Piezoelectric Actuator Modeling Using MSC/NASTRAN and MATLAB

Mercedes C. Reaves and Lucas G. Horta
Langley Research Center, Hampton, Virginia

October 2003
Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:


- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA STI Help Desk at (301) 621-0134

- Phone the NASA STI Help Desk at (301) 621-0390

- Write to:
  NASA STI Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076-1320
The use of trademarks or names of manufacturers in the report is for accurate reporting and does not constitute an official endorsement, either expressed or implied, of such products or manufacturers by the National Aeronautics and Space Administration.

Available from:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000
ABSTRACT

This paper presents a procedure for modeling structures containing piezoelectric actuators using MSC/NASTRAN and MATLAB. The paper describes the utility and functionality of one set of validated modeling tools. The tools described herein use MSC/NASTRAN to model the structure with piezoelectric actuators and a thermally induced strain to model straining of the actuators due to an applied voltage field. MATLAB scripts are used to assemble the dynamic equations and to generate frequency response functions. The application of these tools is discussed using a cantilever aluminum beam with a surface mounted piezoelectric actuator as a sample problem. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the example problem are provided. Analysis results are generated in terms of frequency response functions from deflection and strain data as a function of input voltage to the actuator.

INTRODUCTION

NASA Langley Research Center, Industry, and Academia have been actively studying and developing induced strain actuation devices for aircraft and aerospace applications since the late 1980's. Strain actuation produced as a result of a thermal load and piezoelectricity involves straining of components due to temperature changes and voltage changes. Piezoelectric materials such as Lead Zirconate Titanate (PZT) ceramics when subjected to an electric field produce mechanical strain or alternately generate an electric charge when subjected to a mechanical strain. This property gives piezoelectric materials the ability to act as actuators or sensors. Using piezoelectric actuators and sensors to form self-controlling and self-monitoring systems to improve performance of aircraft and space structures has attracted interest in the research community.

Numerous studies have been completed on analyses and models for piezo-electrically controlled structures. Some of these studies include: a high-order theory to model composite laminates with surface bonded or embedded piezoelectric sensors by Seely; a three-dimensional finite element code to analyze the mechanical-electrical response of laminated composites containing distributed piezoelectric ceramics developed by Sung Kyu Ha; the use of classical laminate theory to estimate the through-the-thickness strain distribution of composite laminates with embedded actuators by Crawley and others. Although those analytical techniques showed good correlation with experimental data, few have been incorporated into commercially available codes for general use.

Due to the increased interest in the design of complex structures with piezoelectric actuators and the need for fast and simple implementation of piezoelectric control systems, technology developers are beginning to incorporate or provide the tools to create piezoelectric elements. Freed developed one and two-dimensional finite elements, which include piezoelectric coupling, for integration into MSC/NASTRAN, however, this option is not currently available. Hauch investigated using ABAQUS electromechanical-coupled finite elements and superelement capabilities for modeling structures with piezoelectric actuators. These advances in the modeling capabilities of
piezoelectric actuators have allowed a number of viable analytical and numerical tools. Reaves\textsuperscript{11} developed benchmark test article structures to validate techniques for modeling structures containing piezoelectric actuators using commercially available Finite Element Analysis (FEA) packages. This document presents detail information on the implementation of one approach described in reference\textsuperscript{11}. The use of these tools is presented using a cantilever aluminum beam with surface mounted piezoelectric actuators as a sample case.

The method described herein use MSC/NASTRAN to model the structure with piezoelectric actuators and a thermally induced strain to model expansion/contraction of the actuators due to an applied voltage field. To reduce the number of structural modes needed for an accurate solution, Ritz vectors\textsuperscript{12} are appended to the structural modes. MATLAB scripts are used to assemble the dynamic equations and to generate frequency response function at the locations of interest. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the case history example are provided. Frequency response functions from deflection and strain responses to applied input voltage are used for evaluation of the methodology.

**PROBLEM FORMULATION**

Modeling of a structure with piezoelectric actuators using finite elements can be performed like any conventional finite element model formulation. However, to include piezo effects, elements that expand or contract as voltage is applied, the thermal strain capability within the element formulation needs to be used to evaluate voltage induced internal forces. The governing equation of motion is written as

\[ M\ddot{x} + Kx = f_p \]  

(1)

where \( f_p \) represents internal loads due to actuation, \( M \) is the mass matrix, \( K \) is the stiffness matrix, and \( x \) contains node displacements. Induced forces are computed by first defining an artificial coefficient of thermal expansion \( \alpha = d_{31} / t \) using the piezoelectric strain to voltage parameter \( d_{31} \) and the piezo material thickness \( t \). The second step is to apply a unit change in temperature (or voltage) in NASTRAN to evaluate the resulting deformation \( T_r \). This deformation vector \( T_r \) is referred to as a Ritz vector, one Ritz vector is required for each independently actuated piezo. Internal loads are now related to displacements and voltages by \( f_p = KT_r \Delta v \). In the reduced order model formulation, mass normalized eigenvectors \( \Phi \) from an eigenvalue solution of Eq. (1) and a set Ritz vectors \( T_r \) can be used as a set of basis vectors for model reduction. With these basis vectors a new coordinate transformation can be defined

\[ x = [\Phi \ T_r] \begin{bmatrix} z_r \end{bmatrix} \]

