

*Measurement
Good Practice Guide*

**Piezoelectric Finite
Element Analysis**

Fiona Lowrie

Defence Evaluation Research Agency,

Farnborough

Mark Stewart, Markys Cain &

Mark Gee

National Physical Laboratory

*Meeting
the challenge*

**Surface
engineering**

Casting

**Ceramics
PROCESSING
composi**

NPL 
National Physical Laboratory

Measurement Good Practice Guide No. 24

Finite Element Analysis of Piezoelectric Materials

Fiona Lowrie

Defence Evaluation Research Agency, Farnborough

Mark Stewart, Markys Cain & Mark Gee

Centre for Materials Measurement and Technology

National Physical Laboratory

Abstract:

This guide is intended to help people wanting to do finite element analysis of piezoelectric materials by answering some of the questions that are peculiar to piezoelectric materials. The document is not intended as a complete beginners guide for finite element analysis in general as this is better dealt with by the individual software producers. The guide is based around the commercial package ANSYS as this is a popular package amongst piezoelectric material users, however much of the information will still be useful to users of other finite element codes.

© Crown copyright 1999
Reproduced by permission of the Controller of HMSO

ISSN 1368-6550

May 1999

National Physical Laboratory
Teddington, Middlesex, United Kingdom, TW11 0LW

This guide was produced as part of the *Materials Measurement* programme, a programme of underpinning research supported by the Department of Trade and Industry and disseminated by the National Physical Laboratory.

For further information on *Materials Measurement* contact the Materials Enquiry Point at the National Physical Laboratory:

Tel: 020 8943 6701

Fax: 020 8943 7160

E-mail: materials@npl.co.uk

Finite Element Analysis of Piezoelectric Materials

Contents

1 Scope	1
2 Background theory	1
2.1 Purpose in performing finite element modelling	1
2.2 Introduction to finite element modelling	1
2.3 Process of performing finite element analysis	5
3 Frequently asked questions	6
3.1 What sort of checks might you do to see if the FEA package and your use of it are sound?	6
3.2 Can you demonstrate that FEA works for a very simple case which can be analytically modelled? What kind of errors between FEA and analytical model might be acceptable? ..	6
3.3 The ANSYS software package asks me to input a piezoelectric parameter [e] , however the manufacturer's data sheets only talk about a [d] matrix. How do I get these data?	7
3.4 The manufacturers data sheet has some data missing, where can I get more information to fill in the missing points?	7
3.5 Can I rely on the data in the manufacturers data sheets or should I measure the parameters myself?	8
3.6 Are there any simplifications/generalisations in FEA that are specific to piezoelectrics and piezo transducers that I should be made aware of?	9
4 The limitations of FEA packages with respect to piezoelectric materials.	10
4.1 Can they cope with dielectric and mechanical loss?	10
4.2 Can they cope with non linear piezoelectric behaviour?	10
4.3 Can they cope with temperature related changes in piezoelectric behaviour?	11
5 General FEA questions	11
5.1 What is the difference between static and modal analysis?	11
5.2 How might you simplify the model via symmetry etc?	11
5.3 How do you decide how fine the mesh needs to be?	11
5.4 What other packages are available to model piezoelectrics?	12
5.5 Bibliography and recommended reading?	12
6 References	13
Appendix	14
Miscellaneous Finite Element Related Web Resources	14
FEA Packages with Piezoelectric Capability	14
<i>ANSYS/ Multiphysics - ANSYS Inc.</i>	14
<i>Abaqus - Hibbitt, Karlsson & Sorensen.</i>	15

<i>Pafec - SER Systems Ltd</i>	15
<i>Pzflex - Weidlinger Associates</i>	16
<i>Atila - Magsoft Corp</i>	16
<i>Adina-T - Adina R & D, Inc</i>	17
An example of the d to e matrix conversion for a PZT-4 material using data from “Five Modern Piezoelectric Ceramics”	17
Bibliography	20

1 Scope

This guide is intended to help people wanting to do finite element analysis of piezoelectric materials by answering some of the questions that are peculiar to piezoelectric materials. The document is not intended as a complete beginners guide for finite element analysis in general as this is better dealt with by the individual software producers. The guide is based around the commercial package ANSYS as this is a popular package amongst piezoelectric material users, but much of the information will still be useful to users of other finite element codes.

2 Background theory

2.1 Purpose in performing finite element modelling

Finite element modelling is a technique which can be used to perform the analysis of continuum field and mechanics problems. By varying the refinement of the finite element model, any required degree of accuracy can be obtained within the limitations of the computing resources available.

Finite element techniques can be used to model problems involving continuum mechanics; the vibrational characteristics of structures, fluid mechanics, electromagnetic fields, electrostatics, and coupled piezoelectric analyses. It is this last area which is the subject of this good practice guide.

