Structural Damage Localization using Optimization Method

W. Luber
Daimler-Benz Aerospace AG
Military Aircraft Division LMT24
P.O. Box 80-1160
D-81663 Munich, Germany

ABSTRACT. The structural defects on aircraft are not directly monitored and indicated like hydraulic failures. The identification of structural damage is important for safety flights, but inspecting airplanes at regular intervals is very costly.

A structural damage identification method is presented which examines the elastic and dynamic behavior of the structure by means of deflections stemming from discrete load cases and from normal mode shapes of the structure respectively as well as the information of the change of natural frequencies. The main objective of the study is to provide an economical and reliable damage detection method for aeronautical structures. An updated finite element model of the structure must be available. The reduced stiffness due to damage will be localized using non-linear mathematical optimization codes. Displacements for selected load cases and normal modes are taken as constraints. Minimum sizing changes with respect to the initial structure is used as objective function. Numerical examples with different structures show that the proposed methods can accurately detect the variations in stiffness in certain cases.

NOMENCLATURE

CAL: calculated
\( \mathbf{e} \): error vector
ERR: calculated error
FE: Finite Element
f(x): objective function
g(x): design constraints function
GRT: Ground Resonance Test
K: stiffness matrix
M: mass matrix
MAC: modal assurance criterion
r: response function
u: displacements of discrete load
x: design variable
\( \delta \): deviation

Indices and markings

A: analytical
i, j: current number
il: i-th component of load case 1
\( \lambda \): eigenvalue
\( \phi \): eigenvector
\( \Phi \): modal matrix

1. INTRODUCTION

This paper describes a procedure to detect and localize damage in elastic mechanical structures via analytical finite element model by means of modal test data and, if available, deflections of discrete load cases. The required modal test data are eigenfrequencies and mode shapes, derived from ground vibration test (GVT). A finite element model of the undamaged aircraft structure must be available. Before using this finite element model the dynamic model must be tuned that the analytical model is improved in and outside the measured frequency range. This objective is reached by a local update method applying an optimization code as described in Ref. 1 to 5. Design parameter as update variable can be any input parameter of an analytical model. The procedure updates mass and stiffness relevant properties, not damping properties, based on the calculation of the gradients (first order derivatives) of eigenfrequencies, mode shapes and deflections of discrete loads with respect to the design parameters.
Damage localization procedures like this one can handle incomplete test data and are successful even with a relative small number of test degrees of freedom. This procedure has been proven in many numerical examples. More detailed examples of the damage detection method are described in Ref. 10 to 12. Three of the numerical examples are given in this paper using analytical generated test data. The computer aided structural optimization code DASA-LAGRANGE was used to perform the applications shown here. This system is based on finite element methods as well as nonlinear mathematical programming codes. Design models are represented by their objective function, their design variables (parameters) and many different constraints (restrictions).

2. TECHNICAL APPROACH

The computer aided structural optimization system DASA-LAGRANGE (Ref. 8,9) was used to perform the applications shown. This system is based on finite element methods as well as nonlinear mathematical programming codes. Design models are represented by their objective function, their design variables (parameters) and many different constraints (restrictions). The general program architecture is shown in Fig. 1 and Ref. 9 explains the system in more detail.

2.1 Description of Optimization Method

The general formulation of structural optimization task can be stated as a nonlinear programming problem (NLP, see Ref. 7):

\[
f(x) = \min_{x \in \mathbb{R}^n} \quad \text{subject to } a_i(x) \leq 0, \quad j \in \mathbb{I} = \{1, \ldots, m\}
\]

The objective function \( I(x) \) is general the weight or volume of the structure which is linear with respect to the design variables. But it is also possible to use other objective functions for the updating with the aim of minimum changes of the FE model:

\[
f(x) = \| x^0 - x \| = \sqrt{\sum_{i=1}^{n} (x_i^0 - x_i)^2}
\]

Furthermore multiobjective problems can be handled by what is called vector optimization.

