Chapter 3

FE Modeling Aspects
of
Smart Structures

Detailed contents

FE Modeling Aspects of Smart Structures

3.1 Introduction

As mentioned in Chapter 1, the smart structural structures and structural systems are so to say the type of Biomimic structures which can get adapted to the new environment. These smart structures are made up of smart materials and smart structural systems.

The Finite Element Analysis (FEA) has been carried out for the following activities to meet the displacement and frequency constraints of the specimen in the Linear Statics and Dynamics domain. The bare beam and bare composite reflectors have been modeled using NISA. The piezo patch related beam specimen and piezo coated composite reflectors have been modeled using ATILA software .

Repetitive iterations of analysis were undertaken to finalize the dimensional parameters enumerated below by modifying the Stiffness (K) to Mass ratio (K/M) which eventually helped in picking the right option for experimentation :

- Thickness of the specimen
- Orientation of specimen
- Size of the specimen
- Thickness of the piezo coating
- Size of the piezo patches
- Percentage coverage of the piezo patches on the test specimen
- Layout options of the piezo patches

3.2 Advantages of Smart Structures & Structural Systems

Smart materials and structural systems have become popular particularly because of the following advantages for which FE modeling is typical as described in section 3.3 :

- Lower overall wt.

  As the components are fastened directly to the structures, and not to additional carriers, thus reduction in overall weight.

- Improved sensor sensitivity

Depending on the area where the sensors are embedded, sometimes to increase sensor area, the whole structure may act as extended sensor itself.

• Improved thermal control

As the functional components are embedded, a simple & effective heat transfer can be achieved by integrating thermal conductors into the structure.

• Higher reliability

With self diagnostic / On–line health monitoring features preflight impact damages can be estimated. Damage detection is also useful for reusable spacecraft FRP structures.

3.3 Theoretical aspects

The theoretical aspects for the constitutive equations for FE modeling of smart structures with smart materials are described in the following form :
Variational principles are used to develop the finite element equations which incorporate the piezoelectric effect. These materials have non-linearities and hysteresis issues at present which pose a challenge in theoretical / mathematical modeling of their behaviors. In the present investigation, these non-linearities and hysteresis losses have not been modeled in the mathematical model. Many formal / informal interactions the author had with academicians and industry to generate the knowledge base to realize the aim of the investigation.

It was decided to use two commercial FEA software (ATILA & NISA) to attempt the highly indeterminate real life practical problems in a more realistic & meaningful way.
The electromechanical constitutive equations for linear material behavior are [183]: (Eq 3.1 – 3.24)

$$\{T\} = [C]\{S\} - [e]\{E\} \qquad \text{Eq. (3.1)}$$

$$\{D\} = [e]^T\{S\} + [\mathit{E}]\{E\} \qquad \text{Eq. (3.2)}$$

or equivalently,

$$\begin{Bmatrix} \{T\} \\ \{D\} \end{Bmatrix} = \begin{bmatrix} [C] & [e] \\ [e]^T & -[\mathit{E}] \end{bmatrix} \begin{Bmatrix} \{S\} \\ -\{E\} \end{Bmatrix} \qquad \text{Eq. (3.3)}$$

where:

{T} = Stress vector

{D} = electric flux density vector

{S} = strain vector

{E} = electric field vector

[e] = Piezoelectric matrix

[E] = Dielectric matrix (evaluated at constant mechanical strain)

[C] = Elasticity matrix (evaluated at constant electric field)

Equations (3.1) and (3.2) are the usual constitutive equations for structural and electrical fields, respectively, except for the coupling terms involving the piezoelectric matrix [e].

The elasticity matrix [c] is as follows:

$$[C]=\begin{bmatrix} c11 & c12 & c13 & c14 & c15 & c16 \\ & c22 & c23 & c24 & c25 & c26 \\ & & c33 & c34 & c35 & c36 \\ \text{Symmetric} & & & c44 & c45 & c46 \\ & & & & c55 & c56 \\ & & & & & c66 \end{bmatrix} \quad \text{Eq. (3.4)}$$

The piezoelectric matrix [e] relate the electric field vector {E} in the order X, Y, Z to the stress vector {T} in the order X, Y, Z, XY, YZ, XZ and is of the form:

$$[e] = \begin{bmatrix} e11 & e12 & e13 \\ e21 & e22 & e23 \\ e31 & e32 & e33 \\ e41 & e42 & e43 \\ e51 & e52 & e53 \\ e61 & e62 & e63 \end{bmatrix} \quad \text{Eq. (3.5)}$$

