COUPLED FINITE ELEMENT ANALYSIS OF A PIEZO-CERAMIC FORCE TRANSDUCER

Behrouz Shiari and Peter M. Wild
Department of Mechanical & Aerospace Engineering, Carleton University, Ottawa, ON, Canada
Contact: bshiari@mae.carleton.ca
Department of Mechanical Engineering, University of Victoria, Victoria, BC, Canada

Received August 2005, Accepted March 2006
No. 05-CSME-50, E.I.C. Accession 2902

ABSTRACT

The coupled finite element method is used to design a novel miniaturized piezoelectric force transducer for the pulp refiners. The analysis focuses on a proposed transducer design using commercially available piezoelectric sensing element and a steel assembled housing. The software ANSYS was used to create a finite element model in order to improve transducer electromechanical behaviour. The primary goal of modeling is to design a transducer with a frequency response higher than a lab-scale refiner. Thus, modal and harmonic responses of the transducer model are studied. A coupled electromechanical model is used for prediction the output of the transducer. In order to enhance the performance of the transducer, the design is studied with respect to output magnitude and a flat frequency response. The modeling results are used for fabrication a prototype force transducer. The experimental results showed good agreement with the modeling.

ANALYSE D'UN TRANSDUCTEUR DE FORCE PIEZO-CÉRAMIQUE EN UTILISANT LA METHODE D'ÉLÉMENTS FINIS COUPLÉS

RESUME

La méthode d'éléments finis couplés est utilisée pour produire un nouveau type de transducteur de force miniaturisé pour le raffinage de la pâte à papier. L'analyse se concentre sur un modèle particulier de transducteur utilisant un senseur piézoélectrique commercial et un boîtier composite en métal. Un modèle d'éléments finis a été crée en utilisant le logiciel ANSYS pour améliorer le comportement électromécanique du transducteur. Le premier objectif de la modélisation est de produire un transducteur ayant une réponse en fréquence plus grande qu'un raffineur à l'échelle du laboratoire. En conséquence de quoi, les réponses modales et harmoniques de ce modèle de transducteur sont analysées. Un modèle électromécanique couplé est utilisé pour prédire le rendement de ce transducteur. Pour améliorer les performances du transducteur, la conception est étudiée en fonction l'efficacité du rendement et de la réponse en fréquence plate. Les résultats du modelage sont utilisés pour la fabrication d'un prototype de transducteur de force. Les résultats expérimentaux sont bien reproduits par ce modelage.

INTRODUCTION

Piezoelectric materials when subjected to an electric field generate mechanical strain or alternately create an electric charge when subjected to a mechanical strain. This property gives piezoelectric materials the ability to act as actuators and transducers. The increasing engineering activities in the development and industrial application of piezoelectric structures require effective and reliable simulation and design tools [**1**]. In the past, the development of piezoelectric transducers was based on experimental investigations that involved several design and fabrication cycles until the optimal specifications were satisfied [**2**]. It is obvious that modeling can change the trial and error procedure to a shorter development cycle and a time-saving approach [**3**]. Unfortunately, the major technical difficulty in conducting a successful simulation of transducers is the lack of understanding of interactions between structural and electrical fields in piezoelectric systems. It is not enough to understand these areas individually, but capturing the coupled effects of these phenomena at the same time makes a step in advance in the transducer design. This is possible by using coupled-field analyses that take into account interactions between both structural and electrical fields.

Coupled analytical solutions of piezoelectric device performance generally examine simple shapes under one-dimensional static load or at resonance conditions. Analytical solutions of complex geometries often involve assumptions which simplify the stress state and electric field distributions within the piezoelectric device. Regularly, this leads to inaccurate predictions of the observed response [**4**]. These complexities forced researchers to pursue numerical methods, in particular, the finite element method (FEM). The advantage of FE analysis is that the strain and electrical fields distributions throughout the device with a complex geometry can be calculated. Even though initial applications of FEM for piezoelectric material configurations can be traced back to three decades ago, the field is still in progress [**5**]. Over the past decade, significant progress has been made in the development of finite elements for piezoelectric applications, but most of these developments are limited to application of commercial finite element codes, such as NASTRAN, ABAQUS and ANSYS [**6**]. The new piezoelectric finite element capability available in these commercial codes makes them very powerful tool in an integrated process of designing, prototyping and testing a transducer.

The aim of this study is to use the capability of coupled field analysis of ANSYS to model the behaviour of a novel dynamic piezoelectric force transducer. The details construction of a coupled FE model for piezoelectric material is described. To validate the model, the modal and harmonic response of force transducer is experimentally measured and compared to that predicted by FE model.