(2)

to transform Eq. (1) to
where $\Lambda$ is a diagonal matrix containing the eigenvalues to the homogeneous form of Eq. (1), $I$ is an identity matrix, and all the other matrices are partitioned consistent with the number of basis vectors used. For example, using $m$ eigenvalues and $r$ piezoelectric actuators, the transformation matrix in Eq. (2) has $m+r$ columns and $n$ rows, where $n$ is the number of degrees-of-freedom in the model. Similarly the transformed displacement vector $z$ has $m$ and $z_r$ has $r$ rows. Often, Eq. (3) is sufficient to conduct simulation studies, however, in this form the equation corresponds to a stiff system of differential equations because the transformed mass matrix is ill conditioned. Ill conditioning in this particular case is due to negligible mass values associated with the Ritz basis vectors. To overcome this problem, one can decompose the mass matrix using singular value decomposition such that

\[
\begin{bmatrix}
I & M_r \\
M_r^T & M_r
\end{bmatrix}
= U \begin{bmatrix}
\sigma & \\
0
\end{bmatrix} V^T
\]

where $U$ contains the left singular vectors, $V$ contains the right singular vectors, and $\sigma$ are nonzero singular values. In the numerical implementation of the procedure it is not likely that some of the singular values will be identically equal to zero but values below a certain threshold are considered zero. Furthermore, the number of singular values with numerical values near zero equals the number of piezoelectric actuators placed in the model. Using the right singular vectors, a new transformation matrix can be defined as

\[
\begin{bmatrix}
z \\
z_r
\end{bmatrix} = V \begin{bmatrix}
\eta \\
\eta_r
\end{bmatrix}
\]

Substituting Eq. (5) into Eq. (3) yields a transformed set of equations

\[
\begin{bmatrix}
\sigma & \\
0
\end{bmatrix} \begin{bmatrix}
\dot{\eta} \\
\dot{\eta}_r
\end{bmatrix} + \begin{bmatrix}
k & k_r \\
k^T_r & k_r
\end{bmatrix} \begin{bmatrix}
\eta \\
\eta_r
\end{bmatrix} = \begin{bmatrix}
b \\
b_r
\end{bmatrix} \Delta \nu
\]

where the bottom set of equations correspond to an algebraic set of equations. Separating the algebraic equation from the differential equations, the resulting properly conditioned set of matrix differential equation is

\[
\sigma \dot{\eta} + (k - k_r k_r^{-1} k_r^T) \eta = (b - k_r k_r^{-1} b_r) \Delta \nu
\]

Time integration of Eq. (7) provides values for $\eta$ and the solution for $\eta_r$ is

\[
\eta_r = -k_r^{-1} k_r \eta + k_r^{-1} b_r \Delta \nu
\]
To recover the response in physical coordinates, results from Eqs. (7) and (8) are substituted into Eqs. (5) and (2). For MATLAB implementation all the equations are transformed into state space form and then integrated using the MATLAB built-in integration software.

**EXAMPLE PROBLEM**

A cantilevered aluminum beam 2.75” x 16” x .04”, Figure 1, with one piezoelectric actuator bonded near the root is used in this study. A Flex-Patch piezoelectric actuator, developed and fabricated at NASA LaRC, is selected for the application. The Flex-Patch consists of a 3”x1.75”x.008” Morgan Matrox PZT-5A piezoceramic encapsulated using a polymer film. The PZT-5A piezoceramic mechanical and electro-mechanical properties are listed in Table 1. MSC/NASTRAN is used to model the structure; Figure 2 shows a finite element representation of the cantilevered beam indicating the location of the piezoelectric actuator and strain sensor output used in the present study. The model contains 396 quadrilateral elements and 450 nodes. PCOMP cards in MSC/NASTRAN are used to specify the properties of the composite lay-up for the actuator and aluminum substrate.

**ANALYTICAL TOOLS IMPLEMENTATION**

The procedure to generate the analytical model of the cantilever beam involves the following steps: 1) construction of a FEM based on the test article and Ritz vectors generation, 2) eigenvectors generation and mass and stiffness matrices calculation to assemble the reduced order model, 3) execution of MATLAB scripts to assemble the dynamic equation and to generate frequency response function at the locations of interest. These steps are described in more detail in the following sections.

**Structural FEM and static Ritz vector generation** - MSC/NASTRAN offers no piezoelectric coupled-field elements capability from which to model smart structures directly. Rather, the analogy between piezoelectric strain and thermally induced strain, is used to allow temperature changes to model piezoelectric voltage actuation. Piezoelectric coefficients characterizing the actuator are input as thermal expansion coefficients (CTE’s) associated with standard elements. In this study the model treats both the actuator and the structure substrates as plies of an integrated laminated plate. PCOMP cards in MSC/NASTRAN are used to specify the properties of the composite lay-up and the applied voltage is modeled as a thermal load. TEMP cards identify the locations at which thermal loads (voltage) is applied. MAT1 cards representing the actuators include non-zero thermal expansion coefficients. The thermal expansion coefficients in the remaining structure must be zero to ensure thermal expansion loads are generated only at the piezoelectric actuator locations. Appendix 1 shows a listing of the MSC/NASTRAN input data deck ‘almq2.dat’, for the cantilevered beam. For brevity, MSC/NASTRAN grid points and element information are not included in the listing. The execution of this file will generate the Ritz vector (static deflection and/or strain data for the beam) using a thermal load equivalent to 1 volt. The DMAP sequence ‘static.dmap’
incorporated in the MSC/NASTRAN input data deck stores the static solution vector in a
binary formatted output file 'S101SV1.OUT'. In applications with multiple actuators
each actuator load and output request are to be included in individual SUBCASES.