The design variables can be considered as three different types of structural parameters:

1. sizing variables like
   - cross sectional areas for truss and beam elements.
   - wall thickness for membrane and shell elements.
   - laminate thickness for all single layers in composite elements.

Design models are represented by their objective function, their design variables (parameters) and many different constraints (restrictions). Available design variables are element thickness, cross sections, fibre angles, and concentrated masses. To handle strength or stiffness designs, restrictions on stresses, strains, buckling loads, natural frequencies, dynamic modes, dynamic responses, aeroelastic efficiencies, flutter speed and generalized displacements can be taken into account as constraints. Besides isotropic, orthotropic and anisotropic materials the design of composite structures is of main concern. The primary aim is to minimize the structural weight subject to the selected restrictions. But sometimes the problem in question is to get a feasible design and the weight is less important. The extended range of mathematical programming codes, the modular architecture and the possibility to take all selected constraints simultaneously into consideration are the highlights of the program system.
2. concentrated masses to balance dynamic behavior.
3. angles of fibre directions for layered composite materials.

The most important part of structural optimization tools is the variety of restrictions that can be imposed on the design model. A typical formulation of the constraint functions is given by the following dimensionless inequality equation:

\[ g(x) = 1 - \frac{r(x)_{\text{act}}}{r_{\text{allow}}} \geq 0 \]

The satisfaction of constraints results in a feasible structure. In principle constraints have to be dealt with simultaneously as they restrict the design space in which the optimized structure will ultimately be found. Using gage constraints as preconditions for manufacturing are included. Which combination of constraints has to be applied depends on the physical problem statement.

The state variables i.e. response of the structure are implicitly dependent on the design variables \( x \).

### 2.2 Constraint’s

For the work performed here the DASA-LAGRANGE programme was modified as shown below. First of all it is worth to mention that the standard linear objective function for weight or volume cannot be used. This is obvious, because we are not searching for a structure with less weight, but for one which is close to real world. Therefore we make use of the linear objective function which reflects our intention to look for minor deviations that will not change the design variables drastically. The objective function is an appropriate target since it means to keep the distance from the initial structure \( x^0 \) as small as possible.

Also important is the set-up of imposed requirements on fitting of frequencies, dynamic modes and displacements by an adequate mathematical model. Although the physical models for displacements and dynamic responses are quite different, the formulation for limiting deviations of measured and calculated response \( r \) is the same:

Imposed structural response \( r(x) \) qualified by an interval \( r \in [r_i, r_u] \) with \( \text{sign}(r_i) = \text{sign}(r_u) \) can be written as

\[ r_i \leq r(x) \leq r_u \]

or equivalently

\[ r - \Delta r \leq r(x) \leq r + \Delta r \]

with \( \bar{r} = \frac{r_u + r_i}{2} \) and \( \Delta r = \frac{r_u - r_i}{2} \) which is the same as

\[ |r(x) - \bar{r}| \leq \Delta r \]

These are the following two constraints on the absolute values of the response.

\[ g_i(r) = \Delta r - (r(x) - \bar{r}) \geq 0 \quad \text{if} \quad r(x) \geq \bar{r} \]

\[ g_u(r) = \Delta r + (r(x) - \bar{r}) \geq 0 \quad \text{if} \quad r(x) < \bar{r} \]

To avoid zigzagging during the optimization process these two constraints will be combined to one constraint by multiplication with the same effect on the feasible domain. In addition the relative response will used to get rid of the dimensions and to stabilize the numerical process:

\[ g(x) = g_i \cdot g_u \cdot \frac{1}{\bar{r}} \geq \epsilon^2 - \left( \frac{r(x) - \bar{r}}{\bar{r}} - 1 \right)^2 \geq 0 \]

where \( \epsilon = \frac{\Delta r}{\bar{r}} \) is the fitting accuracy and \( \bar{r} \) is the measured separation to match.