BACKGROUND THEORY

The finite element technique approximates any continuous function, such as stress, strain or electric field, by discretisation the original volume into finite elements within which the function is approximated by a polynomial. To perform finite element analysis involving piezoelectric effects requires coupled field elements that take into account structural and electrical coupling. At each vertex of an element is a node with which, the "degrees of freedom" (DOF) are associated. The coupled-field elements should contain all the necessary nodal degrees of freedom for electrical-structural coupling. In a simple coupled-field element, there are four DOF at each node, three displacement components u_i $(i = 1,2,3)$ and voltage. Corresponding to each displacement DOF is a reaction, F_i $(i=1,2,3)$, and corresponding to the voltage is a charge, Q[4].

The electromechanical constitutive relation for linear behaviour of a piezoelectric continuum consists of two fundamental equations: (1) the electric displacement (D_n) equation, and (2) the Duhamel-Neumann stress equation (T_{ij})[9]:

$$D_n = e_{nkl}(S_{kl} - S_{kl}^0) + \varepsilon_{nm}^S E_m \tag{1}$$

$$T_{ij} = c_{ijkl}^E (S_{kl} - S_{kl}^0) + e_{mij} E_m \tag{2}$$

In these equations, e_{nkl} are the piezoelectric coefficients; S_{kl} are the strains; S_{kl}^0 are the initial strains; ε_{nm}^S are the dielectric constants or permittivities; E_m are the electric fields; and c_{ijkl}^E are the elastic constants. In addition, the superscripts, $(.)^E$, $(.)^S$, denote the coefficients defined at a constant electric field and strain, respectively. The superscript $(.)^0$ denotes initial value.

Equations (1) and (2) can also be written in terms of displacement components and voltage. The strain S_{ij} and electric field equation E_i are respectively defined by the gradient of displacement u_i and electrical potential ϕ.

$$S_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}) \tag{3}$$

$$E_i = -\phi_{,i} \tag{4}$$

Application of the variational principle of finite element discretisation to the coupled finite element yields the following equations [10].

$$c_{ijkl}^E (u_{k,lj} - u_{k,lj}^0) + e_{kij}\phi_{,kj} + f_{b_i} = \rho \ddot{u}_i \tag{5}$$

$$e_{ikl}(u_{k,li} - u_{k,li}^0) - \varepsilon_{ik}^S \phi_{,ki} = 0 \tag{6}$$

The boundary conditions at surfaces are:

(a) Boundary displacement and force:

$$u_i = \overline{u_i} \quad , \; T_{ij} l_j = f_i \tag{7}$$

(b) Electrical boundary conditions:

$$\phi = \overline{\phi}\,, \; D_i l_i = -Q \tag{8}$$

where f_i and Q are surface force on a unit surface and surface charge, respectively. $(\bar{.})$ denotes a known boundary value. l_i are the direction cosine components.

Finite element discretization of these relations is performed by establishing element shape functions over an element domain which approximate the exact solution. Assuming the shape functions, one can define all field variables in matrix notations

$$\{u\} = [N_u]^T \{U\}$$
$$\phi = [N_\phi]^T \{\Phi\}$$

where $[N_u]$, and $[N_\phi]$ are the shape function matrices for the nodal displacement vector $\{U\}$, and nodal potential vector $\{\Phi\}$.

Then the strain vector $\{S\}$and electrical field $\{E\}$ are related to the displacements and potentials, respectively, as:

$$\{S\} = [B_u]\ \{U\}$$
$$\{E\} = -[B_\phi]\ \{\Phi\}$$

where $[B_u] = [L_u]\{N_u\}^T$, and $[B_\phi] = [L_\phi]\{N_\phi\}^T$, and L is a differential operator.

After the application of the finite element discretization and assembling all the element matrices, one can derive the global system equation as (in a compact matrix form)[**10**]:

$$\begin{bmatrix} M_{uu} & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \ddot{U} \\ \ddot{\Phi} \end{bmatrix} + \begin{bmatrix} C_{uu} & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \dot{U} \\ \dot{\Phi} \end{bmatrix} + \begin{bmatrix} K_{uu} & K_{u\varphi} \\ K_{\varphi u} & K_{\varphi\varphi} \end{bmatrix} \begin{bmatrix} U \\ \Phi \end{bmatrix} = \begin{bmatrix} F_u \\ F_\phi \end{bmatrix} \qquad (9)$$

where a dot above a variable denotes a time derivative. M_{uu} is mass matrix, K_{uu} is mechanical stiffness matrix, $K_{u\varphi}$(or $K_{\varphi u}$) is the piezoelectric stiffness matrix, $K_{\varphi\varphi}$ is the dielectric stiffness matrix, C_{uu} is damping matrix, F_u is vector of nodal forces, surface forces, and body forces and F_ϕ is applied nodal charge vector. Equation (9) clearly shows the coupling of displacement and electric fields.