The essential advantage of the finite element method over analytical techniques is that it forms a universal tool which can be applied to all geometries of components and devices. It can cope with complex assemblies of different materials, and the analysis can be performed again and again with changes in the applied test conditions so that the sensitivity of results to alterations in conditions can be examined very easily.

The main drawback with the technique is that, for any particular problem, an analytical solution is not obtained. This means that, to get a new solution for altered conditions, the finite element model does need to be re-run. Also, like all computer programs, it can be very easy to generate poor finite element models which can give misleading and confusing results. The purpose of this guide is to give some advice on some of the common questions that arise in the finite element analysis of piezoelectric materials.

2.2 Introduction to finite element modelling

The fundamental basis of finite element analysis (FEA) is that any continuous function, such as stress, strain or electric field, can be approximated by discretisation. The original volume is

divided into elements, and within each element the function is constant or a simple function of position, either linear or quadratic. At each vertex of an element is a 'node' and the number of variables acting at each nodal site is called the 'degree of freedom' (DOF) [1].

For piezoelectric materials there are four DOF at each node; u_x , u_y and u_z (3D displacement) and voltage V . Corresponding to each DOF there is a reaction force F_x , F_y , F_z to the displacement and charge Q to the voltage V . In the case of piezoelectric analysis it is necessary to use 'coupled-field analysis' to couple the interaction between applied stress and electric field. ANSYS (commercially available FEA software) has three specific coupled field elements for piezoelectric analysis:

SOLID5 - 3D Coupled-field solid element

SOLID13 - 2D Coupled-field solid element

SOLID98 - Tetrahedral coupled-field solid element

The choice between these three coupled field elements is dependent upon the sample geometry being modelled. For example, a tetrahedral coupled-field solid element is more suited to dividing a 3D spherical body into elements, whereas SOLID5 constructs with cuboid elements and hence is used to discretise 3D cuboid bodies.

The electromechanical constitutive equations for linear behaviour of the elements are [2]:

$$\{T\} = [c]^E \{S\} - [e]^t \{E\} \quad \text{eqn. 1}$$

$$\{D\} = [e] \{S\} + [\epsilon]^S \{E\} \quad \text{eqn. 2}$$

where,

$\{T\}$ = stress vector

$\{D\}$ = electric flux density vector

$\{S\}$ = strain vector

$\{E\}$ = electric field vector

$[c]$ = elasticity matrix

$[e]$ = piezoelectric matrix

$[\epsilon]$ = dielectric matrix

The superscript E indicates a values at a constant field, and S at a constant strain. The subscript t indicates the transpose of the matrix.

Application of the variational principle of finite element discretisation to the coupled finite element discretisation yields the following equation.[3]

$$[M_{uu}]\ddot{u} + [C_{uu}]\dot{u} + \begin{bmatrix} K_{uu} & K_{u\phi} \\ K_{u\phi} & K_{\phi\phi} \end{bmatrix} u = \begin{bmatrix} F \\ Q \end{bmatrix} \quad \text{eqn. 3}$$

where

$[K_{uu}]$ = mechanical stiffness matrix derived from $[c]$ matrix

$[K_{u\phi}]$ = piezoelectric stiffness matrix derived from $[e]$ matrix

$[K_{\phi\phi}]$ = dielectric stiffness matrix derived from $[\epsilon]$ matrix

$[C_{uu}]$ = mechanical loss matrix

$[M_{uu}]$ = inertia matrix derived from density and volume

u = displacement vector

ϕ = voltage vector

F = mechanical force vector

Q = charge

Therefore to undertake finite element analysis and solve the above equations the following information is required:

- device geometry and loading conditions - this would define the quantities of stress, strain and electric field in equations 1 and 2
- material property matrices as required in equations 1 and 2 [3].