**Eigenvectors and reduced model generation** - After computing Ritz vectors, a general
eigenvalue/eigenvector MSC/NASTRAN solution containing special structural matrix
transformation routines (DMAP) combine the Ritz vectors with eigenvectors and
calculates the mass and stiffness matrices in Eq. (3) needed to assemble a reduced order
model. NASTRAN commands for the cantilever beam example are listed in the file
named 'almq2e.dat' and the DMAP sequence is listed in 'genkm_v2.dmap' (Appendix
2). The 'almq2e.dat' file uses the file 'S101SV1.OUT' as input and generates two
ASCII formatted output files: 'almq2e.asc' and 'almq2e.pch' containing the reduced
mass/stiffness matrix and the displacements and/or strains for the structural modes
including the Ritz vectors, respectively. Output observation points are for displacements
at nodes 348 and 349 and strain for element 116. To facilitate importing the matrices to a
MATLAB environment the file 'almq2e.asc' needs to be edited to remove lines starting
with the one containing "eigensum" up to but not including the line containing "genk".
The first line in the file should contain 'genk' and must not have blank lines in the
beginning of the file.

**Dynamic equation assembly and results generation** - MATLAB scripts are required to
assemble the dynamic equations and generate frequency response functions at the
observation points. The MATLAB script 'makemodel.m' used in the cantilever beam
case study is listed in Appendix 3. The script creates space state model matrices based on
the number of inputs (number of actuators) and MSC/NASTRAN output request (element
strain and/or nodal displacement). The script uses as input the files generated and edited
in the previous step: 'almq2e.asc' and 'almq2e.pch'. The MATLAB scripts are problem
specific depending on the type of output requests, but are easy to modify.

A flow chart summarizing the complete procedure to solve the cantilevered aluminum
beam problem is shown in Figure 3.

**EXAMPLE PROBLEM RESULTS**

The frequency response functions for input voltage, tip displacement, and strain gage data
are generated from analysis. NASTRAN analysis results using the modeling technique
described herein are shown in Figure 4 for the first four beam/actuator mode shape
deformations with corresponding frequencies of 5.68, 33.59, 60.24 and 91.21 Hz. To
examine the input/output relationship of the system with the actuator, Figure 5 shows a
comparison of the frequency response function magnitude (top) and phase (bottom) from
the beam tip displacement to the piezo-actuator input voltage. Furthermore, Figure 6
shows results from the nearly collocated strain gage.
SUMMARY

A procedure for modeling structures containing piezoelectric actuators using MSC/NASTRAN and MATLAB is presented. The procedure describing how to use these tools is presented using a cantilever aluminum beam with surface mounted piezoelectric actuators as example problem. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the example problem are provided. Analysis results are generated in terms of frequency response functions from deflection and strain data to input voltage to the actuator.

REFERENCES


Table 1. Material properties of PZT-5A piezoceramic and T300/976 graphite/epoxy composite

<table>
<thead>
<tr>
<th></th>
<th>PZT-5A</th>
<th>T300/976</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of elasticity</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(lb/in²)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E₁</td>
<td>1.0E+7</td>
<td>2.17E+7</td>
</tr>
<tr>
<td>E₂</td>
<td></td>
<td>1.305E+6</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
<td>0.3</td>
</tr>
<tr>
<td>Shear Modulus (lb/in²)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G₁₂</td>
<td>3.82E+6</td>
<td>1.03E+6</td>
</tr>
<tr>
<td>G₁₈</td>
<td>1.03E+6</td>
<td></td>
</tr>
<tr>
<td>G₂₂</td>
<td>363600</td>
<td></td>
</tr>
<tr>
<td>Density, (lb·sec²/in⁴)</td>
<td>7.16E-4</td>
<td>1.49E-4</td>
</tr>
<tr>
<td>Piezoelectric constant, (in/Volt)</td>
<td>6.73E-9</td>
<td></td>
</tr>
<tr>
<td>d₃₁</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Electrical permittivity, (farads/in)</td>
<td>4.2E-10</td>
<td></td>
</tr>
<tr>
<td>ξ₈</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Figure 1. Cantilevered aluminum beam with piezoelectric actuator.

Figure 2. Aluminum beam with piezoelectric actuator finite element mesh.