PIEZO-CERAMIC MATERIAL DATA

Finite element modelling cannot predict piezoelectric device performance unless the piezoelectric element's material properties are known. Accuracy of the finite element model also hinges on the accuracy of the material constitutive properties and those provided in manufacturer specification sheets are often incomplete or inaccurate. In addition, most manufacturers of piezoelectric materials do not publish the material properties in a format that can be readily entered into commercial FE codes. The published data must be converted in order to populate the necessary material matrices that define the piezoelectric material input data [**11**]. For example, the commercial FE code, ANSYS 5.6, requires a dielectric matrix [ε] (consisting of dielectric constants or permittivities), a piezoelectric matrix [e] (consisting of piezoelectric constants), and either a compliance matrix [d] or a stiffness matrix [c] (consisting of elastic constants). These matrices correspond to the $K_{u\varphi}$, $K_{\varphi\varphi}$ and K_{uu} matrices in equation (9).

For the piezoelectric elements, care must be taken when defining the material properties as they are not isotropic. The alignment of the piezoelectric poling axis is determined by the how piezoelectric element constants are entered into the piezoelectric, dielectric and stiffness matrices. For example, consider the piezoelectric element shown in Figure 1. ANSYS requires that the z-axis correspond to the poling axis, which is generally referred to as axis 3 in literature pertaining to piezo-ceramics. In the event that the system to be modeled cannot be oriented so that axis 3 of all piezo elements coincides with the z-axis, a macro can be written to convert the manufacturers data to desired local coordinate system.

The piezoceramic material used in this application (see section 4.0) was PKI-502, manufactured by Piezo Kinetics Inc. This material, known generically as Navy Type II, is an appropriate choice for a high electromechanical activity application such as a force transducer [**12**]. This material maintains low dielectric losses, high dielectric constant and high resistance to depolarization under large mechanical loads and high electric voltages. The PKI-502 piezoelectric element data is reported in the Appendix.

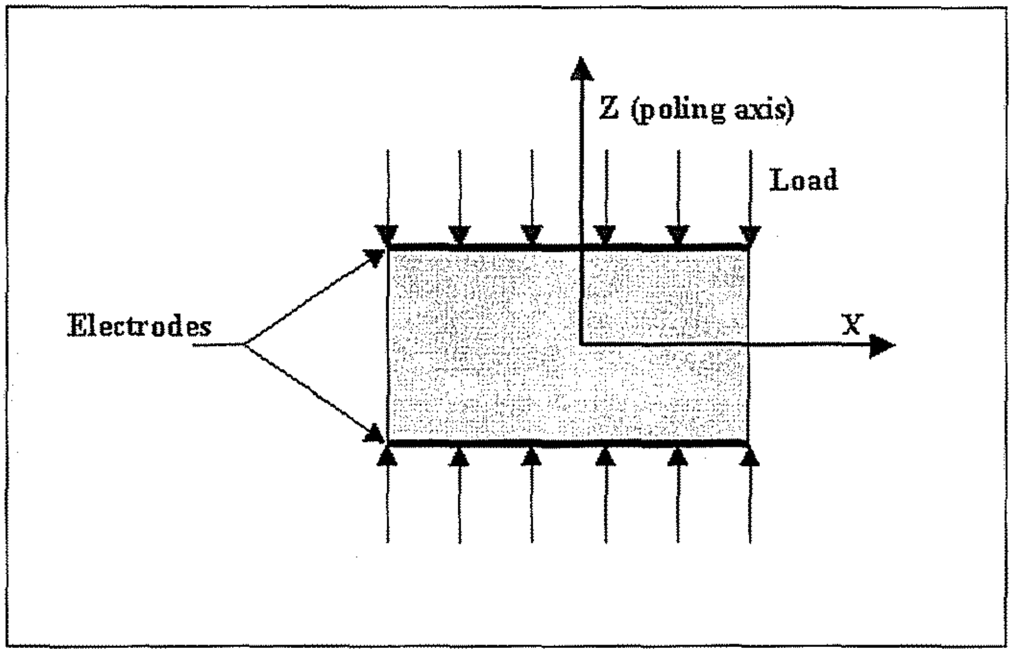

Figure 1: Load Diagram of a piezoelectric element.

THE APPLICATION

The transducer that is the subject of this study is embedded into a bar of a plate segment from a mechanical wood-pulp refiner, as shown in Figure 2. In a refiner, these plate segments are assembled onto opposed discs, one of which is rotating at high speed. A slurry of water and wood chips is introduced at the axis between the discs and, as it flows radially outward, the chips or pulp are refined by the repeated mechanical loads resulting from the passage of the bars on one disc over the bars on the opposed disc.

The structure of the transducer is shown in cross-section in Figure 3 and in and exploded isomtric view in Figure 4. The T-shaped probe is supported within a split housing by the four piezoelectric elements and the entire assembly is secured to the back of a refiner plate segment. The tip of the T-shaped probe projects through a custom manufactured hole, replacing a short (5 mm) segment of a bar. Tangential and normal forces applied to the tip of the probe are transferred to the piezoelectric elements. The resulting signals from the piezoelectric elements are used to uniquely determine the magnitudes of the normal and shear forces.