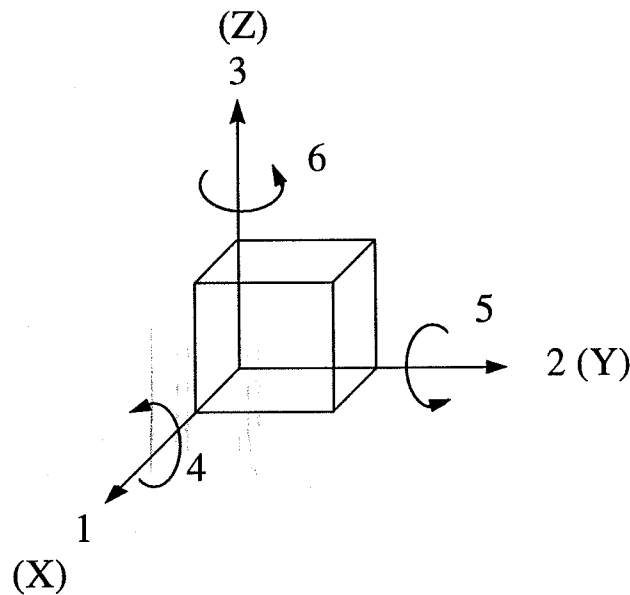


Figure 1: Axis Notation

In order to describe the anisotropy of piezoelectric ceramics the axis notation in figure 1 is commonly used. For poled ceramics the material symmetry is $6mm$ and the poling axis is taken to be in the Z or 3 direction. All the piezoelectric coefficients have two subscripts, the first identifies the direction of excitation, and the second the direction of the response. So for example the coefficient d_{31} in relation to figure 1 would refer to an excitation in the 3 (poled) direction, giving rise to a response in the 1 direction. A shear deformation around one of the three orthogonal axes would be represented by a subscript 4, 5 and 6. So the piezoelectric coefficient d_{15} represents a shear deformation around the 2 (Y) axis brought about by an excitation in the 1 direction (perpendicular to the poling direction). Since the 1 and 2 direction are equivalent all occurrences of the 2 axis can be replaced by the 1 axis, and due to the material symmetry conditions many of the coefficients for the $6mm$ case are zero, leading to the following :-

The dielectric matrix

$$\begin{bmatrix} \epsilon_{11} & 0 & 0 \\ 0 & \epsilon_{11} & 0 \\ 0 & 0 & \epsilon_{33} \end{bmatrix}$$

The piezoelectric matrix

$$\begin{bmatrix} 0 & 0 & d_{31} \\ 0 & 0 & d_{31} \\ 0 & 0 & d_{33} \\ 0 & d_{15} & 0 \\ d_{15} & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

The stiffness matrix

$$\begin{bmatrix} c_{11} & c_{12} & c_{13} & 0 & 0 & 0 \\ c_{12} & c_{11} & c_{13} & 0 & 0 & 0 \\ c_{13} & c_{13} & c_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & c_{44} & 0 \\ 0 & 0 & 0 & 0 & 0 & c_{66} \end{bmatrix}$$

2.3 Process of performing finite element analysis

Finite element modelling is performed by a multi-stage process. The different steps are:

- Definition of model
- Meshing
- Application of boundary conditions
- Optimisation
- Solution
- Presentation of results

In the definition of the model, all the information required for the solution to the problem is gathered together including knowledge of the geometry and properties of the constituent materials of the component or device, any boundary conditions such as applied forces and voltages (for piezoelectric analysis) or geometrical constraints.

Meshing is the process of defining the subdivision of the model into the discrete elements of the appropriate element type. This is an extremely important step, which can adversely affect the efficiency, and in some cases the accuracy of the analysis if it is carried out incorrectly. As the accuracy of the results depends on the size of the elements in the mesh, it is important to refine the mesh in the areas of particular interest or where known concentrations in the variables occur. It is also important to try to obtain elements with an optimised shape. Here, the more equiaxed the shape of an element the better. It is also important to place nodes and element boundaries at positions where known boundary conditions will be applied.

The process of meshing is normally aided considerably by the FE package and can often be carried out completely automatically. However, a good knowledge of the processes involved can be beneficial in guiding the process to an efficient and accurate solution.

The application of the boundary conditions is normally straightforward, but sometimes care needs to be taken in defining exactly what is required. Particular care needs to be taken in defining applied pressures. Different FE systems often have different rules for the application of pressures across the face of an element or a group of elements and it is important that the correct interpretation is used.

Optimisation of the model is an automatic process where the set of simultaneous equations that defines the model is rearranged to minimise the bandwidth of the set of arrays and thus reduce the processing time to the minimum required.

Solution of the model is performed using normal numerical techniques.

The results of the analysis are calculated on a node by node basis for each element in the model. Since each node can belong to more than one element, the different values of parameters at a particular node associated with different elements can be different. The magnitude of this difference is a measure of the errors in the analysis. A process of reconciliation is then carried out where a common value of parameters is derived for each node.

The results are then displayed by a graphics program. These normally have the facilities for displaying results as two or three dimensional contour maps, and can also show the variation of variables along paths specified by the user. Deformation of the device or component is shown directly in a plot where the deformed shape is overlaid on the original shape, often with a specified magnification in scale for the deformation. Where deformation changes in a well defined way with time, for example when the structure vibrates, a sequence of frames of deformation at different times in the process can be generated as a movie file.