The dielectric matrix [e] is of the form:

$$[E] = \begin{bmatrix} \epsilon_{11} & 0 & 0 \\ 0 & \epsilon_{22} & 0 \\ 0 & 0 & \epsilon_{33} \end{bmatrix} \qquad \text{Eq. (3.6)}$$

The finite element discretization is performed by establishing nodal solution variables and element shape functions over an element domain which approximate the solution.

$$\{u_e) = [Nu]^T \{u\} \qquad \text{Eq. (3.7)}$$

$$V_e = \{N^V\}^T \{V\} \qquad \text{Eq. (3.8)}$$

where: $\{u_e\}$ = displacements within element domain in the x, y, z directions

Ve = electrical potential within element domain

$[N^u]$ = matrix of displacement shape functions

$\{N^V\}$ = vector of electrical potential shape function

$\{u\}$ = vector of nodal displacements

$\{V\}$ = vector of nodal electrical potential

Expanding these definitions:

$$[N^u]^T = \begin{bmatrix} N_1 & 0 & 0 & N_n & 0 & 0 \\ 0 & N_1 & 0 & \ldots 0 & N_n & 0 \\ 0 & 0 & N_1 & \ldots 0 & 0 & N_n \end{bmatrix} \qquad \text{Eq. (3.9)}$$

$$\{N^V\}^T = [\ N1\ \ N2\ \ \ldots\ Nn\ ] \qquad \text{Eq. (3.10)}$$

where: Ni = shape function for node i

$$\{u\} = [UX1\ \ UY2\ \ UZ3\ \ \ldots\ \ UXn\ \ UYn\ \ UZn\ ]^T \qquad \text{Eq.(3.11)}$$

$$\{V\} = \begin{Bmatrix} V_1 \\ V_2 \\ . \\ . \\ . \\ V_n \end{Bmatrix} \qquad \text{Eq. (3.12)}$$

where: n = number of nodes of the element

Then the strain {S} and electric field {E} are related to the displacements and potentials, respectively, as:

$$\{S\} = [B_u]\{u\} \qquad \text{Eq. (3.13)}$$

$$\{E\} = -[B_v]\{V\} \qquad \text{Eq. (3.14)}$$

$$[B_u] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial z} \end{bmatrix} [N^u]^T \qquad \text{Eq. (3.15)}$$

$$[Bv] = \begin{Bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{Bmatrix} \{N^V\}^T \qquad \text{Eq. (3.16)}$$

After the application of the variational principle and finite element discretization, the coupled finite element matrix equation derived for a one element model is:

$$\begin{pmatrix} [M] & [0] \\ [0] & [0] \end{pmatrix} \begin{Bmatrix} \{\ddot{u}\} \\ \{\ddot{v}\} \end{Bmatrix} + \begin{pmatrix} [C] & [0] \\ [0] & [0] \end{pmatrix} \begin{Bmatrix} \{\mathring{u}\} \\ \{\acute{v}\} \end{Bmatrix} + \begin{pmatrix} [K] & [K^z] \\ [K^z]^T & [K^d] \end{pmatrix} \begin{Bmatrix} \{u\} \\ \{v\} \end{Bmatrix} = \begin{Bmatrix} \{F\} \\ \{L\} \end{Bmatrix} \qquad \text{Eq. (3.17)}$$

where a dot above a variable denotes a time derivative. The following equations provide an explanation of the sub matrices in equation (3.17):

Structural mass:

$$[M] = \int vol\, \rho\, [N^u]\, [N^u]^T\ d(vol) \qquad \text{Eq. (3.18)}$$

where, p = mass density

Structural damping:

It is same as the matrix of [C]

Structural stiffness:

$$[K] = \int vol\, [B_u]^T\ [C]\, [B_u]\ d(vol) \qquad \text{Eq. (3.19)}$$

Dielectric Conductivity:

$$[K^d] = - \int vol\ [B_v]^T\ [E]\ [B_v]\ d(vol) \qquad \text{Eq. (3.20)}$$

Piezoelectric coupling matrix:

$$[K^z] = \int_{vol} [B_u]^T\ [e]\, [B_v]\ d(vol) \qquad \text{Eq. (3.21)}$$

Structural load vector:

{F} = vector of nodal forces, surface forces, and body forces.