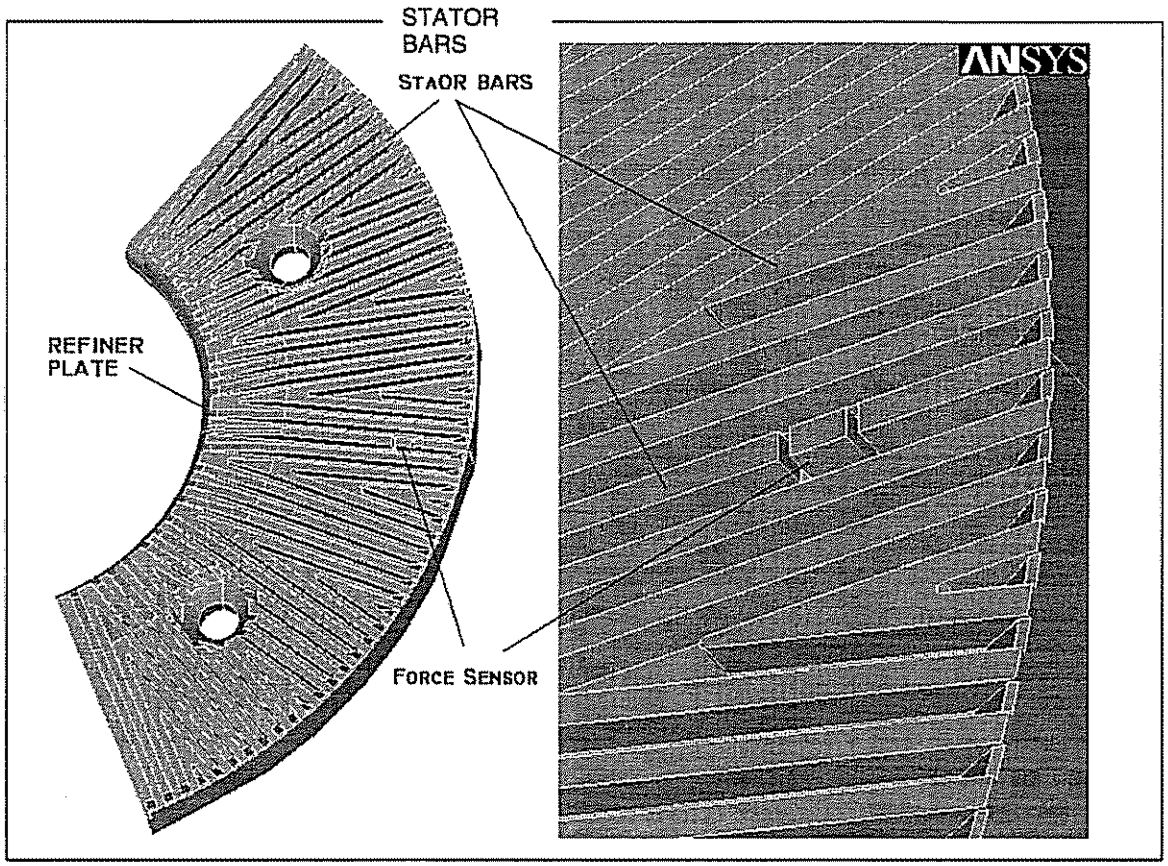

Figure 2: Schematic of force transducer installed in refiner plate segment.

THE FINITE ELEMENT MODEL

A parametric finite element model of transducer was developed in ANSYS, as shown in Figure 5. This parametric model can be automatically modified, rebuilt and then solved repeatedly until an "optimized" design is achieved. This model includes: piezo-ceramics, insulating layers, upper and lower housings, and the screws that hold the sensor together. The model also includes pre-compression of the piezoelectric elements as occurs during assembly.