3 Frequently asked questions

3.1 What sort of checks might you do to see if the FEA package and your use of it are sound?

To validate a finite element (FE) modelling approach, a direct comparison between experimental data, analytical calculations, and finite element predictions should be performed. This analysis can be performed in a variety of ways, but the most straightforward would be a measure of strain developed under an applied field, charge developed in response to an applied stress (closed circuit conditions), or comparison of admittance spectra. If a dynamic model is required it is often useful to examine the static solution first, as this might highlight problems that could otherwise take much processing time to discover.

3.2 Can you demonstrate that FEA works for a very simple case which can be analytically modelled? What kind of errors between FEA and analytical model might be acceptable?

Before any FE model is generated incorporating a new piezoelectric material, simple structures are constructed and the tests mentioned in (1a) are performed. These are normally performed on a simple cube of the piezoelectric material because of the faster solution speeds associated with quadratic mesh elements.

Experience has shown that under these simple conditions, deviation between analytical and finite element models would not be expected to exceed 5% for calculation of static strain and the frequencies of natural modes of vibration. It is worth noting that an increase in mesh density is required to maintain accurate predictions of high frequency material response

3.3 The ANSYS software package asks me to input a piezoelectric parameter [e] , however the manufacturer's data sheets only talk about a [d] matrix. How do I get these data?

The ANSYS FE code requires a description of piezoelectric activity in the form of the piezoelectric stress constant matrix, [e] and is expressed in units of coulomb/metre. In general, transducer material manufacturers give a measure of activity of piezoelectric materials in terms of the piezoelectric charge or strain coefficient matrix, [d]. This can be expressed in terms of either coulombs/newton or metres/volt depending on whether the material is to be used as an actuator or a sensor. The [e] coefficient of a material is a measure of the stress generated by the transducer element per unit of applied electric field, under constant strain conditions. If direct experimental characterisation of this matrix is not possible, then it is necessary to utilise matrix manipulation to convert the data. For example, the [e] matrix can be obtained if the full compliance matrix, [s], and piezoelectric voltage matrix, [d], are known, by firstly inverting the compliance matrix to give the stiffness matrix, [c], which is then multiplied by the [d] matrix to give the desired [e] matrix. This is expressed mathematically as:

$$[e] = [c][d]$$

It must be noted that manipulation of matrices compounds errors gained from direct material measurements - and must therefore be considered an inferior approach compared with direct experimental derivation. It is also worth noting that the order of the ANSYS matrices for stiffness, compliance and piezoelectric coefficients do not conform to the international standard, with rows 4, 5, and 6 in each matrix needing to be rearranged (see ANSYS User Manual, Revision 5.0, ch. 8, p. 30) . However, these coefficients are only required for systems involving significant magnitude shear vectors. For an example of how to carry out this procedure please refer to the appendix.

3.4 The manufacturers data sheet has some data missing, where can I get more information to fill in the missing points?

Although manufacturer data for a material are usually satisfactory, situations can arise where some data from the matrices are missing, such as during a fatigue test where test sample geometry limits the coefficients which can be obtained experimentally. It is feasible to replace

missing material data by selecting data from materials of a similar class. Before this route is undertaken an evaluation should be conducted of the dependence of the structure behaviour on the unknown coefficient. For example, in the report 'Finite Element Modelling of Electroceramics'[CMMT(A)150, January 1999] it was found that the d_{33} value had no effect on the extent of deflection of a bimorph transducer structure and hence an accurate d_{33} value was not necessary. For insertion of nominal data, a sensitivity analysis should be carried out wherein individual coefficients used in the model are altered in magnitude by realistic amounts to allow insight into whether or not any coefficient is critically influencing the predicted (modelled) performance. In carrying out the sensitivity analysis the variability of the material must also be considered, the following question gives some ideas on the likely spread of results.

3.5 Can I rely on the data in the manufacturers data sheets or should I measure the parameters myself?

The complete measurement of all the parameters required for a finite element analysis is probably beyond the scope of many users of piezoelectric materials, and so as a first approach it is easiest to use the data in the manufacturers data sheets. However in considering the reliability of the manufacturers data it must be born in mind that the data contained in the data sheet is an average measurement probably taken from a batch of material measured some time before the data sheet was issued. It is possible that since those measurements were made the manufacturer will have changed his process leading to different material characteristics. Most manufacturers will perform some simple measurements on the material, such as capacitance and loss, but the cost involved in making the full set means that this is not done on every batch. Of course for a cost either the manufacturer or an independent measurement laboratory can carry out the required measurements.

The variability of the materials is rarely quoted, but can be up to $\pm 20\%$ for some of the parameters. To get some estimate it is worth comparing the measurements of the complete range of a single manufacturer. For instance the density of a broad range of PZT materials do not vary much more than a few percent, so this is likely to be the case for a particular batch, whereas the d_{33} value can range from 100 to 600 indicating that this parameter will vary more from batch to batch. Of course as suggested previously it is always wise to carry out some sort of sensitivity analysis in the FE coding to see exactly what parameters are important for each particular case. It must be stressed that the preceding discussion is not a substitute for incomplete data, or actual measurements but merely as a guide in material selection exercises.