E116 – strain sensor output location
Nodes 348, 349 – displacement output location
Create Finite Element Model of test structure

Run MSC/NASTRAN static analysis almq2.dat with DMAP sequence static.dmap

Output file: S101SV1.OUT4
Static Ritz vectors

Run MSC/NASTRAN eigenvalue analysis almq2e.dat with DMAP sequence genkm_v2.dmap

Output files: almq2e.asc, almq2e.pch
- Merged Ritz and vibration modes vectors
- Reduced M and K matrices

Edit almq2e.asc output file to remove eigenvalue data not needed

Run MATLAB script makemodel.m
Creates space state model

Figure 3. Steps to generate analytical model of aluminum beam with piezoelectric actuator.
Mode 1
@ 5.68 Hz

Mode 2
@ 33.59 Hz

Mode 3
@ 60.24 Hz

Mode 4
@ 91.2 Hz

Figure 4. Mode Shapes of aluminum beam with piezoelectric actuator.

Figure 5. FRF of aluminum beam tip displacement as a function of piezoelectric actuator input voltage.
Figure 6. FRF of aluminum beam strain as a function of piezoelectric actuator input voltage.
Appendix 1

The following NASTRAN input data deck listing is contained in the almq2.dat file. The deck is set up to generate the Ritz vector corresponding to a single piezoactuator mounted on the structure. TEMP cards identify the locations at which thermal loads (voltage) is applied. The DMAP static.dmap incorporated in the data deck by the 'include' command stores static solution vectors in the output file 'S101SV1.OUT4' specified in the ASSIGN statement.

almq2.dat data deck:

ASSIGN OUTPUT4='S101SV1.OUT4' UNIT=15 DELETE
INIT MASTER(S)
ID PIEZOELECTRIC QUAD ELEMENTS
SOL 101
$
compile sestatic,souin=mscsou,nolist,noref$
include 'static.dmap'
$
  TIME = 60
CEND
TITLE = NASTRAN FILE TRANSLATOR -- UNITS = SI
$ GLOBAL CASE
  SPC = 1
  TEMP = 1
$ Define set of elements for strain output
SET 4 =116
$ Define set of nodes for displacement output
SET 5 =348,349
DISPLACEMENT(PRINT)=5
STRAIN(PRINT,FIBER)=4
$
BEGIN BULK
$
PARAM AUTOSPC YES
PARAM POST -2
PARAM,K6ROT,100.
GRID 1 0 0 7.14E-3 5.75E-2 0.00000 0
GRID 2 0 0 7.14E-3 4.87E-2 0.00000 0
GRID 3 0 0 7.14E-3 3.98E-2 0.00000 0
.
.
CQUAD4 396 2 552 513 519 555-90.0000
$ Property card definition for aluminum beam substrate
PCOMP 2-5.08E-4 0.00000 0.00000 0.00000 +
+ 2 1.27E-5 0.000000YES 2 9.91E-4 0.000000YES +
+ 2 1.27E-5 0.000000YES

$ Property card definition for aluminum beam substrate and piezoactuator lay-up
PCOMP 5-5.08E-4 0.00000 0.00000 0.00000 +
+ 2 1.27E-5 0.000000YES 2 9.91E-4 0.000000YES +
+ 2 1.27E-5 0.000000YES 3 1.27E-5 0.000000YES +
+ 3 2.54E-5 0.000000YES 4 2.03E-4 0.000000YES +
+ 3 2.54E-5 0.000000YES 3 1.27E-5 0.000000YES

$ Aluminum material properties
MAT 1 2 7.2E+10 0.33333 2758.0 0.0

$ Kapton film material properties
MAT 1 3 2.76E+9 0.45000 1358.0 0.0

$ Piezo ceramic material properties, non-zero CTE
MAT 1 4 6.9E+10 0.3 1000 7700.8.42E-7

$ SPC1 1 123456 504THRU 506
SPC1 1 123456 519THRU 523
SPC1 1 123456 554THRU 555

$ Unit thermal load (voltage) applied to actuator
TEMP 1 1 1.000 2 1.000 3 1.000
TEMP 1 4 1.000 5 1.000 6 1.000
TEMP 1 7 1.000 8 1.000 9 1.000
TEMP 1 10 1.000 11 1.000 12 1.000
TEMP 1 13 1.000 14 1.000 15 1.000
TEMP 1 16 1.000 17 1.000 18 1.000
TEMP 1 19 1.000 20 1.000 21 1.000
TEMP 1 22 1.000 23 1.000 24 1.000
TEMP 1 25 1.000 26 1.000 27 1.000
TEMP 1 28 1.000 29 1.000 30 1.000
TEMP 1 31 1.000 32 1.000 33 1.000
TEMP 1 34 1.000 35 1.000 36 1.000
TEMP 1 37 1.000 38 1.000 39 1.000
TEMP 1 40 1.000 41 1.000 42 1.000
TEMP 1 43 1.000 44 1.000 45 1.000
TEMP 1 46 1.000 47 1.000 48 1.000
TEMP 1 49 1.000 50 1.000 51 1.000
TEMP 1 52 1.000 53 1.000 54 1.000
TEMPD 1 0.0
ENDDATA
Below is a listing of the ‘static.dmap’ DMAP:

```
$compile sestatic, souin=mscsou, nolist, noref $
$
Following DMAP was coded for MSC/NASTRAN and verified to
work in version 2001.0.1 for the current application
MSC says this will work for versions 68.2,69,69.1 or 70
This DMAP stores static solution vectors in an output file
'S101SV1.OUT4' specified in the ASSIGN statement.
If the FEM does not contain rigid body modes (or mechanisms),
then the L-set d.o.f. are written to the output file.
If there are rigid body modes (a SUPORT card should be included
in the Bulk Data deck), then the T-set d.o.f. are written
to the output file.

alter 'CALL SUPER3' (1,0)
alter 685 $ after static solution SSG3
If there is no R-set, write the L-set vectors to file
if (norset < 0) then $
output4 ul,,,,//-1/15/1 $
$ Print L-set solution vector
matprn ul// $
endif $
If the R-set exists, write the T-set vectors to file
alter 822 $ After SDR1 module
if (norset <> -1) then $
$ Partition the G-set solution, UGVS, into the T-set vectors
if (nogset = notset) then $ no partitions if g-set=t-set
output4 ug,,,,//-1/15/1 $
$ matprn ug//$
else $
upartn use,tugvs,,,,/g't'o'1 $
$ matprn tugvs// $ T-set solution displacement vector
output4 tugvs,,,,//-1/15/1 $
$ $ end of "if (nogset=notset)"
endif $ end of "if (norset <> -1 )"
$
endalter
```
Appendix 2