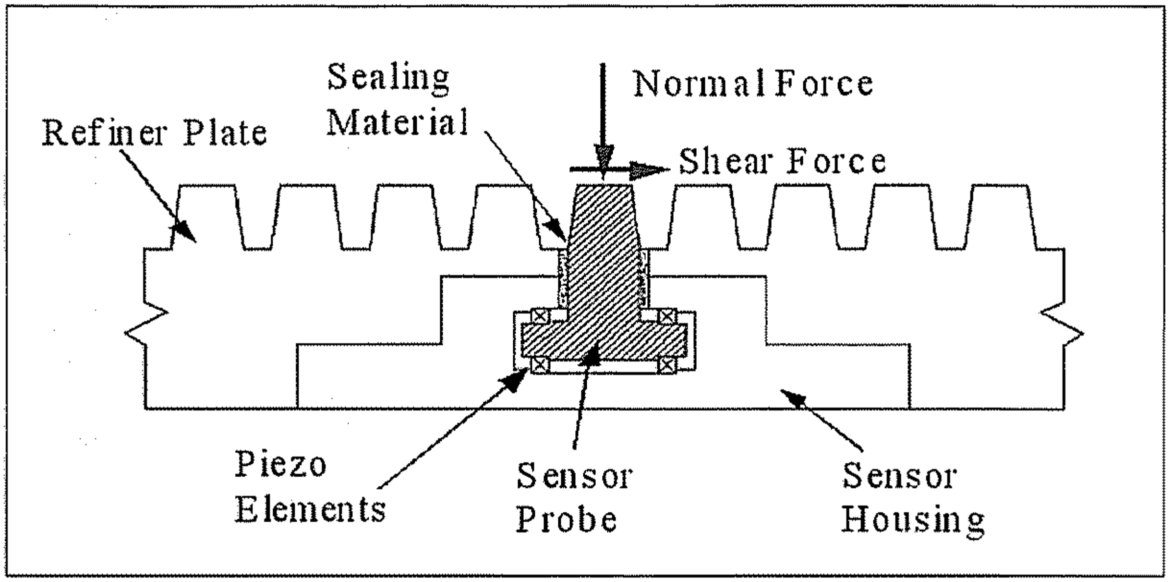

Figure 3: Schematic cross-section of force transducer.

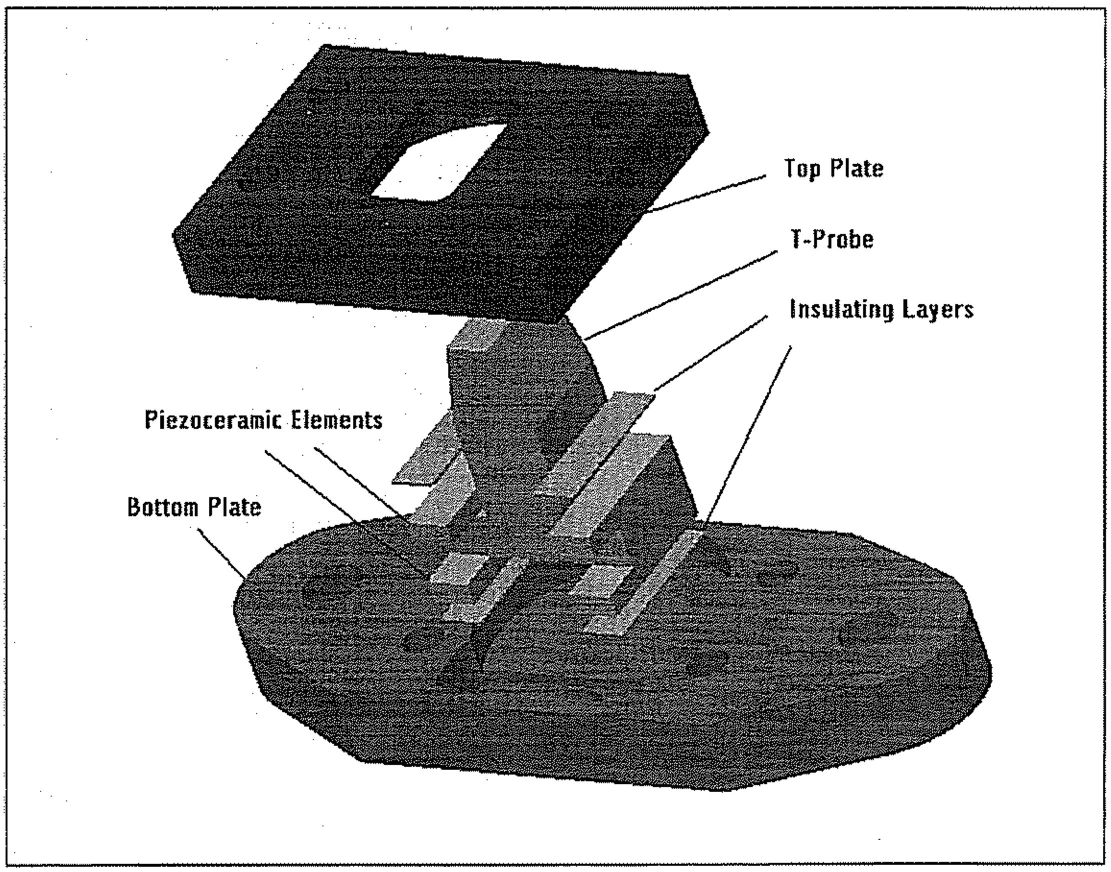

Figure 4: Exploded isometric view of the transducer.