3.6 Are there any simplifications/generalisations in FEA that are specific to piezoelectrics and piezo transducers that I should be made aware of?

In devices there are hardware features in addition to the piezoelectric material which must be modelled, for example adhesive layers and electrodes. In structures where different sections of materials are joined together by means of an adhesive layer, the programmer has two options. The more elaborate route is to include the adhesive layer in the model with its associated dimensions and material properties; this may substantially increase solution time if the adhesive layer is thin, because comparable mesh length and width dimensions should be maintained throughout the model. The simpler option involves making the assumption that the adhesive layer has negligible influence. If the piezoelectric volumes are merged (the sharing of keypoints, lines and areas of the adjacent surfaces in the model), an infinitely thin adhesive layer is created.

In many applications the adhesive and its properties play a large part in the device response and more effort must be aimed at including it in the model. In order to get some idea of the effect of the adhesive an FE model neglecting the electrodes could be compared to the real behaviour. Any deviations could be attributed to the losses in the adhesive, and by appropriate tuning of the material parameters to match real behaviour, a set of modified materials parameters that included the adhesive behaviour could be obtained.

Another approach for many layered structures is to combine all the adhesive layers into one thick layer. This includes the stiffness of the adhesive layer, and reduces computation, however the spatial arrangement of the device is then altered which may affect the dynamic performance of the model.

The same considerations which were outlined for the adhesive layer apply to the electrodes, with the complexity of the solution being dependent upon the precision required and the nature of the model. When modelling high modulus ceramics, such as PZT, it is considered that a thin layer of electrode material is of little consequence; therefore when modelling electrodes a plane (termed a 'coupled-plane') with no physical properties is created on the surfaces of the piezoceramic, allowing an applied voltage to be constant across the entire material surface.

4 The limitations of FEA packages with respect to piezoelectric materials.

4.1 Can they cope with dielectric and mechanical loss?

It is generally considered that ANSYS does not have the ability to simulate dielectric and mechanical losses; however, it is possible to generate a damping matrix, which, although having no direct correspondence to the physical mechanisms responsible for the energy dissipation, can be used to more accurately simulate the 'lossy' effects. This damping matrix can be extended to include a frequency dependent term. Success of this technique is highly dependent on the experience of the programmer.

4.2 Can they cope with non linear piezoelectric behaviour?

The most important limitation in ANSYS models is the assumed linear coupling of the electric field to strain, explicitly defined by the use of constant coefficients of the matrices [c] and [e]. Modelling of non-linear behaviour, arising in a piezoelectric material under large stresses or electric fields, can be achieved by manually adjusting the coefficients in these matrices. Automation of this adjustment is non-trivial and requires the use of the ANSYS command language (ADPL). The coupling that exists between the stimulus causing the non-linear behaviour and the effects of the non-linearities on material parameters means that the solution must often proceed in an iterative manner. This process was demonstrated in the report 'Time Dependent Behaviour of Electroceramics'[CMMT(A)151, February 1999], where the high electric fields used to produce a deflection in a piezoelectric bimorph, and the consequent internal stress distribution, caused a non-linear variation in the piezoelectric coefficient, d_{31} . An initial measure of internal stresses was obtained using a low field d_{31} value. Then an iterative process began involving adjustment of the d_{31} value until it was in accordance with the value measured experimentally at correspondingly high field levels. This d_{31} change resulted in a new stress prediction by the model, which was used to refine the d_{31} value further. This process was repeated until the value of d_{31} remained constant for repeat iterations.

Another similar approach was used in [4] to try and model the effect of non linear behaviour on the charge output of stressed piezoelectric. Here a stress map of a simple shape which was loaded in a number of ways. In ANSYS the d_{33} value is assumed to be a constant for all boundary conditions, however in this case the d_{33} was found to be dependant on the stress level. Using a linear relationship between applied stress and d_{33} the charge developed at each node was determined and because of the conservation of charge this was summed for all nodes to give the total charge output.

4.3 Can they cope with temperature related changes in piezoelectric behaviour?

A change in material properties due to a temperature variation can be incorporated by adjusting the material properties in the finite model to values acquired from characterising the bulk piezoelectric material at the same temperatures. There is no direct coupling in the modelling between electrical loss and temperature rise in the piezoelectric material and so for modelling purposes, it is normally assumed that the device will be operated at a duty cycle low enough to avoid notable temperature rises. Under extreme circumstances, where the material is being driven at high electrical loads or is thermally isolated this assumption may not be valid.

