STUDY OF VIBRATION RESPONSE OF IMPULSE INPUT IN GEARBOX
THE FINITE ELEMENT METHOD

Shengda Wang

Department of Mechanical and Industrial Engineering
University of Massachusetts, Amherst

ABSTRACT

Vibration monitoring is the most investigated and widely used approach of helicopter gearbox fault diagnosis. In this approach, the vibration signals are measured by accelerometers mounted on the housing and analyzed online for fault detection and isolation. There are two basic issues that need to be handled: how many accelerometers needed and where to located them for a best performance of the fault diagnosis. For the problems of sensor location and fault diagnosis, the propagation of vibrations due to the component faults needs to be studied. This project is intended to study the vibrations due impulse force with finite element of gearbox. A finite element model of the whole gearbox was built in ANSYS. Modal analysis and transient analysis were also made. The results of the model were compared with the experimental data and some simple calibrations and validations were also implemented.

Keyword: Vibration monitoring, Finite element analysis, Modal analysis, Transient Analysis.

1 INTRODUCTION

Vibration monitoring is the most investigated and widely used approach of helicopter gearbox fault diagnosis. In this approach, the vibration signals are measured by accelerometers mounted on the housing and analyzed online for fault detection and isolation. There are two basic issues that need to be handled: how many accelerometers needed and where to located them for a best performance of the fault diagnosis. Both of these two problems together with fault diagnosis issue make it important to study the propagation of vibrations due to the component faults. Such vibrations can be simulated as the vibrations caused by an impulse on the fault location (usually by hammering the location) and can be measured by accelerometers located in different places on the gearbox. Usually a Mass-lumped model is used to study the vibration propagation. The disadvantage of this method is that it cannot deal with the distributed property of the transmission path of vibration. This project is intended to study the vibration propagation and to simulate the relationship between the vibrations and the impulse input with a finite element model.

First, a finite element model of the gearbox was built with ANSYS. Second, modal analysis of the model was done to study the natural frequencies and mode shapes. Then transient analysis was made for an impulse input on the gearbox. Such impulse input simulates the component fault. The resulted vibrations on several locations (nodes in the finite element model) due to this impulse input were studied. The results of the model with different damping values were also studied and compared with the experimental data. Some methods were employed to calibrate the damping ratio and elasticity moduli by the experimental data to improve the simulation results.

1.1 Experiments and Experimental Data

Component faults can be simulated by hammering the location where the faults are supposed to be and the response vibrations can be measured by accelerometers mounted on different locations in the gearbox. The impulse by hammering can be measured by the sensor mounted on the hammer. In each experiment, a sampling time of 0.00002 seconds was used and signal in a period of 0.4 seconds was measured.

The configuration of the experiments for this project is shown in Figure 1. Location S2 is hammered and measurements on location S1 are taken. The impulse input and the vibration measurements are illustrated in Figure 2. Figure 3 illustrates the power spectral density of the response signal.

Figure 2 shows the input impulse and the response measurements. Here the initial bias of the sensors has been
removed during the data preprocess. From Figure 2, the response signal dies out after the first 0.004 seconds. This time can be used to estimate the damping value of the material.

Power spectral density of the response signal is illustrated in figure 3 and will be used to estimate the elasticity moduli of the materials.

Figure 1: Configuration of Experiments

Figure 2: Experiment Data: impulse and response

Figure 3: Power spectral density of the response

2 GEARBOX AND THE FINITE ELEMENT MODEL

The gearbox used here is an OH-58A transmission (Figure 4).

For simplicity the following assumptions are made for the modeling of the gearbox:

- The influences of the rotation of gears in the gearbox are ignored.
- The influences of the teeth mesh are ignored and contact mechanics is considered.
- The non-linearity due to the material and rotation of the gears is not considered and a linear model is adopted.
- The rotational degree of freedoms is not included.
First, material properties are defined, then areas for all parts are defined, and they are meshed. The areas are made up from key-points and lines. The lines are divided to allow for a mapped mesh. The areas are rotated 180 degree about their respective symmetry axis, i.e., the volumes rotational symmetric to the vertical axis are rotated about that axis, the input shaft areas are rotated about the symmetry axis of the input shaft assembly, and the areas of the accessory tool drive are rotated about their symmetry axis. The planets (one full planet, and one half planet, the latter one having its “open” side in the symmetry plane of the half-way rotated gearbox) and their assembly are created from key-points, lines, and areas. The planets are meshed by rotation about their symmetric axis. The assembly volumes are meshed by sweeping the top area towards the bottom.

To join the input shaft and accessory drive assembly with the main components, two cylinders are created for each assembly. The volumes of the main assembly are unmeshed, and with a boolean operation, the cylinders cut into the main components. Finally, the cylinders are deleted. The components are meshed by free mesh and the assemblies are joined to the main components by node and key-point merge operations.

To obtain a full model, the above half model is expanded into full model with a symmetry operation, using the symmetry plane. Finally, another merge of nodes and key-points is performed.

Two types of elements are used in this model. For the areas, PLANE42 elements have been used. For the extrusion, SOLID45 elements have been used.

Three different materials have been used, one for the housing of the gearbox, one for bearings, and one for the gears.

3 MODAL ANALYSIS

To study the dynamic feature of the gearbox model, modal analysis on the full model is done in ANSYS. The options of the modal analysis process are summarized in Table-1.

Displacement constraints are applied on four nodes (2822, 22928 and their symmetrical counterparts) with zero displacements for all DOFs. For simplicity all of the rotational DOFs are fixed to zero.