Electrical load vector:

$\{L\} = \{L^{nd}\}$ = applied nodal charge vector

In a harmonic response analysis, the potential DOF is allowed as a master DOF. Energy coefficients are calculated for each piezoelectric element as follows :

Elastic energy:

$$U_E = ½ \ \{u\}^T \ [K]\{u\} \qquad \text{Eq. (3.22)}$$

Dielectric energy:

$$U_D = ½ \ \{V\}^T \ [K^d]\{V\} \qquad \text{Eq. (3.23)}$$

Electromechanical coupling energy: Eq. (3.24)

$$U_M = \frac{1}{4} (\{ u \}^T [K^2] \{ V \} + \{ V \}^T [K^2]^T \{u\})$$

3.3.1 Steps of FEA

The major steps of FEA carried out have been enumerated a follows:

Mechanical vs Electro-Mechanical problem

NISA	ATILA
(Conventional Structural Modeling of a Mechanical problem)	(Smart Structures Modeling of a Electro-Mechanical problem)
1. Discretization of geometry model	1. Discretization of geometry model
2. Choosing a shape function	2. Choosing a shape function
3. Formulation of Element member stiffness matrices, $[S_m]$	3. Formulation of, [e]= Piezoelectric matrix [*E*]**=** dielectric matrix (At constant mechanical strain) [C]**=**Compliance or Elasticity matrix (evaluated at constant electric field)
Ux, Uy, Uz, Rx, Ry & Rz (3 translations, 3 rotations)	Ux, Uy and Uz and φ (3 translations, 1 electric potential)
4. Transformation to [K] Structure Stiffness matrix introduce Boundary Conditions(BCs)	4.Transformation to [K] Structure Stiffness matrix with BCs for Piezoelectric coupled-field analysis
5. {p}=Load vector compilation	5.Force {F} and nodal voltage $\{L^{nd}\}$
6. Solution of equations	6. Solution of equations
7. Evaluation of secondary quantities viz., δ, σ , ε ….	7. Evaluation of secondary quantities Viz., δ, Stress {T} , {D}electric flux density vector

3.3.2 A typical Electro-Mechanical 3D model problem approach

Firstly, smart structures modeling packages require a dielectric matrix [*E*], a piezoelectric matrix [e], and either a compliance matrix or a stiffness matrix [K].

- The dielectric matrix,

  $[E] = [\ \ ]_{3x3}$, units of Farads/meter.

- The piezoelectric matrix,

  $[e] = [\ \ ]_{6x3}$, units of Coulombs/meter$^{2.}$

- Compliance matrix or the stiffness matrix for a piezoelectric coupled-field analysis,

  $[C] = [\ \ ]_{6x6}$, Compliance meter^2/Newton, or

  $[K] = [\ \ ]_{6x6}$, Stiffness matrix Newtons/meter^2 is commonly used.

The modulus of elasticity, the Poisson ratio, and the coefficient of thermal expansion are directly input into the matrices.

Using the piezo manufacturer's published data, the proper ATILA input has to be computed for the piezoelectric material. This is accomplished using Macro in-built into the element Library data file of the ATILA. The published material data is converted by this Macro into the format needed for the FE model (without losses case) for the following types of tensors where the subscript notation, 11 and 33, indicate the transverse and axial directions respectively:

- Piezoelectric charge constants :

d_{31} meters / volt

d_{33} meters / volt

d_{15} meters / volt

These constants relate the voltage and displacement behavior of the Piezoelectric material and the subscript notations indicate the transverse coupling factor (31), the longitudinal coupling factor (33) and the shear coupling factor (15).

- Elastic constants , short circuit :

S^E_{11} m^2N

S^E_{33} m^2N

S^E_{44} m^2N

S^E_{12} m^2N

S^E_{13} m^2N

These constants define the material compliance, measured as the strain over the stress. The superscript notation, E, indicates that the data is measured with the electrodes connected, i.e., at a short circuit condition. The subscripts identify the transverse directions (1 and 2), the axial direction (3), and the shear axes (4, 5, and 6) as per the details given in Chapter 4 which deals with the theoretical aspects of Piezo materials.

Then the transformation of the element matrices is done to generate the structure stiffness matrix populated with the Electro-mechanical-coupling terms.

Finally, after the solution of equations the secondary quantities evaluated in linear static analysis are as follows:

{S} represents 6 components of strain; {D} represents the 3 components of the electric flux density vector.

