S. Shkarayev, A. Raman, and A. Tessler: Computational and Experimental Validation Enabling a Viable in-Flight Structural Health Monitoring Technology, Proc. First European Workshop on Structural Health Monitoring, Cachan, Paris, France, 2002 (https://ifem.larc.nasa.gov/publications.html).

Proceedings of the First European Workshop

STRUCTURAL HEALTH MONITORING 2002

Edited by

Daniel L. Balageas

Department of Structures and Damage Mechanics
French National Aerospace Research Establishment
(ONERA)
Châtillon, France

Proceedings of the First European Workshop on
Structural Health Monitoring held at the
Ecole Normale Supérieure, Cachan (Paris), France
July 10-12, 2002

DEStech Publications

Computational and Experimental Validation Enabling a Viable In-Flight Structural Health Monitoring Technology

S. SHKARAYEV, A. RAMAN and A. TESSLER

ABSTRACT

An important and challenging technology aimed at the next generation of aerospace vehicles is that of structural health monitoring. The key problem is to determine accurately, reliably, and in real time the applied loads, stresses, and displacements experienced in flight, with such data establishing an information database for structural health monitoring.

The present effort is aimed at developing a finite element-based methodology involving an inverse formulation that employs measured surface strains to recover the applied loads, stresses, and displacements in an aerospace vehicle in real time. The inverse interpolation formulation is based on a parametric approximation of the loading and is further constructed through a least-squares minimization of calculated and measured strains. This procedure results in the governing system of linear algebraic equations, providing the unknown coefficients that accurately define the load approximation.

The computational and experimental validation of the methodology is carried out for a wing box-type structure. The structural component used in this study is the left half stabilizer of a Grumman C-1A Trader aircraft. The stabilizer represents a typical semi-monocoque structure. The computational procedure uses a standard finite element model of an airframe, with subsequent application of the inverse interpolation approach. Accuracy of the method is discussed in detail.

The structural testing of the stabilizer is also discussed. Strain gages are applied on both the lower and upper surfaces. The magnitudes of the applied forces are monitored via a load cell. The computational results are correlated and quantified using the test data.

INTRODUCTION

The development of advanced structural health monitoring is extremely important to the future progress of aeronautical and space systems. Recent achievements in the design of health monitoring systems for aerospace structures are highlighted in [1]. This technology prevents catastrophic degradation and excessive deformations of airframes and is comprised of three facets: (a) determination of stresses and deformations of structural components (b) identification of external loads, and (c) detection of critical damage mechanisms such as cracking, delamination, and corrosion. The development of such information technology involves multidisciplinary research in the areas of computational mechanics, intelligent information systems, and sensor networks.

The information system for structural health monitoring includes sensors, data acquisition devices, computers, software, and a database. A schematic of the information system is presented in Figure 1.

The main objective of this effort is an experimental validation of a recently proposed computational approach to the solution of an *inverse problem* by Shkarayev et al. [2]. This computational approach is based upon an *inverse interpolation* concept coupled with a parametric approximation of the loading and a least-squares minimization of the calculated and measured strains. It was shown in [2] that accurate reconstruction of the structural response is possible for problems involving various levels of structural approximation. The next logical step would be an experimental validation of the methodology by way of testing of a real structural component.

In the present study, structural tests were conducted for the wing box-type structure. The structural component selected is the left half stabilizer of a Grumman C-1A Trader airplane. Strain gages were applied on both the lower and upper surfaces with the applied loads monitored via a load cell. The numerical results using the approach of [2] were correlated and quantified by means of the test data. Further comparisons were also demonstrated using results of the finite element analyses. It is further noted that ground structural testing allows working out of appropriate procedures and recommendations for the in-flight measurement strategy.

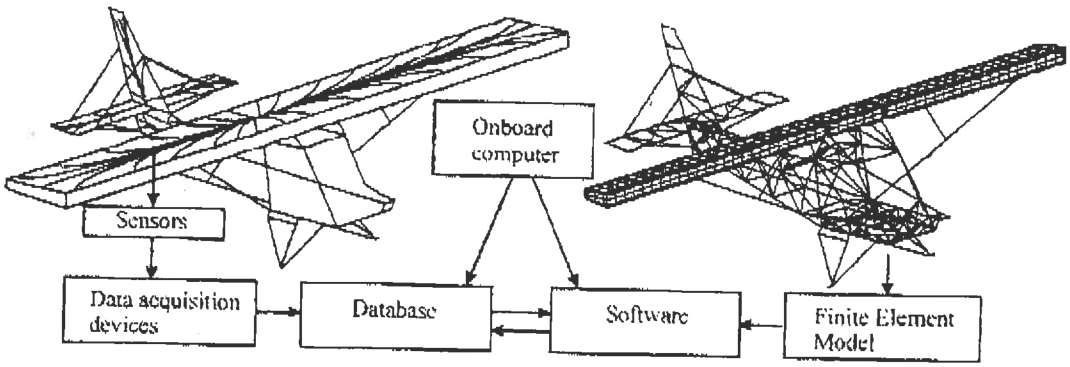

Figure 1. Onboard structural health monitoring system.