In physical terms the imposed constraints are

- limited deviations of eigenvalues
  \[ |\lambda_j(x) - \lambda_j^M| \leq \epsilon \]
  where \( \lambda_j \) and \( \lambda_j^M \) are the calculated and measured j-th eigenvalues respectively

- limited deviations of eigenmodes
  \[ |\varphi_{ij}(x) - \varphi_{ij}^M| \leq \epsilon \]
  where \( \varphi_{ij} \) and \( \varphi_{ij}^M \) are the i-th components of mode j for calculated and measured values respectively

- limited deviations of displacements
  \[ |u_{ij}(x) - u_{ij}^M| \leq \epsilon \]
  where \( u_{ij} \) and \( u_{ij}^M \) are the i-th components of load case 1 for calculated and measured values respectively.
2.3 Sensitivity

Sensitivity analysis is necessary to localize the weakness of a structure. This method assumes that the eigenvalues are a function of stiffness, mass, damping and geometric data. The discrete eigenvalue problem associated with linear vibration is defined in terms of two symmetric matrices K and M. In general, the eigenvalue problem is guaranteed to have real eigenvalues if either K or M is positive definite and the discussion here is limited to this case. When the eigenvalues are distinct each derivative is given by (Ref. 8):

\[
\frac{\partial \lambda_i}{\partial \chi} = \Phi_i^T \left( \frac{\partial K}{\partial \chi} - \lambda_i \frac{\partial M}{\partial \chi} \right) \Phi_i.
\]

Eigenvalue sensitivities are useful when resonance frequencies need to be restricted. Exact analytical expressions for eigenvalue sensitivity can readily be derived for the case of non-repeated roots.

The eigenvector derivative is given by:

\[
\frac{\partial \Phi_i}{\partial \chi} = \Phi_i^T \frac{\partial M}{\partial \chi} \Phi_i + \sum_{j=1}^{n} \frac{1}{\lambda_j - \lambda_i} \Phi_j^T \left( \frac{\partial K}{\partial \chi} - \lambda_j \frac{\partial M}{\partial \chi} \right) \Phi_j.
\]

A second useful method is a combination between the dynamic force method and the above described method of eigenvalue sensitivity. The error vector is defined as the difference between the theoretical stiffness matrix and the mass matrix multiplied by the measured eigenmodes. The gradient is given by:

\[
\frac{\partial e_y}{\partial \chi} = \Phi^T \left( \frac{\partial K_A}{\partial \chi} - \frac{\partial M_A}{\partial \chi} \lambda \Phi \right) = 0.
\]

This is the sensitivity of the error matrix which shows the changes of the energy which is necessary if the analytical structure is forced to vibrate in the measured mode shape. For the purpose of damage localization it may be useful to evaluate the sum of all elements of the error sensitivity matrix.

\[
\delta_y = \sum_{i=0}^{n} \sum_{j=1}^{m} \left| \frac{\partial e_y}{\partial \chi} \right|
\]

2.4 Major Steps of the Damage Detection Procedure:

- Calculate modal data and displacements of the FE-model.
- Compare eigenvalues and eigenmodes with the ground vibration test.
- Compare displacements of discrete loads with the stiffness test results.
- Make a sensitivity analysis and try to localize the deviations. Choose design variables by selecting parameters relating to finite elements in the area where deviations of the structure is expected.

3 ANALYSIS A DEFECTED CANTILEVER BEAM

In order to validate the method, cantilever beams were used in the analyses and tests to prove that a finite element model can represent the dynamic behaviour of a test article. Three aluminium beams were fabricated to the dimensions shown in Fig. 3.

![Figure 2: FE-model of the test structure](image)

These beams were successively milled down at elements 1, 3 and 7 by 10%, 25% and 50% of their height and clamped as cantilever. In order to determine the sensitivities of these beams, vibration modes and frequencies corresponding to the simulated defects were calculated for each cantilevered beam. The expression simulated defects is used because the defect is introduced on one finite element of the beam and does not represent a real defect such as a crack whose stiffness would be non-linear due to the gap closing or opening. Damage was introduced symmetrically about the beam’s neutral axis in both the finite element model as well as in the test beam Fig. 4. The finite element model shown in Fig. 2 consists of 10 simple beam elements (in FE-notation called BAR) having constant cross sectional areas and constant inertia moments of area. Movement was allowed only in the y-z plane, see Fig. 3.