Table-2 is a small part of the natural frequency report of the modal analysis.

![Figure 6: Full model of OH-58A gearbox](image)

![Figure 5: Finite element model of OH-58A gearbox (half)](image)
3 TRANSIENT ANALYSIS

Due to the long time that transient analysis needs, a half model is used in this study. For the first computation, no damping ratio is included. The power spectral density of the model result is compared with that of the experimental data and refined elasticity moduli are estimated. Then a damping value is included and the result of the model together with the experimental data is used to estimate the refined damping value. For simplicity, only $\beta$ damping is considered.

3.1 ANSYS Options for Transient Analysis

The ANSYS options for transient analysis are listed in Table-3.

<table>
<thead>
<tr>
<th>ANSYS Options</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANTYPE</td>
<td>Transient</td>
</tr>
<tr>
<td>Solution method</td>
<td>Reduced</td>
</tr>
<tr>
<td>Lumped mass approx</td>
<td>Yes</td>
</tr>
<tr>
<td>Damping effects</td>
<td>Not included</td>
</tr>
<tr>
<td>Incl prestress effects</td>
<td>No</td>
</tr>
</tbody>
</table>

3.2 Boundary Conditions

Symmetrical boundary conditions are applied on the area of X-Y plane with Z = 0.

Zero displacement for all of the DOFs is applied on several nodes on the bottom of the model.

3.3 The Impulse Load

The impulse force load in Y direction was applied on one node in the spider. The force load is illustrated in Figure 7. The bottom figure is the enlarged figure for the upper one. The width of the impulse signal is $0.00002 \times 3 = 0.00006$ (s) and the magnitude is 16666.7 with a direction of $-Y$.

The time step of the computation is 0.00002 (s), which is the same as the sampling time of the experiments. The simulation period is 0.08 (s), during which the transient process is supposed to converge.

3.3 Results of Undamping Model

The vibration responses of the impulse load are measured on several nodes. Figure 8 illustrates the locations of the load and measurements. Totally 16 nodes are selected and $16 \times 3 = 48$ master DOFs were used for computation. As the results the displacements of these master DOFs are saved for review and further study. Since the accelerations are of more interests, the displacements for these master DOFs are differentiated twice to get the accelerations.
For the first computation damping effects are not included. The results are illustrated in Figure 9 - 11.

Without damping effects, there is almost no decay in the responses. Since the acceleration response is more interesting to us and it is illustrated together with the experimental data in Figure 12.

3.4 Calibration of the Elasticity Moduli and Damping

From the experimental data, the elasticity moduli and damping value can be estimated.

Consider the following equation

$$ M\ddot{x} + C\dot{x} + K = F(t) \quad (1) $$

Using Beta damping, one has

$$ M\ddot{x} + \beta K\dot{x} + K = F(t) \quad (2) $$

The natural frequency of the above system is
\[ \omega = \sqrt{K / M} \]

Then
\[ \omega_1 / \omega_2 = \sqrt{K_1 / K_2} \]

and \( K_2 \) can be estimated by
\[ K_2 = \frac{\omega_2^2}{\omega_1^2} K_1 \]

From Figure 13, \( \omega_2 \approx 2\pi \times 5000 \) and \( \omega_1 \approx 2\pi \times 1160 \), so one has
\[ K_2 = \frac{\omega_2^2}{\omega_1^2} K_1 = \left( \frac{5000}{1160} \right)^2 K_1 = 18.49K_1 \]

The elasticity modeuli of the three types of materials are computed as
\[ E_1 = 18.49E_{10} = 18.49 \times 1.0e11 = 1.849e12 \]
\[ E_2 = 18.49E_{20} = 18.49 \times 2.04e11 = 3.772e12 \]
\[ E_3 = 18.49E_{30} = 18.49 \times 3.846e11 = 1.849e12 \]

The damping value can be estimated as follows:
\[ e^{-\xi_1 t_1} / e^{-\xi_2 t_2} = 1 \]

Then
\[ \xi_2 = \xi_1 t_1 / t_2 \quad (3) \]

### 3.4 Results of Refined Model

With beta damping value of 3.63e-6, the results are shown in Figure 14.

From Figure 14, we have \( t_1 = 0.08 \), \( t_2 = 0.004 \) and \( \beta_1 = 3.63e-6 \), so
\[ \beta_2 = \beta_1 t_1 \omega_1 / (\omega_2 t_2) = 1.6884e - 5 \]

Due to the non-linearity and distribution of the vibration transmission, this calibration process needs to be repeated several times. After computation, a value of \( \beta_2 = 8.44e - 6 \) provides us a better result, which is shown in Figure 15. The power spectral density of this result together with the power spectral density of the experimental data is shown in Figure 16.

**Figure 14:** Result with damping value=3.63e-6

**Figure 15:** Result with calibrated parameters

**Figure 16:** PSD comparison of experimental data and model results with calibrated parameters.

### 3 SUMMARY
In this project the finite element model of gearbox is used to simulate the impulse response vibration. A simple method is introduced to calibrate the elasticity moduli and damping value to get a better simulation. Since several assumptions are made for simplicity, further study is needed to examine the effects of these simplifications. This study can be taken as the first step for a more detailed research.

REFERENCES


James, M. L., 1994, *Vibration of Mechanical and Structural Systems*, HarperCollins College Publisher.

Ian R. Grosse, 2002, *Transient analysis*, MIE605 handout, University of Massachusetts, Amherst.

ASYS manual script, 5.7.

MATLAB manual script, 5.3.