The following NASTRAN input data deck listing is contained in the almq2e.dat file. The executive deck is set up to generate vibration modes. The DMAP sequence genkm_v2.dmap incorporated in the data deck by the ‘include’ command combines the Ritz vectors and eigenvectors and calculates the mass and stiffness matrices needed to assemble a reduced order model. The almq2e.dat file uses the file S101SV1.OUT4 generated by executing almq2.dat as input and generates two ASCII format output files: almq2e.asc and almq2e.pch containing the reduced mass and stiffness matrix and the displacements and/or strains for the structural modes including the Ritz vectors respectively.

ASSIGN INPUT4='S101SV1.OUT4' UNIT=15
ASSIGN OUTPUT4='almq2e.asc' UNIT=16 FORM=FORMATTED DELETE
ID ALBEAM WITH ACTUATOR,PROB
SOL 103
include 'genkm_v2.dmap'
   TIME   60
CEND
TITLE   NASTRAN FILE TRANSLATOR -- UNITS = SI
$ GLOBAL CASE
   SPC   1
METHOD=25
$ Output requests are for displacements at nodes 348 and 349
$ and strain for element 116.
$
SET 4 =116
SET 5 =348,349
DISPLACEMENT(PUNCH)=5
STRAIN(PUNCH,FIBER)=4
$
PARTN=5
$
BEGIN BULK
EIGRL,25,,,10
$
PARAM,K6ROT,100.
PARAM AUTOSPC  YES
PARAM POST    -2
GRID       1 0 7.14E-3 5.75E-2 0.00000 0
.
.
The rest of the data deck is identical to the listing included in ‘almq2.dat’ in Appendix 1. TEMP cards are ignored in an eigenvalue analysis.
The following is a listing of the ‘genkm_v2.dmap’ DMAP:

$MSC/NASTRAN DMAP
$This dmap was converted to V69 rjb 1/14/98
$This dmap was verified to work for version 2001.0.1
$This DMAP accepts user selected static solution vectors from an
$input file. The projection of these vectors onto a subspace
$perpendicular to the subspace of mass weighted normal modes
$is computed. These new static vectors are appended to the
$calculated normal modes, and generalized stiffness and mass
.matrices are formed. The static partition of these generalized
matrices are placed on an ASCII output file. Also written to
the ASCII output file are: a normal modes summary matrix,
and user selected static vectors in external sort order.