For Eigen value analysis along with the Mass [M] matrix following matrices are needed for extraction of Eigen values :

[K] diagonal matrix is the system stiffness matrix. {S} represents 6 components of strain; [e] is the piezoelectric matrix; {E} represents the three components of the electric field; {D} represents the 3 components of the electric flux density, $[K^d]$ is the dielectric Conductivity matrix relating the electric field to electric flux density and $[K^2]$ is the piezoelectric coupling matrix.

3.4 General Presentation of ATILA

ATILA is a finite element code that has been specifically developed to aid in the design of sonar transducers, but can also be used for all types of transducers (industrial machining, cleaning, welding, nondestructive testing, acoustic imaging, actuators) or for passive structures. Its working domain is one of small and linear strains. It permits the static, modal, harmonic and transient analysis of unloaded elastic, piezoelectric, or Magnetostrictive structures, as well as the harmonic and transient analysis of radiating elastic or piezoelectric structures (in any fluid, water or air, for example) and modal or harmonic analysis of periodic structures with 1D, 2D or 3D periodicity. It is able to perform analyses of ax- symmetrical, bi or three-dimensional structures. Depending upon the problem, it provides: the displacement field, the nodal plane positions, the stress field, near-field and far-field pressures, transmitting voltage response, directivity patterns, electrical impedance.

For the present investigation of the piezoelectric powder cantilever beam composite specimens & piezo coated reflectors, ATILA software has been used in the linear dynamic domain to predict the natural frequency of the specimen.

3.5 Assumptions

Following are the assumptions made in the FE modeling aspects:

- It is assumed that the piezo actuators are perfectly bonded to the test specimen and the strains in the actuators are compatible with the base substrate.

- A linearly varying strain is assumed across the thickness of specimen & the actuator.
- The strains developed in the crystals are within elastic limits.
- Adhesives have not been considered in the FE modeling.
- Equivalent stiffness properties of the laminate have been considered in FE model.
- Elastic isotropic behavior has been assumed for the piezoelectric material, hence all tensors have been evaluated for without losses case.
- The piezo powder layer on the composite specimen acts like viscoelastic layer and is not subjected to any electrical field at varying temperatures.

3.6 FE Model Definition

Firstly, it is necessary to define the kind of analysis-needed in ATILA, for example: computation of the static deformation of an elastic structure under a concentrated mechanical load, electrical short-circuited modal analysis of a piezoelectric ceramic stack, or computation of the voltage response of a transducer. Secondly, it is important to choose the type(s) of elements needed to describe the region under study: elastic, piezoelectric, Magnetostrictive or fluid elements, bi- dimensional, three-dimensional or ax symmetrical elements, plane stress, plane strain, plate or shell elements, etc.

At this point, a first estimation of the computation size can generally be made by estimating the number of nodes, elements, and degrees-of-freedom. Also at this point, all the physical parameters are determined.

3.7 Mesh Generation

Mesh generation involves the splitting of the aforementioned specimen into FE elements. This is accomplished by defining the nodes and by specifying their coordinates in a given order, called node-numbering order. Then, the elements are described by listing the nodes for each element in a given order, called topology. Appropriate finite elements are chosen from the ATILA Library of elements. These elements allow curved line modeling and lower density meshes. However, it is essential to ensure the validity of the geometrical aspect of the elements as well as the mesh size. Mesh generation can be completed by manual approach or if the structure's shape allows the use of an automatic mesh feature can be exercised. In case of parabolic reflectors, the coordinates of the parabolic profile have been calculated and entered and then rotated 360^0 about the axis of symmetry to generate the surface of reflector.

The ATILA solver is the interface to the program that sets the array sizes, calls up the subroutines for reading the data files, computes the elementary matrices, assembles these matrices into global matrices, solves the equations, and stores the results. It creates a results file and several data files for post-processing.

3.8 Piezoelectric Material Data & Characterization

Most manufacturers of piezoelectric materials do not publish the material properties in a format that can be directly entered into a FE model. The published data needs to be converted and transformed to populate the necessary material matrices that FE package expects for the piezoelectric material input. The basic piezo constants which are available in the brochures are fed into the database by sometimes running a typical Macro developed for the chosen type of finite element for modeling the given geometry.

The material properties which populate the matrices in Eqs. (3.1) and (3.2) are not easily measured directly. However, the properties in the inverses of the matrices are readily available; therefore, an alternate form of the equations is usually introduced.

Some packages require a dielectric matrix [*E*], a piezoelectric matrix [e], and either a compliance matrix or a stiffness matrix [K].