INVERSE INTERPOLATION METHOD

The problem under consideration is the structural response monitoring of an aircraft during its flight. As the aircraft performs a maneuver or is subject to atmospheric turbulence, the changes in the external loading result in changes in the linear and angular displacements, strains, and stresses throughout the airframe. To measure and monitor these changes, strain sensors are embedded along specified patterns, and the strain measurements ε^* are stored in an onboard computer. The strain data are used as input to the present computational scheme for which a finite element model of the airframe is first developed.

Consider that the external loads are applied to the airframe incrementally. Any changes in the loads are assumed to be small. At each increment or step of loading, the linear equilibrium equations at the structure level of approximation have the form

$$\mathbf{K}\mathbf{u} = \mathbf{p} \tag{1}$$

where $\mathbf{K}$ is a stiffness matrix, $\mathbf{u}$ is a nodal displacements vector, and $\mathbf{p}$ is a vector of equivalent nodal forces at the current load increment.

The averaged nodal strains can be expressed in terms of nodal displacements as

$$\boldsymbol{\varepsilon} = \mathbf{B}\mathbf{u} \tag{2}$$

where $\mathbf{B}$ is the strain-displacement matrix encompassing all element contributions.

Given a surface load, $p(s)$, the nodal forces are readily expressed as

$$\mathbf{p} = \sum_{e=1}^{N_{el}} \int_s \mathbf{n}_e^T p(s) ds \tag{3}$$

where $\mathbf{n}_e$ is a row-vector of displacement shape function of the e^{th} finite element, and N_{el} is the total number of elements. In Eq. (3) the integration is performed over the surface where the load is applied; the summation represents the well-known procedure of finite element matrix assembly.

The inverse interpolation approach is formulated as follows. The surface load is approximated by a linear combination of spatial distribution functions, $R_j(s)$, as

$$F(s) = \sum_{j=1}^{J} a_j R_j(s) \tag{4}$$

where a_j are unknown loading parameters.

The approach proceeds with a direct finite element analysis performed for each j^{th} load, $R_j(s)$. Using approximation (4), the displacements, $\mathbf{u}$, and strains, $\boldsymbol{\varepsilon}$, corresponding to this load are determined from Eqs. (1) and (2) as

$$\mathbf{u} = \sum_{j=1}^{J} a_j \mathbf{K}^{-1} \sum_{e=1}^{N_{el}} \int_s \mathbf{n}_e^T R_j(s) ds \tag{5}$$

$$\varepsilon = \sum_{j=1}^{J} a_j \mathbf{B}\mathbf{K}^{-1} \sum_{e=1}^{N_{el}} \int_{s} \mathbf{n}_e^T R_j(s) ds \tag{6}$$

The a_j $(j = 1, J)$ parameters are determined by minimizing the following least-squares functional with respect to these parameters

$$\text{Minimize:} \quad S = \left\{\varepsilon - \varepsilon^*\right\}^T \left\{\varepsilon - \varepsilon^*\right\} \tag{7}$$

where the parameterized strains, ε, are defined in (6). Equation (7) results in a governing system of linear algebraic equations that is readily solved for a_j $(j = 1, J)$. The applied loads are then calculated from Eq. (4), with the displacements and strains computed from Eqs. (5-6). The stresses are then computed from Hooke's constitutive relations.

COMPUTATIONAL MODELING

The present approach is validated using a two-step procedure. A finite element model is first developed for a given linearly elastic structure under the applied load $p(s)$. Surface strains ε^* are then computed at n specified locations, with these quantities representing the "measured" experimental strains. In the second step of the analysis, an inverse method is employed that uses these measured strains, ε^*, and the parametric approximation of the load, $F(s)$, as given in Eq. (4). The strains ε are computed from the loads represented by the $R_j(s)$ functions. The parametric approximation of the load is obtained via a least-squares minimization method. Then, the applied load and structural response are recovered. The finite element analysis is performed using "SHELL 63" shell elements of ANSYS [3].