5 General FEA questions

5.1 What is the difference between static and modal analysis?

Many types of structural analysis are available in finite element packages. In piezoelectric studies two of the most commonly used are the static analysis and the modal analysis. The static analysis is used to determine the displacements, stresses, strains and forces in structures or components due to loads that do not induce significant inertia or damping effects. Modal analysis is used to determine the natural frequencies and mode shapes of a structure or component.

5.2 How might you simplify the model via symmetry etc?

Many objects have some kind of symmetry, be it repetitive symmetry (such as evenly-spaced cooling fins on a long pipe), reflective symmetry (such as a moulded plastic container), or axisymmetry (such as a light bulb). When an object is symmetric in all respects (geometry, loads, constraints and material properties), this can often be taken advantage of to reduce the size and scope of the model, thus requiring less computer resources and providing solutions in a shorter time-scale.

5.3 How do you decide how fine the mesh needs to be?

One of the most frequent uncertainties when developing a model is how fine to make the element mesh in order to obtain good results. Unfortunately, there is no general answer to this question, but there are a few techniques used to evaluate models. Adaptive meshing is commonly used in structural analysis to generate a mesh which meets acceptable 'Energy

Error Estimate'¹ criteria, but this technique cannot be used for coupled field (piezoelectric) analysis. Comparing the results of a preliminary analysis with independently derived experimental or known accurate analytical results is another acceptable method of mesh evaluation. Then by refining the mesh in regions where the discrepancy between known and calculated results is too large, confidence in the mesh density can be obtained. Another effective technique is to perform an iterative meshing process. An initial analysis using what is considered to be a reasonable mesh is performed, followed by a re-analysis of the problem using twice as many elements in critical regions. If comparison between the two solutions yields very similar results, then the original mesh is probably adequate. If the two meshes yield substantially different results, then further refinement is required. This process should then be performed in an iterative manner until an increase in mesh density produces results nearly identical to the preceding model. It cannot be over-emphasised how important it is to use an appropriate mesh density; if the mesh is too coarse, the results can be seriously in error, yet if the mesh is too fine, computer resources will be wasted, excessively long run-times experienced, and it might even be found that the model is too large to run on the computer system in use. An initial estimate of the mesh size could be speeded up by running on a simplified model of the device, however the final model would need to be examined again as more complex geometry/elements are introduced.

5.4 What other packages are available to model piezoelectrics?

This good practice guide concentrated mainly on the use of ANSYS in the analysis of piezoelectric modelling, because it is a widely used finite element package for this purpose in the UK. However, there are a number of other existing finite element codes offering coupled field piezoelectric modelling, including PAFEC, PZFLEX and ATILA. For more information please refer to the appendix.

5.5 Bibliography and recommended reading?

The most comprehensive source of information as to how to model piezoelectrics is in the ANSYS manuals supplied on purchase of the package. Also, please refer to compilation of web related resources and extended bibliography in the appendix.

¹ Energy Error Estimate is the sub-programme within ANSYS which is used to obtain guidance for mesh geometries employed.

6 References

- [1] ANSYS Coupled-field Analysis Guide, version 5.5
- [2] 'Electroceramics'; A. J. Moulson & J. M. Herbert, 1991
- [3] 'Finite element for piezoelectric vibration'; H. Allik & J. R. Hughes, International Journal of Numerical Methods of Engineering, no. 2, 1970
- [4] 'Finite Element Modelling of Electroceramics'[CMMT(A)150, January 1999]
- [5] 'Time Dependent Behaviour of Electroceramics'[CMMT(A)151, February 1999]

Appendix

Miscellaneous Finite Element Related Web Resources

<http://www.flinthills.com/~ramsdale/EngZone/design.htm>

The Engineering Zone, information about engineering software including CAD, CAM and FEA links.

<http://femur.wpi.edu/>

FEMur, finite element method universal resource, some interactive tutorials and general FEA links.

http://www.engr.usask.ca/~macphed/finite/fe_resources/fe_resources.html

A comprehensive list of FEA related resources.

http://www3.sympatico.ca/peter_budgell/

Introduction to FEA and some ANSYS tips.

Finite Element Related Newsgroup: *Sci.Engr.Analysis*

XANSYS News The XANSYS Listserver.

To subscribe to this mailing list, send an email to subscribe-xansys@onelist.com. For those not familiar with mailing lists this is a discussion group carried out by e-mail. It can generate a large number of e-mails in the users e-mail account. Alternatively the archive of the discussion can be read retrospectively, this is kept at <http://www.infotech.tu-chemnitz.de/~thieme/ansys/ansys.html>.

FEA Packages with Piezoelectric Capability

ANSYS/ Multiphysics - ANSYS Inc.

