, and the elliptical motion locus is generated at the output end of the driving foot in combination with the lever amplification characteristic and the flexible trigonometrical mechanism, and the active cell produces a macroscopic linear motion based on the friction drive principle. The lateral displacement determines the positioning accuracy of the linear motor, and the longitudinal displacement determines the output force of the linear motor. Taking the driving foot structure designed by our research group as a case, based on the ANSYS workbench simulation analysis software, the stress and modal of the driving foot structure are analyzed. In order to improve the amplification factor of driving foot, the structure of the driving foot is optimized, which can provide the reference for the design of the driving foot structure\cite{8}.

The Structural Analysis of Driving Foot
ANSYS Workbench is used to simulate the multi-physical environment of the actual engineering problems, can be achieved by numerical simulation of structural static analysis, structural dynamics
analysis, structural thermodynamic analysis, optimization analysis. The structural static analysis, modal analysis and optimization analysis were used in this research. The basic flow chart is shown in Figure 1.

```
Analysis type
  Build the model
  Define material properties
  Meshing
  Apply a load constraint
  Solve
  Results
```

Figure 1. The basic flow chart.

**The Previous Treatment**

The parametric modeling of the driving foot structure was designed by Pro/E, and put it into ANSYS Workbench 15.0. The material of the driving foot is 45 steel, the density $\rho=7890\, \text{kg/m}^3$, the young's modulus $E=2.09\times10^7\, \text{Pa}$, the Poisson's ratio $\nu=0.269$. Sizing control method is used to divide the grid and Element Sizing is 1mm. The results of the meshing are shown in Figure 2.

![Meshing of driving foot.](image)

Figure 2. Meshing of driving foot.

From the design methods of the linear piezoelectric motor, the constraint position of the driving foot is the bottom end. In the ANSYS Workbench, the bottom face can be defined as the fixed support. The inputs are two displacement of 2μm. The constraint results of driving foot are shown in Figure 3.

![Constraint results of driving foot.](image)

Figure 3. The constraint results of driving foot.
Static Structural

The driving foot as the transformation structure, the stability has a great impact on the output performance of motor. The static structural analysis of the driving foot is analyzed, which provides a reference for the design of the driving foot. As shown in Figure 4., the maximum equivalent stress of the driving foot is on flexible hinge, its value is 49MPa and less than the allowable working stress.

![Figure 4. The equivalent stress of driving foot.](image)

Modal Analysis

The driving foot is used to transfer the energy from the piezoelectric stack to the active cell. During the transmission, the driving foot itself is in the vibration state. If the vibration generated during the transmission is close to the natural frequency of the driving foot, which not only works contrary to the principle of non-resonant linear motor, but also affects the stability of the motor. In order to avoid resonance, the ANSYS Workbench is used to obtain the natural frequency of driving foot, and the input voltage frequency must avoid the natural frequency of driving foot. The natural frequency of the driving foot is shown in TABLE 1.

![Table 1. The natural frequency of the driving foot.](image)

<table>
<thead>
<tr>
<th>Mode</th>
<th>Natural Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6852.1 Hz</td>
</tr>
<tr>
<td>2</td>
<td>12863 Hz</td>
</tr>
<tr>
<td>3</td>
<td>19017 Hz</td>
</tr>
</tbody>
</table>

The Amplification Factor

The static structural analysis is used to calculate the effect of the structure under a fixed load, and the amplification factor of the driving foot can be obtained from the deformation results, in ANSYS Workbench software. The maximum deformation of driving foot is 9.8μm from the Figure 5., and the amplification factor is 4.9.

![Figure 5. The deformation of driving foot.](image)

Optimizing Design

The longitudinal displacement of the foot output as the optimization target, the parameters of the driving foot structure are optimized based on the multi-objective optimization method. The
parameters are set into P1-ds_1, P2-ds_2 and P3-ds_3, which P1-ds_1 means the angle between the flexible beam and the horizontal, P2-ds_2 means the thickness of the flexible beam and P3-ds_3 means the position of load. The design variable can be found in TABLE 2.

<table>
<thead>
<tr>
<th>Design variables</th>
<th>Initial Value</th>
<th>Optimal Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1-ds_1</td>
<td>12°</td>
<td>5°~15°</td>
</tr>
<tr>
<td>P2-ds_2</td>
<td>0.8mm</td>
<td>0.5mm~1.5mm</td>
</tr>
<tr>
<td>P3-ds_3</td>
<td>1mm</td>
<td>0~5mm</td>
</tr>
</tbody>
</table>

ANSYS workbench analysis can be used to test the interaction of design variables to optimize the results of the overall impact of the target to increase the reliability of optimal design results. According to the generated local sensitive Figureure, we can see that the P1-ds_1 and P2-ds_2 have a great impact on the deformation. The P1-da_1 and P2-ds_2 have a great impact on the equivalent stress.

After optimized calculation, the deformation of driving foot in the optimized driving foot is improved obviously. The optimization result is shown in Figure 7.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>ds_1</th>
<th>ds_2</th>
<th>ds_3</th>
<th>Deformation</th>
<th>Equivalent Stress</th>
<th>First-order Natural Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Optimization</td>
<td>12°</td>
<td>0.8mm</td>
<td>1mm</td>
<td>9.8μm</td>
<td>49MPa</td>
<td>6852.1 Hz</td>
</tr>
<tr>
<td>After Optimization</td>
<td>8°</td>
<td>0.7mm</td>
<td>3.5mm</td>
<td>15.2μm</td>
<td>86MPa</td>
<td>7119.1Hz</td>
</tr>
</tbody>
</table>
Conclusions

The equivalent stress, deformation and modal of driving foot are studied in this research. In order to improve the amplification factor, the structural parameters of driving foot is optimized.

According to the optimized date, the maximum deformation of the driving foot is 15.2μm, and the amplification factor is 7.6, which is 55% higher than that before optimization. After optimization, the maximum equivalent stress of the driving foot is 86MPa, and the first-order natural frequency is 7119Hz, which meets the requirements.

Acknowledgments

This research was financially supported by the Jiangsu Province Natural Science Foundation Youth Project (Grant No. BK20140474), the National Natural Science Foundation of China (Grant No. 51405420), the Jiangsu Province Ordinary University Professional Degree Graduate Innovation Project (Grant No. SJJZ16_0294) and the Basic Scientific Research Program of China.

References