$ alter 'CALL.*IFPL.*DMINDX' $ after dmindx is formed
compile mods souin=mscsou, nolist, noref $ alter 'LAMA,OEIGS(1,1)' $ after REIGL and OFP modules
$ NEIGV = number of eigenvalues computed
$ NSV = number of columns in static vector matrix STATV
$ STATV is either an L-set or T-set dimensioned static vector matrix
input4 /statvlll,/l/15/-l/l $ read matrix of static vectors
$ Matrix SSV is an optional input. Matrix SSV is used to select
$ which static vectors will be used in the following analysis.
$ If matrix SSV is not input, then all the static vectors in the
$ input4 file will be used in the analysis.
$ DMIIN module is required to read the DMI bulk data entries.
$ The name of the input DMI data block is SSV.
$ type parm, ,i ,n,indxo k
message /'indxok = //'indxok $ domin dmi,dmindx/ssv,,, ,, ,, ,, l/99/99/0/0/s,indxo k
if (yesv) then $ If matrix SSV is input perform following operations
$ Postmultiply STATV by SSV to select the desired static vectors.
$ The resulting vectors are stored in DSV.
$mipyad statv,ssv,ldsv $ $ Let DSV overwrite STATV
$matprm dsv// $ equivalv dsv/statv/-1 $ DSV is renamed to STATV
$matprm statv// $
$param statv/trailer/1/s,n,nsv $ nsv=number of columns in STATV
$ message /"number of static vectors="/nsv/ modes=/neigv $ 
$ matprn statv,phix,kxx,mxx/ $ 
$param /"add's,n,nsr/ $ $nriz=number of modal+static vectors 
$ Form the following summation - 
$ [sproj]=[statv] - [phix][phix]'[mxx][statv] 
$ smpyad phix,phix,mxx,statv,,statv/sproj/4/-1/1/1/1 $ 
$ matprn sproj/ $ 
$ Append columns of STATV to columns of PHIX to form PHINEW 
magen ,/cpvec/6/nriz/neigv/ $ column partitioning vector 
merge phix,,statv,,cpvec,/phinew/l $ 
$ matprn phinew/ $ 
$ Compute system generalized stiffness and mass matrices 
smpyad phinew,kxx,phinew,/,genk/3/1///1/1///1/1/6 $Symmetric stiffness matrix 
smpyad phinew,mxx,phinew,/,genm/3/1///1/1///1/1/1/1/6 $ Symmetric mass matrix 
$ matprn genk,genm/ $ 
$ Extract the (NSV by NSV) static vector partition from the generalized 
$ stiffness and mass matrices 
$partn genk,cpvec,/,sgenk/-1 $ 
$partn genm,cpvec,/,sgenm/-1 $ 
$ matprn sgenk,sgenm/ $ 
lamx, ,lama/eigsum/-1 $ Convert LAMA from table to matrix format 
$ Expand LAMA to include the static vectors 
magen ,/pvfm/6/5/2/2/1 $ 
$partn eigsum,pvfm,/,fandm,/1 $ 
$ matprn fandm/ $ 
matgen ,/xcol/6/3/1/1/1 $ 
$ matgen ,/xrow/6/nriz/neigv/nsv $ 
merge fandm,,xcol,xrow/mlama/1 $ expanded transposed LAMA matrix 
$ matprn mlama/ $ 
$trnsplama/tlama $ 
lamx tlama,lamnew $ Convert LAMA matrix back into table form 
equivx lamnew/lama/-1 $ Store new table in LAMA 
equivx phinew/phix/-1 $ Store augmented eigenvectors in PHIX 
$ neigv = nriz $ NEIGV=no. of modes + no. of static vectors 
$ output4 eigsum,genk,genm,///-1/16/0 $ 
$ $ 
$Exiting sedrcvr nolist noref 
$alter 1157,1157 $ Before SDR2 
alter 'SDR2'( 1 ,-1) $ 
$ Convert displacements to the basic coordinate system 
vecplot ug,bgpds,eqexins,cstms,casedr,,,/ugvb/O/O/l////// $ 
$ $
$ Convert from internal to external sort
matgen eqexins/intext/9//lusets $
mpyad intext,ugvb./xgall/1 $
$ matprn xgall/1 $
$
$ Using Case Control input PARTN, create a partitioning vector for
the grids whose output is desired. The partitioning vector has
G-set dimension, and a value of 1.0 associated with the d.o.f.
matmod eqexins,uset,sils,casedr.,/ucpv,17//1 $ 
$ matprn ucpv/1 $ 
$ Convert partitioning vector to external sort order
mpyad intext,ucpv,/xucpv/1 $ 
$ matprn xucpv/1 $ 
$ Apply partitioning vector to extract desired grids
partn xgall,,xucpv/xcomp,xdgp,,/1 $ 
$ matprn xdgpl $ 
$
$ Output in ASCII format
$ Note: user must include in File Management Section the next line
$ assign output4='ritzv.asc' unit=16 form=formatted
output4 xdgpl,.,/0/16/0 $ 
endalter $

$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
Appendix 3

The following MATLAB script is used to create space state model matrices based on the number of inputs (number of actuators) and MSC/NASTRAN output request (element strain and/or nodal displacement). **Caution:** The MATLAB scripts are problem specific depending on the type of output requests, but easy to modify.

makemodel.mat

%******************************************************************************
% Script file MAKEMODEL reads NASTRAN Files
% with the reduced mass, stiffness, mode shapes, and Ritz vectors
% A state space model of the system is constructed with input in terms
% of voltages to the piezoactuators and output as desired
% Created by L.G. Horta and M.C. Reaves
% NASA Langley 8-01-01
%******************************************************************************

Fname2=input(' Enter NASTRAN ascii FileName for reduced mass and stiffness (CR=default):= ',s);
if isempty(Fname2);
    Fname2='almq2e.asc';disp(Fname2);
end

[FID,MESSAGE] = fopen(Fname2,'rt');

Stiff,Mass]=readMK(FID);

Fname2=input(' Enter NASTRAN Punch FileName (CR=default):= ',s);
if isempty(Fname2);
    Fname2='almq2e.pch';disp(Fname2);
end

[FID,MESSAGE] = fopen(Fname2,'rt');

nq=input(' Number of Output nodes (CR=default):= ');
if isempty(nq);
    nq=2;disp([' nq ',num2str(nq)]);
end

nm=input(' Number of Modes in File (CR=default):= ');
if isempty(nm);
    nm=11;disp([' nm ',num2str(nm)]);
end

ne=input(' Number of Strain ELEMENTS (CR=default):= ');
if isempty(ne);
    ne=1;disp([' ne ',num2str(ne)]);
end

Nplies=input(' Number of Plies per element (CR=default):= ');
if isempty(Nplies);
    Nplies=3;disp([' Nplies ',num2str(Nplies)]);
end