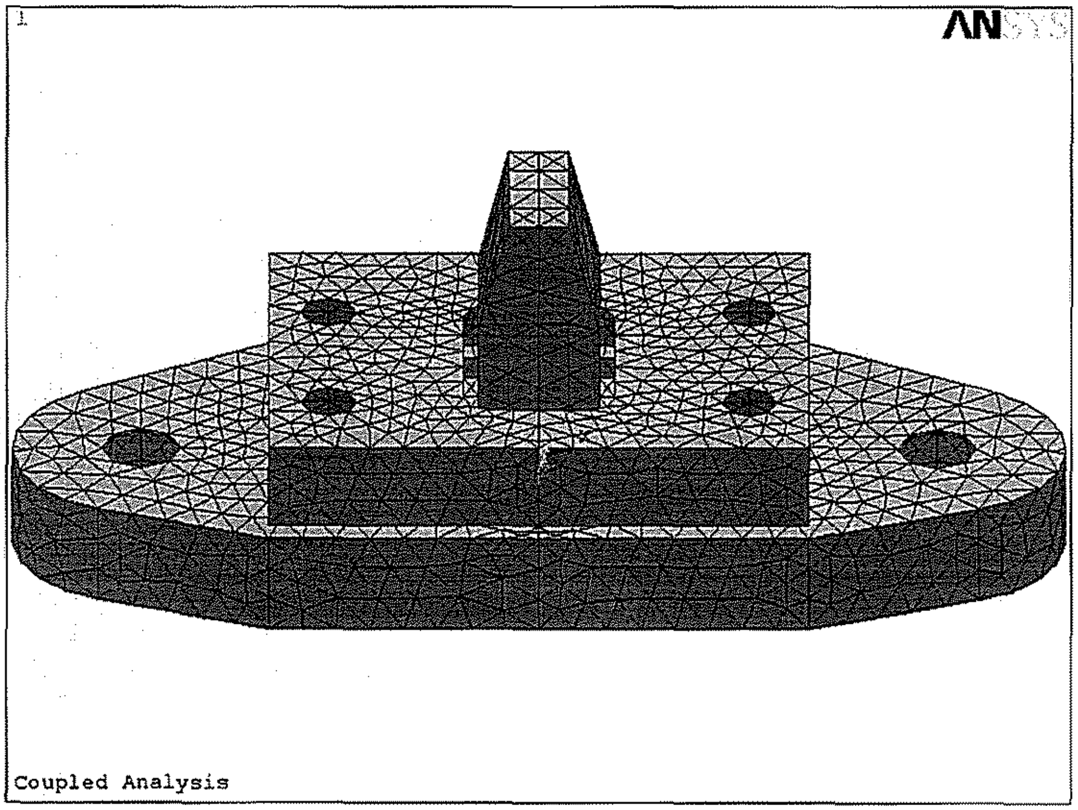

Figure 5: Finite element model of force transducer.

Boundary Conditions

Analyses were conducted for the transducer in a free state and with boundary conditions intended to represent the fixation of the sensor to the refiner plate. This boundary condition was modelled by constraining to zero, all displacements in an annular region on the upper surface of the bottom plate, surrounding the bolt hole.

Analyses were conducted for two cases of electrical boundary conditions. In the first case, commonly called the "resonance condition", a constant voltage of zero is applied at the electrodes (interfaces) of each piezoelectric element. This is a "short-circuit" condition. The second case, called "anti-resonance", a common zero voltage is applied only at the electrode that is closest to the probe. The other electrode has no voltage specification. This condition represents an "open-circuit".

Preload

This preload is modeled by assigning a coefficient of thermal expansion to the screws and then decreasing the temperature of the screws only. This causes the screws to contract and thus compress the piezoelectric elements. Where the screws pass through the top plate of the housing, the elements of the plate are "glued" to the elements of the screw whereas there is a small annular gap between the screw and the mating holes in the bottom plate. To avoid introducing unwanted loads to the top plate, orthotropic coefficients of thermal expansion are assigned to the screws. The coefficient is set to 1.76×10^{-5} °C^{-1} in the z-direction, which is the correct value for stainless steel. In the x- and y-directions, the coefficients are set to zero. Thus, as is the case in the actual transducer, only preload in the z-direction is developed. It was determined that, to achieve a preload of 25 N on each piezoelectric element, a temperature change of -1.47 °C was required. The deformed shape of the transducer, much exaggerated for the purposes of visibility, is shown in Figure 6.