ANSYS/ Multiphysics is the package that contains the piezoelectric elements, and there is a reduced element version called ANSYS/ED.

ANSYS Europe Ltd
Suite 2
Wyvols Court
Swallowfield
Reading
RG7 1PY
Tel: (44) 118 9880229
Fax: (44) 118 9880925
<http://www.ansys.com>

Abaqus - Hibbitt, Karlsson & Sorensen

ABAQUS/Standard is a general-purpose, production-oriented finite element program designed specifically for advanced analysis applications. As well as the standard static, transient stress analyses, it can also deal with coupled problems including piezoelectric elements.

HIBBITT, KARLSSON & SORENSEN (UK) LTD.

The Genesis Centre
Science Park South, Birchwood
Warrington, Cheshire WA3 7BH
Tel: (44) 1 925 810166
Fax: (44) 1 925 810178
E-mail: hotline@hks.co.uk
<http://www.abaqus.com>

Pafec - SER Systems Ltd

PAFEC-FE offers wide range of element types and solution alternatives will allow models to be created for static, dynamic, non-linear, thermal and acoustic conditions.

SER Systems Ltd.
39 Nottingham Road
Stapleford
Nottingham
NG9 8AD
Tel: +44 (0)115 935 7060
Fax: +44 (0)115 935 7064
Email: marketing@SERuk.com
<http://www.pafec.com>

Pzflex - Weidlinger Associates

PZFLEX is a time domain finite element program for solving the coupled mechanical-piezoelectric-acoustic equations. It is specialized for piezoelectric, ultrasonic, and bioengineering wave propagation problems.

Weidlinger Associates
375 Hudson Street
12th Floor
New York, NY 10014-3656
USA
(212) 367-3000 (Voice)
(212) 367-3030 (Fax)
<http://www.wai.com/Contact/staff.html>

Atila - Magsoft Corp.

Atila is a finite element program that aids in the design of structures that contain elastic, piezoelectric, magnetostrictive, and fluid materials (combined or not). Atila can perform static, modal, harmonic, and transient analyses of active structures.

Cedrat S.A.
10, Chemin de Pré Carré ZIRST
38246 Meylan, France
Phone: (33) 4 76 90 50 45
Fax: (33) 4 76 90 16 09
<http://www.magsoft-flux.com/>

Adina-T - Adina R & D, Inc.

The ADINA-T program is used for linear and nonlinear heat transfer analysis. The major ADINA-T capabilities include: 2-D, 3-D and shell steady-state and transient analysis, conductivity, convection and radiation, latent heat effects, thermal eigenvalue calculations and electrostatic, seepage, piezo-electric and soil consolidation analysis.

Product Development Services Limited

21A Leslie Square

Paper Mill End

Great Barr, Birmingham

B44 8NH United Kingdom

Tel: 44 (121) 356 7509

Fax: 44 (121) 356 7529

<http://world.std.com/~adina/>

An example of the d to e matrix conversion for a PZT-4 material using data from “Five Modern Piezoelectric Ceramics”

The Vernitron bulletin “Five modern Piezoelectric Ceramics” (revised January 1976), is one of the few manufacturers data sheets to give a complete listing for the [d], [e], [c] and [s] matrices for several materials. Consequently, this data set is often used in FEA of piezoelectric materials, even though the data in these sheets is at least 23 years old, and the company under that name no longer exists. This data sheet can still be obtained from the Morgan Electro Ceramics, Transducer Products Division in Southampton.

The [d] and [s] matrix from the data tables are as follows

$$d \text{ matrix} = \begin{bmatrix} 0 & 0 & -123 \\ 0 & 0 & -123 \\ 0 & 0 & 289 \\ 0 & 496 & 0 \\ 496 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} * 10^{-12} \text{ metres / volt}$$

$$s \text{ compliance matrix} = \begin{bmatrix} 12.3 & -4.05 & -5.31 & 0 & 0 & 0 \\ -4.05 & 12.3 & -5.31 & 0 & 0 & 0 \\ -5.31 & -5.31 & 15.5 & 0 & 0 & 0 \\ 0 & 0 & 0 & 39 & 0 & 0 \\ 0 & 0 & 0 & 0 & 39 & 0 \\ 0 & 0 & 0 & 0 & 0 & 32.7 \end{bmatrix} * 10^{-12} \text{ metre}^2 / \text{volt}$$