![Figure 3: Physical model of the test beam structure with a crack at element 3](image)
3.1 Comparison of test and analysis

Correlation between calculated and measured data are in good agreement as can be seen in Table 1 where mode frequencies are shown. Since there was good correlation it was decided to use the analytical beam results throughout for defect location. To represent experimental scatter on measured data, noise was introduced on the analytical mode shapes.

<table>
<thead>
<tr>
<th>Crack in percent</th>
<th>element</th>
<th>Mode</th>
<th>0%</th>
<th>10%</th>
<th>25%</th>
<th>50%</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>I</td>
<td>.97</td>
<td>.95</td>
<td>.96</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>II</td>
<td>.98</td>
<td>.99</td>
<td>.98</td>
<td>.99</td>
</tr>
<tr>
<td></td>
<td></td>
<td>III</td>
<td>.96</td>
<td>.99</td>
<td>1.0</td>
<td>.98</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>I</td>
<td>.97</td>
<td>.94</td>
<td>.95</td>
<td>.96</td>
</tr>
<tr>
<td></td>
<td></td>
<td>II</td>
<td>.98</td>
<td>.96</td>
<td>.95</td>
<td>.92</td>
</tr>
<tr>
<td></td>
<td></td>
<td>III</td>
<td>.96</td>
<td>.96</td>
<td>.95</td>
<td>.94</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>I</td>
<td>.97</td>
<td>.94</td>
<td>.94</td>
<td>.94</td>
</tr>
<tr>
<td></td>
<td></td>
<td>II</td>
<td>.98</td>
<td>.95</td>
<td>.93</td>
<td>.93</td>
</tr>
<tr>
<td></td>
<td></td>
<td>III</td>
<td>.96</td>
<td>.95</td>
<td>.94</td>
<td>.95</td>
</tr>
</tbody>
</table>

Table 1: Relative differences of eigenfrequencies $f_{\text{measured}} / f_{\text{analysis}}$ for Modes I, II, III

3.2 Detection and Localization of Defects

A finite element model of the test beam structure was created. No updating of this model was performed since correlation between measured and calculated vibration data was good as shown previously. For more complicated structures such an update may be required since this establishes the no defect situation. The vibration amplitudes of the defected beam in the three locations 4, 7 and 11 were used as constraints for the computer code as shown in Section 2.2. A sensitivity study was performed in order to establish the level of damage and at which beam location this could be detected with the computer code. The parameters investigated were the number of input modes and the fitting error between these modes and those which resulted from the updating, modal matching process, see dynamic constraints description in section 2.2. Before presenting the results it should be emphasized that the only design variable on a beam element is its cross section area i.e. $x = A$. No internal cross sectional parameters are considered as design variables. For fabrication reasons the test beam crack in Fig. 4 has been milled down in height only. In order to compare the measured response of the damaged structure with the calculated response by the DASA-LAGRANGE program system, the cross sectional properties of the test beam cracks have been used to get the simulated crack analysis response.

$$A(p) = p \cdot b \cdot h,$$
$$I_x(p) = \frac{b \cdot p^3 \cdot h^3}{12}$$

where $p$ = percentage of height.

Within DASA-LAGRANGE the cross sectional area depicted in Fig. 4 is used as design variable

$$x = \bar{A} = \bar{b} \cdot \bar{h} = ab \cdot ah$$

Thus the area moment of inertia is a quadratic function of $x$ because of assumed geometrical similar changes of cross sectional shape:

$$A(p) = p \cdot bh = (ab)(bh) \Rightarrow$$

$$\bar{b} = \sqrt{p \cdot b}, \quad \bar{h} = \sqrt{p \cdot h}$$

$$\bar{I}_x = \frac{\bar{b} \bar{h}^3}{12} = \frac{\bar{b}^2 \bar{h}^2 (h)}{12b} = \bar{A}^2 \cdot z = x^3 \cdot z$$

where $z = h/(12b)$ is a constant shape factor.