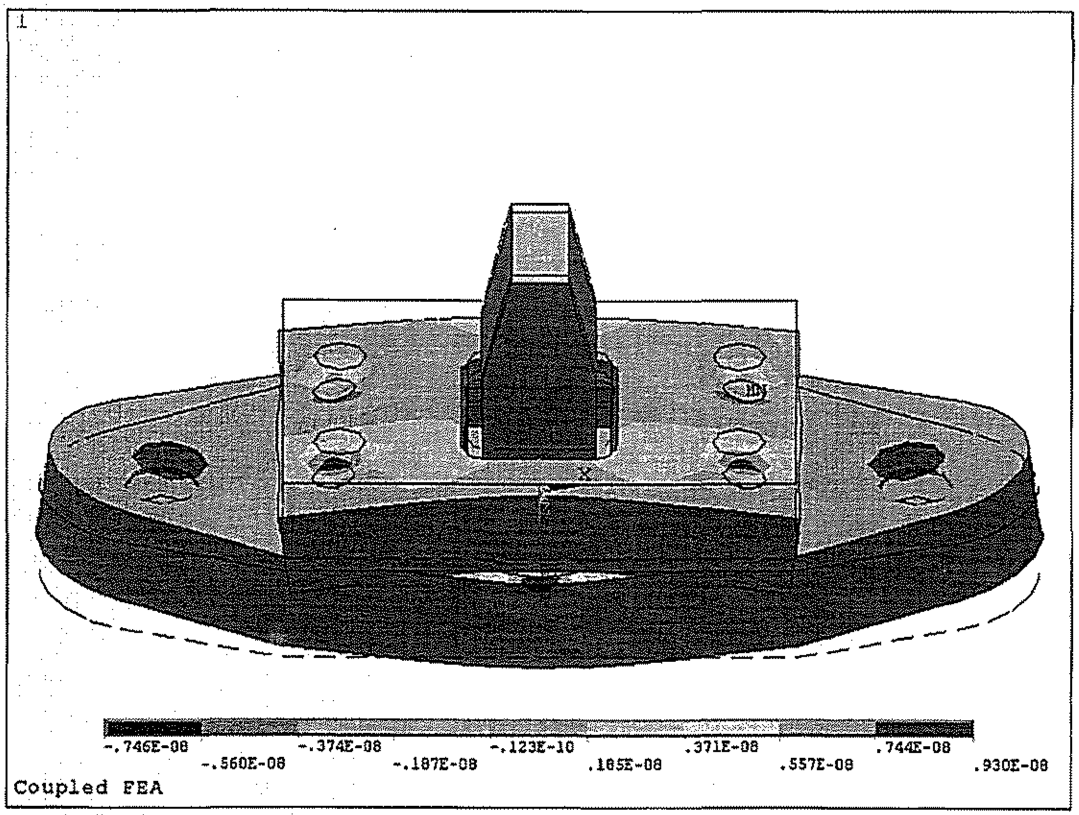

Figure 6: Transducer deformation due to preload ($\Delta T = -1.47\,^{\circ}C$)

Element Types and Mesh Development

Two different types of elements were used to model the transducer. The T-shaped probe and housing were modeled with SOLID92 structural solid elements. SOLID98 coupled solid elements were used to model the piezoelectric elements.

When using finite element software, it is critical to ensure that the desired results have converged. This is accomplished by first running test cases with a relatively coarse mesh to ensure the model is properly constructed. Then the mesh is gradually refined so that more elements are used in the solution. With each solution, the new results are compared to the previous case. As the number of elements is increased, the accuracy of the results will increase, as will the time required for the computer to perform the calculations. Eventually, increasing the number of elements will no longer change the results significantly, at which time the results are said to have converged. The procedure described in the previous section was followed several times for models with increasing number of elements. The total number of elements was 15,474 and the total number of nodes was 27,886 after mesh convergence studies.

ANALYSES

A static analysis was first performed to adjust the original stiffness matrix to include the effect of preload. Following this, a modal analysis was performed using this preloaded stiffness matrix. The Block Lanczos method used for the modal analyses as this method is both acurate and computationally efficient in comparison with other techniques such as the Subspace, Power Dynamics, Reduced (Householder) and Unsymmetric methods [**8**]. The frequencies and mode shapes of the first three modes were extracted.

In addition, a harmonic analysis of the sensors response to a forced excitation was performed. This analysis was based on the Jacobi Conjugate Gradient (JCG) method [**13**] and was performed over a frequency range of 1-30 kHz in equal steps of 350 Hz with shear and normal force magnitudes of 10 N and 30 N, respectively. At each frequency, computes the steady-state response of the system to a sinusoidally varying input is determined. Damping was assumed negligible in this analysis.

EXPERIMENTAL DETERMINATION OF FREQUENCY RESPONSE

Frequency response experiments were conducted on the assembled force transducer using and impact hammer (PCB model 086D80) and a dynamic signal analyser. Transfer functions were calculated based on an average of 20 vertical (or horizontal in shear force case) strikes with the force hammer. These averages where calculated for the transducer in both the free condition, and attached to the refiner plate (mock-up) in the operational configuration.

RESULTS

The effects of constrained and unconstrained boundary conditions on the modelled first natural frequencies are shown in Table 1 for both the open-circuit and the closed circuit cases. Also shown are the experimentally determined first natural frequencies for the constrained and unconstrained cases. These natural frequencies are estimated from the frequency response curves shown in Figures 7 and 8. There is relatively good agreement between the experimental results and the modelled results for both the constrained and the unconstrained cases. In addition, the effects of the open versus closed circuit boundary conditions are minor relative to the resolution with which these natural frequencies can be experimentally determined. All subsequent results are for the closed circuit case.