The s matrix can be inverted to give:-

$$\text{c stiffness matrix} = \begin{bmatrix} 139 & 7.8 & 7.4 & 0 & 0 & 0 \\ 7.8 & 139 & 7.4 & 0 & 0 & 0 \\ 7.4 & 7.4 & 115 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2.6 & 0 & 0 \\ 0 & 0 & 0 & 0 & 2.6 & 0 \\ 0 & 0 & 0 & 0 & 0 & 31 \end{bmatrix} * 10^{10} \text{ newton / metre}^2$$

and using $[e] = [c][d]$ this gives :-

$$\text{e matrix} = \begin{bmatrix} 0 & 0 & -5.2 \\ 0 & 0 & -5.2 \\ 0 & 0 & 15.08 \\ 0 & 12.72 & 0 \\ 12.72 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \text{coulomb / metre}^2$$

As stated before in order to enter the above stiffness and piezoelectric matrix into ANSYS the last three rows must be cycled from the normal IRE layout above to give the following:

$$\text{c stiffness matrix} = \begin{bmatrix} 139 & 7.8 & 7.4 & 0 & 0 & 0 \\ 7.8 & 139 & 7.4 & 0 & 0 & 0 \\ 7.4 & 7.4 & 115 & 0 & 0 & 0 \\ 0 & 0 & 0 & 31 & 0 & 0 \\ 0 & 0 & 0 & 0 & 2.6 & 0 \\ 0 & 0 & 0 & 0 & 0 & 2.6 \end{bmatrix} * 10^{10} \text{ newton / metre}^2$$

$$\text{e matrix} = \begin{bmatrix} 0 & 0 & -5.2 \\ 0 & 0 & -5.2 \\ 0 & 0 & 15.08 \\ 0 & 0 & 0 \\ 0 & 12.72 & 0 \\ 12.72 & 0 & 0 \end{bmatrix} \text{coulomb / metre}^2$$

As a consequence to input the data into tables for a 3D analysis the following commands would be needed.

```
TB,PIEZ,1,,0
TBMODIF,1,3,-5.2
TBMODIF,2,3,-5.2
TBMODIF,3,3,15.08
TBMODIF,5,2,12.72
TBMODIF,6,1,12.72
```

TB,ANEL,1,,,0
TBMODIF,2,1,13.9E10
TBMODIF,3,1,7.8E10
TBMODIF,4,1,7.4E10
TBMODIF,8,1,13.9E10
TBMODIF,9,1,7.4E10
TBMODIF,13,1,11.5E10
TBMODIF,17,1,3.1E10
TBMODIF,20,1,2.6E10
TBMODIF,22,1,2.6E10

Bibliography

“Physical Properties of Crystals”, J F Nye, Oxford University Press, 2nd edition 1985.

A good introduction to the matrix notation and operations needed for piezoelectric materials.

“A Review on the Modeling of Piezoelectric Sensors and Actuators Incorporated in Intelligent Structures”, C Y K Chee, L Tong and G P Steven, J. of Intelligent Material Systems and Structures, Vol 9, Jan 1998, 3-19.

A general review of finite element methods used on piezoelectrics, with a comprehensive list of references. Not specific to ANSYS.

“Finite-Element Modeling of Ferroic Domain Switching in Piezoelectric Ceramics”, T Steinkopff, Journal of the European Ceramic Society, vol 19, 6-7, 1999, 1247-1249.

Uses ANSYS to explore the effect of grain to grain interaction on domain switching of ferroelectrics.

“Finescaled Piezoelectric 1-3 Composites: Properties and Modeling” R Steinhausen, T Hauke, W Seifert, H Beige, W Wattzka, S Seifert, D Sporn, S Starke, and A Shoenecker, Journal of the European Ceramic Society, vol 19, 6-7, 1999, 1289-1293.

Uses ANSYS to model the piezoelectric and dielectric behaviour of 1-3 composites. Comparison of the FEA permittivity with that from the rule of mixtures.

“Fatigue Life Predictions of PZT using Continuum Damage Mechanics and Finite Element Methods”, T Chung, Z Wang, M Hill and G White. Fracture of Ceramics vol 12, 1996, Plenum Press, 135-148.

Develop fatigue life prediction algorithms for piezoelectric materials for use in ANSYS.

“Finite Element Analysis of 1-3 Composite Transducers”, J Hossack and G Hayward, IEE Transactions on Ultrasonics, Ferroelectrics and Frequency Control, vol 38, 6, 1991, 618-629.

Using ANSYS to model the behaviour of 1-3 composites.

“Using Ansys5.0 to Analyze Piezoelectric Materials”, Diane Hageman, Delco Remy, document available from ANSYS.

Some useful guidelines for ANSYS users starting out in piezoelectric analysis.

VM175 “Frequency Response of Electrical Input Admittance for a Piezoelectric Transducer” and VM176 “Natural Frequency of a Piezoelectric Transducer”.

These are two piezoelectric examples included in the ANSYS verification manual.

“Piezoelectric Analysis of a Beam Resonator”. ANSYS Examples Supplement for Revision 5.1.

ENERGY