The left horizontal stabilizer of a Grumman C-1A Trader airplane was utilized in the study. The length of the stabilizer is 2.7 m, with a maximal chord of 1.07 m and maximal thickness of 0.21 m. A schematic of the structure, along with the locations of the strain gages and loading are illustrated in Figures 2 and 3.

In the numerical simulation, the loading function is assumed in the form $p(x,z) = b_1\sqrt{l^2 - z^2}$ $(b_1 = 0.5)$ and applied normal to the upper surface of the stabilizer. There are twelve strain-gage locations $(n = 12)$ assumed in the determination of ε_e^*. The load approximation function is selected as $F(x,z) = a_1\sqrt{l^2 - z^2}$.

In order to study the effect of measurement errors on the accuracy of the method, the "measured" strains are assumed in the form $\tilde{\varepsilon}_e = \varepsilon_e^* \Delta^*$. The components of the row-vector of relative errors Δ^* are given as $\Delta_i = 1 + 0.07\,\delta_i$, where δ_i is a random variable having a standard normal distribution. Based on ten sets of random numbers, the recovered coefficient a_1 is in the range of 0.480-0.513. These values agree well with the actual load factor $b_1 = 0.5$.

EXPERIMENTAL VERIFICATION

In carrying out the current project, the equipment in the Aircraft Structures Laboratory of the Department of Aerospace and Mechanical Engineering at the University of Arizona was utilized. The Laboratory houses a wing loading facility to position and support the structure and loading and measuring systems (Figure 3).

The stabilizer was cantilevered to a vertical stand capable of withstanding all components of forces and moments. The loading frame is adjustable in order to realize all combinations of bending, shear, and torsion components in different loading zones. Two loading templates were installed over the Rib 3 and Rib 5 that allow transmission of forces to the structure from the loading tree. They are connected to the loading tree, which in turn is connected to a hydraulic cylinder. The maximum applied force is 8.9 kN, which was transmitted to the uniform line distributed load $p(x,z) = b_1 =$ 10.1 kN/m over the contact lines between loading templates and the stabilizer.

Strain gages were attached to both the upper and lower surface of the stabilizer. A total of 12 gages were applied to the structure (10 on the top and 2 on the bottom). The strain gages manufactured by Measurements Group, Inc, were both the 90° rosette type and the uniaxial type. Both types are fabricated out of constantan foil with a flexible polyimide backing. The rosettes have an active gage length of 12.5 mm, whereas the uniaxial gages have an active length of 25.0 mm. Both gage types have a nominal