The dielectric matrix, [*E*], defines the electrical permittivity in typical units of Farads/meter. This matrix is a 3x3, diagonal matrix for a 3D model.

The piezoelectric matrix relates the electric field to stress and typically has units of Coulombs/meter2. For a 3D model, this matrix is a 6x3 matrix and the data is entered into ATILA as a material data.

While ATILA accepts stiffness data in terms of the elastic modulus and Poisson's ratio, but as a common practice it is defined either in the form of compliance matrix or the stiffness matrix [K] for a piezoelectric coupled-field analysis. Units for the terms in the compliance matrix are typically meter^2/Newton, while in the stiffness matrix [K], Newtons/meter^2 is commonly used. Other matrices are as follows:
[K] diagonal matrix is the system stiffness matrix. {S} represents 6 components of strain; [e] is the piezoelectric matrix; {E} represents the three components of the electric field; {D} represents the 3 components of the electric flux density, [K^d] is the dielectric Conductivity matrix relating the electric field to electric flux density and [K^2] is the piezoelectric coupling matrix.

In the structural analysis w.r.t the present investigation, the material properties for all the layers, excluding the PZT layer (bare specimen), are isotropic and have been analyzed using a FE package called NISA (Numerically Integrated elements for System Analysis). The modulus of elasticity, the Poisson ratio, and the coefficient of thermal expansion are directly input into the matrices.

These constants define the material compliance, measured as the strain over the stress. The superscript notation, E, indicates that the data is measured with the electrodes connected, i.e., at a short circuit condition. The subscripts identify the transverse directions (1 and 2), the axial direction (3), and the shear axes (4, 5, and 6) as per the details given in Chapter 4 dealing with the theoretical aspects of Piezo materials.

Using the piezo manufacturer's published data, the proper ATILA input has to be computed for the piezoelectric material. This is accomplished using Macro in-built into the element Library data file of the ATILA. The published material data is converted by this macro into the format needed for the FE model. For example for typical tensors of the following two cases; the data can be generated using the above approach:

- Case of a Piezoelectric Material without Losses
- Case of a Piezoelectric Material with Losses

For the case of an elastic isotropic material without losses, following are the parameters :

E, NU, RHO

where E is Young's modulus, NU is Poisson's ratio and RHO the density.
For the case of an elastic isotropic material with losses, following are the parameters:

E' , NU', RHO,0.0 ,NU", E"

where E' and E" are, respectively, the real and imaginary parts of Young's modulus, NU' and NU" the real and imaginary parts of Poisson's ratio, and RHO the density. E" must be positive and NU" comprised between, E"(1 +NU')/E' and E"(O.5-NU')/E' to have a material with losses.

The tensor details for both the options are as follows:

Case of a Piezoelectric Material without Losses as per ATILA ,

0.0	0.0	RHO	0.0	0.0	0.0
s_{11}^E	s_{12}^E	s_{13}^E	s_{14}^E	s_{15}^E	s_{16}^E
s_{21}^E	s_{22}^E	s_{23}^E	s_{24}^E	s_{25}^E	s_{26}^E
s_{31}^E	s_{32}^E	s_{33}^E	s_{34}^E	s_{35}^E	s_{36}^E
s_{41}^E	s_{42}^E	s_{43}^E	s_{44}^E	s_{45}^E	s_{46}^E
s_{51}^E	s_{52}^E	s_{53}^E	s_{54}^E	s_{55}^E	s_{56}^E
s_{61}^E	s_{62}^E	s_{63}^E	s_{64}^E	s_{65}^E	s_{66}^E
d_{11}	d_{12}	d_{13}	d_{14}	d_{15}	d_{16}
d_{21}	d_{22}	d_{23}	d_{24}	d_{25}	d_{26}
d_{31}	d_{32}	d_{33}	d_{34}	d_{35}	d_{36}
ε_{11}^s	ε_{12}^s	ε_{13}^s	0.0	0.0	0.0
ε_{21}^s	ε_{22}^s	ε_{23}^s	0.0	0.0	0.0
ε_{31}^s	ε_{32}^s	ε_{33}^s	0.0	0.0	0.0

where (s^E) is the constant electrical field elastic tensor, (d) the piezoelectric tensor, (ε^S) the constant strain dielectric permittivity tensor, and RHO the density as per the ATILA manual [173]&[174].