PlyNum=input(' Ply number where sensor data is measured (CR=default):= ');
if isempty(PlyNum);
    PlyNum=3;disp([' PlyNum ',num2str(PlyNum)]);
end

NR=input(' Enter Number of Ritz-Vectors (CR=default):= ');
if isempty(NR);
    NR=1;disp([' NR ',num2str(NR)]);
end

[eigen.Shape]=readShape(FID,nm,nq);
% read node information

[Strain]=readply(FID,Nplies,ne,PlyNum,nm):
% read element information

fclose(FID);

%******************************************************************************
% End of Read Punch File
%******************************************************************************
% construct State Space Model
%
ShapeStrain=[Shape: Strain];            % Stack output matrices
[A,B,C,D,ModelDesc]=ConstructABCD70301(NR,Mass,Stiff,ShapeStrain);
%          
% Output order is defined by NASTRAN File node order followed by element info
%          
% For Example: Node = [ 1x 1y 1z 2x 2y 2z ... nqx nqy nqz Element 1 .... ]

SYS=ss(A,B,C,D);
[Nout,junk]=size(C);
Nfpts=1600;
freq=linspace(.1,400,Nfpts)';
System=ss(A,B,C,D);
[mag,phase]=bode(System,freq*2*pi);
Mag=reshape(mag,[Nout*NR Nfpts]);   % Stored Columnwise
Phase=reshape(phase,[Nout*NR Nfpts]);
for j=1:NR
  for i=1:Nout
    subplot(2,1,1).semilogy(freq,Mag((j-1)*Nout+1,:));
    xlabel(' Freq (Hz)');ylabel(' Mag');
    title([' Input No. = ' num2str(j) ' Output No. = ' num2str(i)]);
    subplot(2,1,2).plot(freq,Phase((j-1)*Nout+1,:));
    xlabel(' Freq (Hz)');ylabel(' Phase');
    disp(' Hit Carriage return to see next plot')
    pause
  end
end

---------------------------------------------------------------

readMK.mat

function [Stiff,Mass]=readMK(FID)
%
% Function to read ASCII File from NASTRAN
% with reduced MASS and Stiffness Matrices
%
% The File must be 81 columns wide
%
% [Stiff,MASS]=readMK(FID)
%
% Input Information
% FID       = NASTRAN ASCII file name pointer (81 col. wide)
%
% Output Information
% Mass      = Reduced Mass Matrix
% Stiff     = Reduced Stiffness Matrix
%
% Written by Lucas G. Horta  01252001

if FID >= 0;
  LINE=fgets(FID);
  NN=str2num(LINE(1:20));   % Two Elements NCOL, NROW
  Mass=zeros(NN(2),NN(1));
  Stiff=zeros(NN(2),NN(1));
  for j=1:NN(1);
    LINE2=fgets(FID);
  end
end
NN2=str2num(LINE2(1:20));  % Two elements COL, RowPos
Mat=[];
TT=fscanf(FID,'%g',NN(1));
TSpace1=fgets(FID);
Stiff(:,j)=TT;
end
LINE2=fgets(FID);
LINE3=fgets(FID);
LINE4=fgets(FID);
for j=1:NN(1):  % Two elements COL, RowPos
  LINE2=fgets(FID);
  NN2=str2num(LINE2(1:20));
  Mat=[];
  TT=fscanf(FID,'%g',NN(1));
  TSpace=fgets(FID);
  Mass(:,j)=TT;
end
fclose(FID);
else
disp(' File not found')
end

readShape.mat

function [eigen,Shape]=readShape(FID,rmodes,nq)

  % Written by Horta-Reaves 7-03-01
  % Function to read PUNCHFILE from NASTRAN
  % with shape information
  % The File must be 81 columns wide
  % [eigen,Shape]=readShape(FID,rmodes,nq)
  %
  % Input Information
  % Filename   = NASTRAN Punch file name (ASCII) (81 col. wide)
  % rmodes     = Number of modes
  % nq         = Number of output nodes
  %
  % Output Information
  % eigen      = Eigenvalue squared (Rad/sec)
  % Shape      = Shape Vector size (ngpt*6 x rmodes)
  %
  %
end

  if FID >= 0;
    for ijk=1:rmodes
      for jj=1:6
        LINE=fgets(FID);  %Header
      end
      LINE7=fgets(FID);  % Eigenvalue ID
eigen(ijk)=str2num(LINE7(14:30));
      tem=[];
      for l=1:nq
        LINE1=fgets(FID);
        [nn,mm]=size(LINE1);
        pxyz=str2num(LINE1(20:mm));
      end
    end
  end
pxyz=pxyz(1:3);
LINE3=fgets(FID);
% Rotation DOF removed
% [nn,mm]=size(LINE3);
% txyz=str2num(LINE3(20:mm));
% txyz=txyz(1:3);
% tem=[tem pxyz txyz];
end
Shape(:,ijk)=tem';
end
if ~exist('File not found')
else
disp(' File not found')
end