To identify the test beam crack due to the elastic behaviour in the $y$-$z$ plane the same area moments of inertia about the $x$-axis are needed.

$$\bar{I}_x(A) = I_x(p) \Rightarrow \bar{A}(p) = p^3 \cdot A$$

Thus we have a smaller cross sectional area to detect by the optimization code DASA-LAGRANGE:

<table>
<thead>
<tr>
<th>Test beam crack</th>
<th>LAGRANGE FE-crack</th>
</tr>
</thead>
<tbody>
<tr>
<td>10%</td>
<td>14.62% ≈ 15%</td>
</tr>
<tr>
<td>25%</td>
<td>35.05% ≈ 35%</td>
</tr>
<tr>
<td>50%</td>
<td>64.64% ≈ 65%</td>
</tr>
</tbody>
</table>

This weight represents the relative weight of a 1 test beam crack (15% LAGRANGE FE-crack) in element 1. The design process converges within 7 iterations.
In Fig. 5 the design variables which are the normalized areas of the beam elements are depicted as they change during the iterations. Element 1 is reduced to 88% of its initial area. All other areas are only changed slightly.

3.3 Sensitivity to Noise on the Input

In order to assess the sensitivity of the method to measuring errors in the input data a limited number of calculations were performed with simulated noise on these data. Results shows that it is possible to identify a 50% crack in element 1 with 14% variance on the amplitudes of points 4, 7 and 11. No additional information is required but the crack itself is not as accurately detected when there is noise on the input data. Results show that for a 50% crack in element 7 more information is required because the crack cannot be identified with the first two modes and frequencies when noise is present on input data. With the additional information from a third mode and corresponding frequency the damage can be detected.

In conclusion it can be stated that the method is still applicable with noisy input data but detection of cracks depends on their size, which has to be larger to be as successful as with noise free input data. Additional information, like fitting a further mode shape, can also assist in overcoming the noise problem.

4. CASE STUDY: Truss Structure

The second example is an application on the benchmark of the Group for Aeronautical Research in Europe (GARTEUR), because the structural model was available. The test structure is a plane clamped free system, fixed on the nodal points 5 (x and y single point constraint) and 6 (x single point constraint), whose members are characterised by $E = 7500$ Mpa, $I = 0.0756$ m$^4$, and $\rho = 2800$ kg/m, with $A_{\text{vertical elements}} = 0.006$ m$^2$, $A_{\text{diagonal elements}} = 0.003$ m$^2$ and $A_{\text{horizontal elements}} = 0.006$ m$^2$. The initial model contains 78 nodal points, and 83 bar elements resulting in 216 DOF, of which 12 (6x and 6y) translation DOF are assumed to be measured. Figure 6 shows the grid points and figure 7 the beam element numbers.

To simulate damage, the following modifications are introduced: Reduction of 50% of the cross sectional properties on the elements between nodal point 26 and 49. From earlier sensitivity analysis it was known, that the middle diagonal of the truss structure has negligible influence on the first mode shapes. This fact was used to prove the robustness of the method. To localize the damage, the first five eigenmodes were used with 12 displacements of each mode. The mode shapes were introduced with four digits without any interpolation. Two static load cases were introduced for improving the localization procedure on nodal point 73 in x and y direction. The first five eigenfrequencies of the undamaged structure were used as constraints with $\pm 0.5$ deviations. The eigenfrequency difference of the undamaged and damaged structure was less than 5%.

![Damage detection in % versus elements](image)
The results are shown in Fig. 8. This localization stage allows us to locate damage elements 170 to 176. The error is calculated on the elements between 55 and 75%. The reason for this overly predicted error margin can be explained by the above mentioned shape factor, because the simulated Lagrange programme error is always higher than the real physical damage. The elements on the connection points are also marked with some damage. This is also an indicator for more detailed analysis. The elements 171 and 175 shows not the right damage.