Case of a Piezoelectric Material with Losses as per ATILA

0.0	0.0	RHO	0.0	0.0	0.0
$s_{11}^{E'}$	$s_{12}^{E'}$	$s_{13}^{E'}$	$s_{14}^{E'}$	$s_{15}^{E'}$	$s_{16}^{E'}$
$s_{21}^{E'}$	$s_{22}^{E'}$	$s_{23}^{E'}$	$s_{24}^{E'}$	$s_{25}^{E'}$	$s_{26}^{E'}$
$s_{31}^{E'}$	$s_{32}^{E'}$	$s_{33}^{E'}$	$s_{34}^{E'}$	$s_{35}^{E'}$	$s_{36}^{E'}$
$s_{41}^{E'}$	$s_{42}^{E'}$	$s_{43}^{E'}$	$s_{44}^{E'}$	$s_{45}^{E'}$	$s_{46}^{E'}$
$s_{51}^{E'}$	$s_{52}^{E'}$	$s_{53}^{E'}$	$s_{54}^{E'}$	$s_{55}^{E'}$	$s_{56}^{E'}$
$s_{61}^{E'}$	$s_{62}^{E'}$	$s_{63}^{E'}$	$s_{64}^{E'}$	$s_{65}^{E'}$	$s_{66}^{E'}$
$d_{11}^{'}$	$d_{12}^{'}$	$d_{13}^{'}$	$d_{14}^{'}$	$d_{15}^{'}$	$d_{16}^{'}$
$d_{21}^{'}$	$d_{22}^{'}$	$d_{23}^{'}$	$d_{24}^{'}$	$d_{25}^{'}$	$d_{26}^{'}$
$d_{31}^{'}$	$d_{32}^{'}$	$d_{33}^{'}$	$d_{34}^{'}$	$d_{35}^{'}$	$d_{36}^{'}$
$\varepsilon_{11}^{s'}$	$\varepsilon_{12}^{s'}$	$\varepsilon_{13}^{s'}$	0.0	0.0	0.0
$\varepsilon_{21}^{s'}$	$\varepsilon_{22}^{s'}$	$\varepsilon_{23}^{s'}$	0.0	0.0	0.0
$\varepsilon_{31}^{s'}$	$\varepsilon_{32}^{s'}$	$\varepsilon_{33}^{s'}$	0.0	0.0	0.0
$s_{11}^{E''}$	$s_{12}^{E''}$	$s_{13}^{E''}$	$s_{14}^{E''}$	$s_{15}^{E''}$	$s_{16}^{E''}$
$s_{21}^{E''}$	$s_{22}^{E''}$	$s_{23}^{E''}$	$s_{24}^{E''}$	$s_{25}^{E''}$	$s_{26}^{E''}$
$s_{31}^{E''}$	$s_{32}^{E''}$	$s_{33}^{E''}$	$s_{34}^{E''}$	$s_{35}^{E''}$	$s_{36}^{E''}$
$s_{41}^{E''}$	$s_{42}^{E''}$	$s_{43}^{E''}$	$s_{44}^{E''}$	$s_{45}^{E''}$	$s_{46}^{E''}$
$s_{51}^{E''}$	$s_{52}^{E''}$	$s_{53}^{E''}$	$s_{54}^{E''}$	$s_{55}^{E''}$	$s_{56}^{E''}$
$s_{61}^{E''}$	$s_{62}^{E''}$	$s_{63}^{E''}$	$s_{64}^{E''}$	$s_{65}^{E''}$	$s_{66}^{E''}$
$d_{11}^{''}$	$d_{12}^{''}$	$d_{13}^{''}$	$d_{14}^{''}$	$d_{15}^{''}$	$d_{16}^{''}$
$d_{21}^{''}$	$d_{22}^{''}$	$d_{23}^{''}$	$d_{24}^{''}$	$d_{25}^{''}$	$d_{26}^{''}$
$d_{31}^{''}$	$d_{32}^{''}$	$d_{33}^{''}$	$d_{34}^{''}$	$d_{35}^{''}$	$d_{36}^{''}$
$\varepsilon_{11}^{s''}$	$\varepsilon_{12}^{s''}$	$\varepsilon_{13}^{s''}$	0.0	0.0	0.0
$\varepsilon_{21}^{s''}$	$\varepsilon_{22}^{s''}$	$\varepsilon_{23}^{s''}$	0.0	0.0	0.0
$\varepsilon_{31}^{s''}$	$\varepsilon_{32}^{s''}$	$\varepsilon_{33}^{s''}$	0.0	0.0	0.0

where (s^E) is the constant electrical field elastic tensor, (d) the piezoelectric tensor, (ε^S) the constant strain dielectric permittivity tensor, and RHO the density as per the ATILA manual [173] & [174]. For the present investigation, the elastic isotropic model has been considered without hysteresis losses.