readply.m

function [Strain]=readply(FID,Nplies,ne,PlyNum,nm)
% % Supporting file to read NASTRAN punch file info from composite elements
% % [Strain]=readply(FID,Nplies,ne,PlyNum,nm)
% % Input
% % FID = File handle
% % Nplies = Number of plies
% % PlyNum = Ply number where output is recovered
% % ne = Number of strain elements
% % nm = Number of modes
% % Output
% % Strain= [ne x nm] with strain influence in the Normal-1 Direction for all elements
% %
% % Read strain on PlyNum NORMAL-1 Direction
% % Created 8-01-01 by L. Horta
Strain=zeros(ne,nm);
Numberoflines=(PlyNum-1)*4;
for ijk=1:nm
    for jj=1:8
        LINE=fgets(FID);  %Header
        LINE1=fgets(FID);
        [nn,mm]=size(LINE1);
        exy=str2num(LINE1(20:mm));
        % Select NORMAL-1 Direction
end
% To get normal-2 change the number 2 to 3 (See NASTRAN Output convention)
exy = exy(2); 
tems = [tems exy];
for z2 = 1:(Nplies-PlyNum)*4+3
    LINE = fgets(FID);           % Remaining plies info not needed
end
Strain(:, ijk) = tems';
end

function [A,B,C,D,ModelDesc] = ConstructABCD70301(NR, Mass, Stiff, Shape)

% Function to construct A,B,C,D. matrices for a piezoelectric actuated system

% Input Information
% NR = Number of Ritz Vectors
% Mass = Mass Matrix
% Stiff = Stiffness Matrix

% Output Information
% eigen = Eigenvalue squared (Rad/sec)
% Shape = Shape Vector size (Number of Outputs x nmodes)
% ModelDesc = Text Describing output definition

% Written by Horta-Reaves 7-03-01 (Mod 8-01-01)

% construct State Space Model with Ritz vector and mode shape info
[NM, Njunk] = size(Mass);

% Reduction of Algebraic equation for Ritz Vectors
I1 = [1:NM-NR];
I2 = [NM-NR+1:NM];
[u, s, v] = svd(Mass);
Index = abs(diag(s)) > 1e-10;
NMnew = sum(Index);
Knew = u * Stiff * v;
Bnew = u * Stiff(:, I2);
K11 = Knew(I1, I1);
K12 = Knew(I1, I2);
K22 = Knew(I2, I2);
b1 = Bnew(I1, :);
b2 = Bnew(I2, :);
K22in = inv(K22);
CP = Shape * v;
C1_P = CP(:, I1);
C2_P = CP(:,12);
% Mod 3-28-01
[Ns1,junk]=size(Shape);
D_P=zeros(Ns1, NR);

Mred=s(Index,Index);

Kred=inv(Mred)*(K11-K12*K22in*K12);
Bred=inv(Mred)*(b1-K12*K22in*b2);
Cred_P=C1_P-C2_P*K22in*K12;
Dred_P=D_P+C2_P*K22in*b2;

[Sae,Ar]=eig(Kred);
Br=Sae*Bred;
Cr_P=Cred_P*Sae;
Dr_P=Dred_P;
disp(' Default damping value is 1 %');

SSS=input(' Do you want to change the damping value for each mode (l=yes,0=no) ?');
Damp=2*sqrt(Ar)*eye(NMnew,NMnew)*.01; % Percent Damping

if SSS == 1;
    DampValue=zeros(NMnew,1);
    for III=1:NMnew;
        disp(' Frequency = num2str(diag(Ar(III,III)))/(2*pi)' '(Hz)'))
        DampValue(III)=input('Enter damping value in % = ');
    end
    Damp=2*sqrt(Ar)*diag(DampValue)/100;
else
    A=[zeros(NMnew,NMnew) eye(NMnew,NMnew); -Ar -Damp];
    B=[zeros(NMnew, NR); Br];
    C_1=[Cr_P zeros(Ns1,NMnew)];
    C_2=[-Cr_P*Ar -Cr_P*Damp];
    D_1=Dred_P;
    D_2=Cr_P*Br+Dr_P;
    C=[C_1;C_2];
    D=[D_1;D_2];
    % Output Definition
    % ModelDesc=strvcat([' Number of output states is equal to ' num2str(Ns1*2)]);
    %[' The first ' num2str(Ns1) ' are translations'];
    %[' From ' num2str(Ns1+1) ' to ' num2str(Ns1*2) ' are accel. ']);
    disp(ModelDesc)
This paper presents a procedure for modeling structures containing piezoelectric actuators using MSC/NASTRAN and MATLAB. The paper describes the utility and functionality of one set of validated modeling tools. The tools described herein use MSC/NASTRAN to model the structure with piezoelectric actuators and a thermally induced strain to model straining of the actuators due to an applied voltage field. MATLAB scripts are used to assemble the dynamic equations and to generate frequency response functions. The application of these tools is discussed using a cantilever aluminum beam with a surface mounted piezoelectric actuator as a sample problem. Software in the form of MSC/NASTRAN DMAP input commands, MATLAB scripts, and a step-by-step procedure to solve the example problem are provided. Analysis results are generated in terms of frequency response functions from deflection and strain data as a function of input voltage to the actuator.

15. SUBJECT TERMS
MATLAB; MSC/NASTRAN; Finite element; Modeling; Piezoelectric actuators; Procedure; Structures