5. CASE STUDY: CFC FIN

Following a successful investigation on a metal beam also a natural progression was to examine a carbon fibre composite fin of a fighter type aircraft. Mode shapes are more complicated because torsion modes occur.

The overall geometry of the fin is given in Figure 9. The surface area is 5.46 m² and the leading edge sweep angle is 45°. The fin box has one shear pick up in the front and one bending attachment at the rear. The rudder actuator is connected with two rods for control actuation. Fin and rudder skins are built as carbon fibre laminates and connected by three hinges.

A finite element model of the fin and rudder structure was generated using the DASA LAGRANGE programme. This code can handle isotropic materials whose elastic properties are not direction dependent as well as anisotropy materials whose elastic properties are direction dependent.

The following types of elements were used:

- Rod pin jointed bar
- TRIA3 linear triangular plate
- QUAD4 linear rectangular plate
- ROD2 pin jointed bar
- TRIA6 linear triangular plate
- QUAD4 linear rectangular plate

Each main fin box skin element consists of a four layer CFC laminate. The fin tip is idealized as quasi-isotropic Glass Fibre Composite (GFC) which means that it contains equal amounts of glass fibre in the four direction angles 0°, +45°, -45°, +90°. The total number of elements used is 188. The number of nodes is 118 and the degree of freedom are 339 for equipment 146 lumped masses were added.

Two sets of modal data were calculated. The first set of mode shapes are for the basic model are described in Table 2. The first three modes can be identified as fin bending, rudder yaw and fin torsion. The two higher frequency modes are characterized by large fin tip motions. The second set of mode shapes were calculated instead of test data with a simulated damage of 50% reduction of the original numbers of the lower fin skin elements.

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Mode Description</th>
<th>CAL Hz</th>
<th>ERR Hz</th>
<th>GRT Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Fin Bending</td>
<td>11.2</td>
<td>9.8</td>
<td>10.6</td>
</tr>
<tr>
<td>2</td>
<td>Rudder Yaw</td>
<td>33.3</td>
<td>32.9</td>
<td>37.5</td>
</tr>
<tr>
<td>3</td>
<td>Fin Torsion</td>
<td>41.3</td>
<td>38.9</td>
<td>32.6</td>
</tr>
<tr>
<td>4</td>
<td>Tip Torsion 1</td>
<td>56.4</td>
<td>47.3</td>
<td>48.7</td>
</tr>
<tr>
<td>5</td>
<td>Tip Torsion II</td>
<td>66.9</td>
<td>66.4</td>
<td>172.8</td>
</tr>
</tbody>
</table>

TABLE 2: COMPARISON OF THE MODAL FREQUENCIES

The modal frequencies for this case are compared with the frequencies of the basic model and the frequencies derived from ground resonance test in Table 2. Considering the differences of the aircraft GRT and clamped FE calculation, it can be concluded that the analytical model predicts the dynamic behavior of these aircraft structure well.
the sensitivity of the method to measuring errors in the input data, a number of calculations were performed with simulated noise input. In general it can be stated that the method is still applicable with noisy input but detection of errors depends on their size which has to be larger to be successful as with noise free input data. After several attempts using vibration mode frequencies only and amplitudes at few points, it was found that accurate damage localization is achieved if the modes are constrained, required an error of 1 % on all nodal point amplitudes. This is shown in figure 10.

6. CONCLUSION

A local method to detected damage at dynamic finite element model by updating design variables has been described. Emphasis was on the development of a procedure which takes into account the modal data as well as the deflections of discrete static load cases. The numerical examples showed that the optimization procedure works even in the presence of noisy test data. In this case eigenfrequency, eigenmode differences in the objective function should be chosen with respect to the reliability and scatter of the measured input data. It could be demonstrated that the design variables can be corrected on the basis of a restricted number of eigenfrequencies and corresponding modes as well as restricted static test data as displacement. A significant improvement of the eigenfrequencies and mode shapes outside the measured range can be achieved. This means also incomplete test data can be handled successfully.

7. REFERENCES