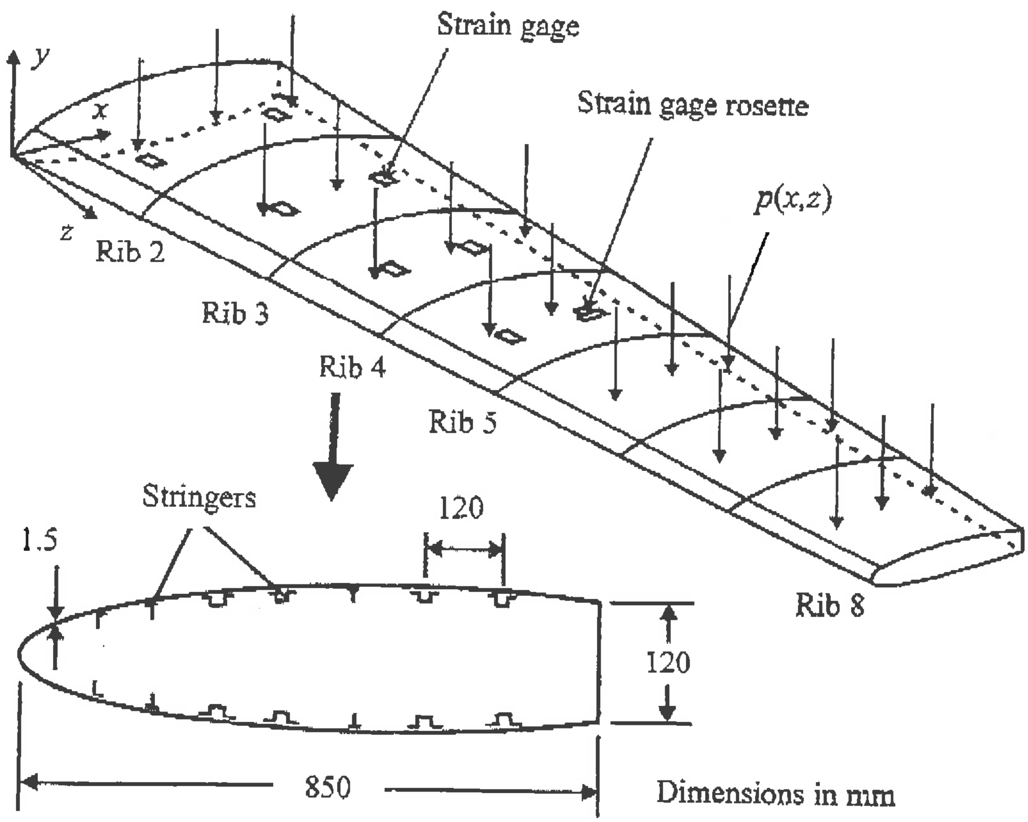

Figure 2. Geometry and loading of the structure.

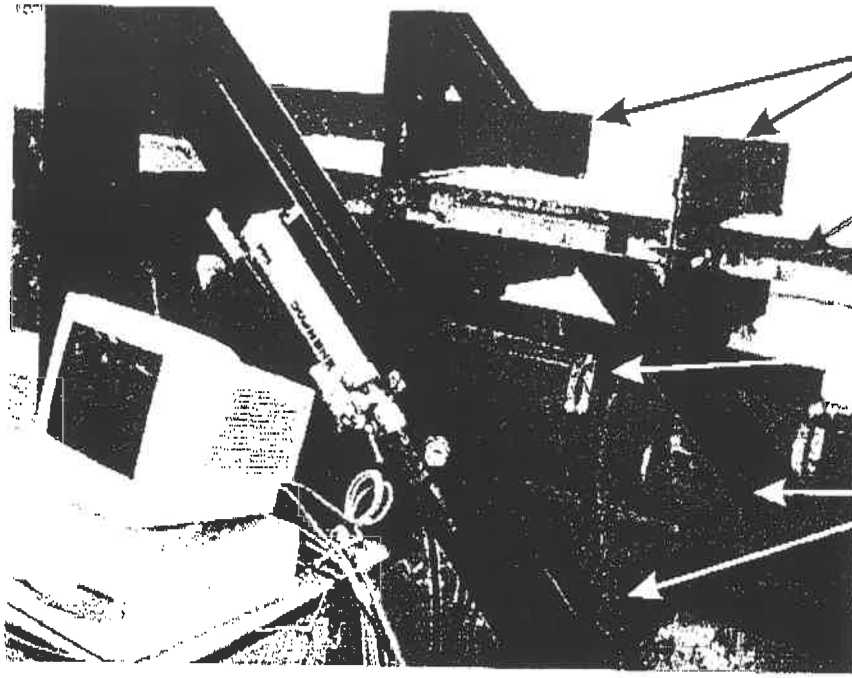

Figure 3. Wing testing facility.

resistance of 120 Ohms. The uniaxial gages were attached closer to the leading and trailing edge with the rosettes being towards the center of the structure. The vertical displacement of 98 mm at the center of the Rib 8 was measured by means of a meter rod.

The experimental strains data ε_z^* was utilized in the inverse interpolation method in order to recover the actual load and displacements. For this purpose, the load approximation was selected as $F(x,z) = a_1$ defined over the contact lines between the templates and the stabilizer. The finite element model used a unit line distributed load over Rib 3 and Rib 5 locations. The load was applied in the negative Cartesian *y*-axis direction at each node that comprises the interface of the template with the upper surface of the stabilizer. The calculated load factor, a_1 = 9.9 kN/m, indicates a favorable comparison against the actual value. The corresponding vertical displacement of 92 mm obtained at the center of Rib 8 agrees very well with the experimental measurement.

REFERENCES

1. Noor, A. K., S. L. Venneri, D. B. Paul, and M. A. Hopkins. 2000. "Structures Technology for Future Aerospace Systems," *Comput. and Struct.*, 74:507-519.
2. Shkarayev, S., R. Krashanitsa, and A. Tessler. 2001. "An Inverse Interpolation Method Utilizing In-Flight Strain Measurements for Determining Loads and Structural Response of Aerospace Vehicles," *Proceedings of 3rd International Workshop on Structural Health Monitoring*, 336-343.
3. *ANSYS User's Manual*, Vol. I. 1992. Canonsburg, PA: Swanson Analysis Systems, Inc.