The characterization of a piezoceramic involves the determination of 10 constants: Five elastic constants, two dielectric constants and three piezoelectric constants. These constants have been described in detail in Chapter 4. The characterization of these constants is done by the manufacturer.

In addition, there is also a contemporary concept nowadays of "very smart materials", but they are still not called intelligent materials. They have learning or tuning function that makes it possible for them to become smarter. These materials take advantage of nonlinear properties such as electrostriction or higher order elastic constants. By applying a bias field, we can tune the level of sensitivity, that is, the degree of "smartness".

3.9 ATILA Finite Elements

Following is the list of useful elements of ATILA from the present investigation point of view. The details of these elements are given in Appendix B:

- ELASTIC

  10 noded, Tetrahedral, TETR10E
  ACTIVE Degrees of Freedom,
  Ux, Uy and Uz (#3 translations)

- PIEZOELECTRIC

  10 noded, Tetrahedral, TETR10P
  ACTIVE Degrees of Freedom,
  Ux, Uy and Uz and φ (#3 translations, 1 electric potential)

Finite Elements used in the modeling of the piezo patches & for coated composite reflectors are chosen from the ATILA manual [173] & [174]. 3D elements like PIEZO-17-PZT5A have been used for the modeling of the piezo layer as well as piezo patches. Bare specimen have been modeled using NKTP 20 General Plate Shell Element of NISA software, where equivalent properties of the composite shell are used to predict the Eigen values of the cantilever beam and reflector specimen at room temperature. The other details are described in Chapter 5.

3.10 NISA Finite Elements

Following is the list of useful elements of NISA from the present investigation point of view. The details of these elements are given in Appendix B:

- NKTP20, NORDR 1

8 noded, General Plate / Thin Shell
ACTIVE Degrees of Freedom,
Ux, Uy,Uz, θx, θy and θz
(#3 translations # 3 rotations)

- NKTP12, NORDR 1

2 noded, 3D Beam elements
ACTIVE Degrees of Freedom,
Ux, Uy,Uz, θx, θy and θz
(#3 translations # 3 rotations)

Bare specimen have been modeled using NKTP 20 General Plate Shell Element of NISA software [175] & [176] , where equivalent properties of the composite shell are used to predict the Eigen values of the specimen at room temperature.

3.11 Raw Material Properties

Following are the structural properties of the various layers of the composites which have been used in the fabrication of the test specimen. The structural properties mentioned below have been used in the FE modeling in Chapter 5 & Chapter 8 respectively for composite reflectors made up of CFRP & GFRP . Linear dynamic analysis cases have been run with the following structural properties in Chapter 5 for composites by assuming the substrate material as homogeneous . Linear static analysis cases have been run with structural properties of Lexan material in Chapter 8 for carrying out the preliminary shape deformation studies of parabolic reflectors.

- Reinforcements

Table 3.1 gives the tested values of the mechanical and physical properties of unidirectional carbon fiber / epoxy composite M55J / and Kevlar fabric prepreg / epoxy composites for space qualified specimen tested at varying temperatures in the present investigation for vibration damping characteristics . For the FE modeling of the composite beams and reflector specimen at ambient temperature conditions , equivalent properties have been taken for structural analysis in the eigen values domain where the adhesive layers have not been modeled in the Finite element analysis :

Table 3.1 Structural Properties of the Composites

S. No.	Property	Unit	M55J carbon fibre /epoxy for 90 %prob. & 95 % CL	Kevlar fabric / 914epoxy for 90 % prob. & 95 % CL
1	Longitudinal tensile strength	N/mm^2	1327.00	245
2.	Longitudinal tensile modulus	N/mm^2	294300.00	27450
3.	Longitudinal compressive strength	N/mm^2	716.13	122
4.	Transverse tensile strength	N/mm^2	21.582	245
5.	Transverse tensile modulus	N/mm^2	5957.50	27450
6.	Transverse compressive strength	N/mm^2	117.40	122
7.	Inplane shear strength	N/mm^2	74.85	33
8.	Inplane shear modulus	N/mm^2	4896.40	1176
9.	Flexural modulus	N/mm^2	902.60	290
10.	Flexural modulus	N/mm^2	264870.00	18000
11.	Interlaminar shear strength	N/mm^2	74.85	34
12.	Major Poisson ratio		0.346	0.058
13.	Density	gm/cc	1.70	1.35
14.	Coefficient of linear expansion of fibre direction	/° C	-2.31×10^{-6}	--
15.	Coefficient of linear expansion in transverse direction	/° C	40.3×10^{-6}	--

- Electrically Conductive Adhesive

1. Designation - Eccobond 60 C /CAT 9
2. Lap shear strength - 4 MP a
3. Curing temperature - 50 C for 2 hrs

- Aluminium alloy

1. Designation - 6061 T 651 A1 Alloy
2. Composition % (Nominal) - Si – 0.6, Cu –0.2
   Mg –1.0, Cr – 0.2
3. 0.2% Proof tensile stress - 240 MP a
4. Ultimate tensile stress (typical) - 255 MP a
5. Ultimate shear stress (typical) - 200 MP a
6. Elongation % (nominal) - 10
7. Modulus of Elasticity , E - 0.7E+06 Kg /cm^2
8. Density , ρ - 2.7 gm / cc

- Film adhesive

1. Designation - REDUX 319L
2. Lap shear at R.T. - 20 MP a
3. Weight in kg/ Sq.m - 0.18
4. Thickness - 0.08 mm
5. Curing temperature - 175 C

- Araldite 2024A & B

1. Designation - 2014 A and 2014 B
2. Mixing ratio - 2:1
3. Lap shear Strength - 5 MP a

- T-300 Carbon fiber cloth / LY 556 + HT 972 Epoxy

1. Tensile Modulus in fiber direction – 45000 MPa
2. Breaking strength of fabric - 1200 N/inch
3. In plane shear strength - 22 MPa
4. Inter Laminar shear strength - 22 MPa
5. Major poisson ratio - 0.103
6. Density - 1.5 – 1.6 gm /cc

- Equivalent Laminate Stiffness Properties & Allowable stress

CFRP lay up for rib	CFRP/KFRP lay up for sandwich panel
(0/09/45/-45)	(0/90)KFRP (0/90/45/-45)CFRP
EX = 104190.92 MP a	EX =73096.6 MP a
EX = 104190.92 MP a	EY = 73096.6 MP a
Gxy = 39554.76 Mp a	Gxy = 24368.5 MP a
Nuxy = 0.317	Nuxy =0.2914
Allowable stress = 270 MP a	Allowable stress = 165.5 MP a

- The EX-1516 Cyanate Ester film adhesive for space qualified specimen

The EX-1516 Cyanate Ester film adhesive has been used in the development of the graphite & Kevlar specimen which have proposed to be tested at varying temperatures. This has been formulated for use in specific applications where low moisture absorption and/or low dielectric constant/low loss are of utmost importance. The resin system's strength and toughness when bonding solid, honeycomb or foam core structures is comparable and often greater than high performance epoxy adhesives.

Due to the Cyanate Ester resin system's inherent low shrinkage during cure, bonded structures retain less inherent stress and will therefore remain dimensionally stable during thermal cycling. This factor is of extreme importance when bonding structures for use in space. Cyanate Ester's, EX-1516 film adhesive displays low out gassing and micro cracking properties to assure structural integrity even after severe environmental exposure and radiation bombardment.

Material properties

Moisture Pickup		0.6-0.7%
Dielectric Constant (10 GHz)		2.6-2.7
Loss Tangent (10 GHz)		0.005-0.006
Outgassing	TML	0.179 %
	CVCM	0.007 %
Density	ρ	1.17 gm /cc

- Structural Adhesive - Araldite AV 138 M / Hardener HV 998

Solvent free thixotropic paste

Heat resistant to 120°C

Epoxy adhesive

Density	ρ	1.65 - 1.75 gm /cc
Shear strength	(DIN 53283)	>12 N/mm^2

This structural adhesive has been found to give good strain transfer and has been used for pasting piezo patches on Aluminum & Polycarbonate materials . It has been described in Chapter 8 of the thesis .

- GFRP : Chop strand Mat with random orientation of fibers

Modulus of Elasticity	, E	-	2.1E+05 Kg /cm^2
Density	, ρ	-	1.78 gm / cc

It has been described in Chapter 5 of the thesis .

The theoretical aspects of piezoceramic materials which link all these parameters are described in Chapter 4.

*****