This model has been used to investigate the effects on the first natural frequency of a variety of proposed design modifications. For example, the effect of the thickness of the piezoelectric elements The first natural frequency of the transducer increases by decreasing the piezoelectric elements thickness as shown in Table 2, but in other hand the net signal of piezoelectric elements on vertically opposite sides of the transducer probe do not increase. The resolution of output signals decreases by using thinner piezoelectric elements and also the risk of cracking of these element increases under dynamic cyclic loading in refiners. By consideration all geometric constraints in instillation of the transducer inside the refiner plate, 1mm thickness is considered for piezoelectric elements.

Table 1: Summary of experimental and modelled first natural frequencies

Boundary Conditions	FEA (kHz)	xperimental (kHz
Unconstrained & Short circuit	10.42	10.4
Unconstrained & Open circuit	10.82	
Constrained & Short circuit	6.24	6.5
Constrained & Open circuit	6.65	

A cyclic load will produce a cyclic response, a harmonic response, in a structural system. Harmonic response analysis can predict the dynamic behaviour of the transducers that will allow one to verify whether or not the design will successfully overcome resonance, and other harmful effects of forced vibrations. Transient effects are assumed negligible in the harmonic analyses.

These outputs will be significant to the calibration of transducer and were also the quantity measured in physical testing. Electrical output results from the ANSYS time history postprocessor are shown in Figure 9, for two of the piezoelectric elements. The outputs of piezoelectric elements show a flat frequency response at least up to 6 KHz. This means that the force transducer can be used to measure the dynamic forces inside the pulp refiner up to 6KHz, successfully. Figure 10 shows the piezoelectric elements outputs under the combined shear and normal forces.

Table 2: Modelled first natural frequencies for three thicknesses of piezoceramic elements (unconstrained and short circuit boundary condition)

	Thickness of Piezoceramic Elements		
	0.5 (mm)	1 (mm)	1.5 (mm)
First Natural Frequency (kHz)	11.27	10.42	9.85

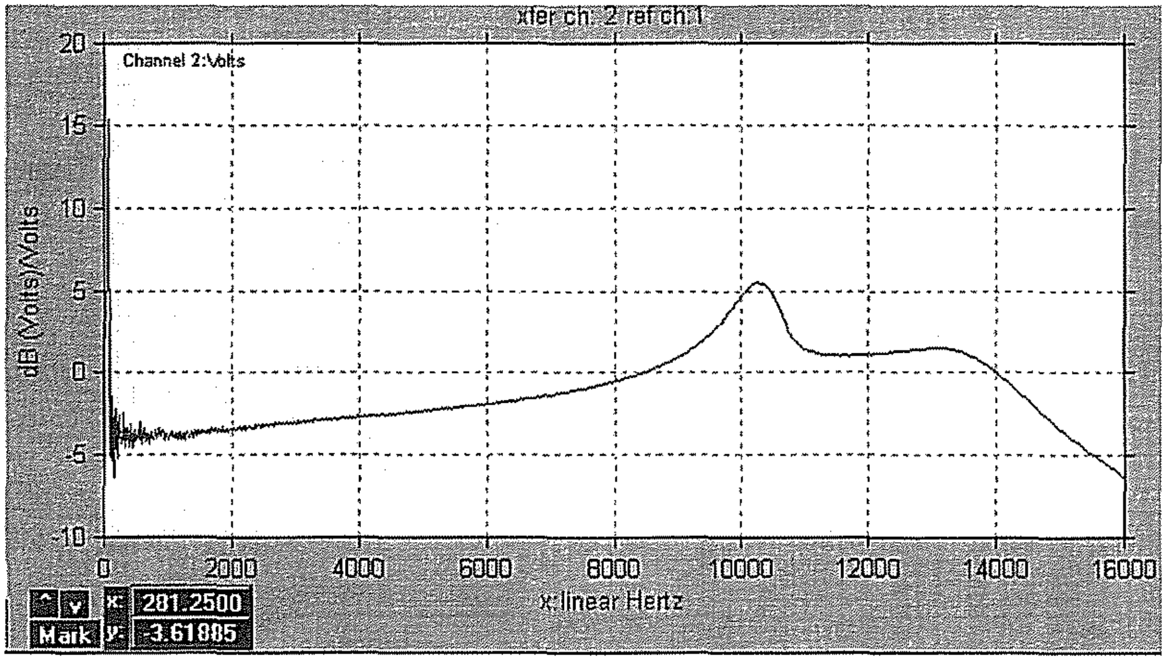

Figure 7: Frequency response of force transducer in the free condition (vertical impact).

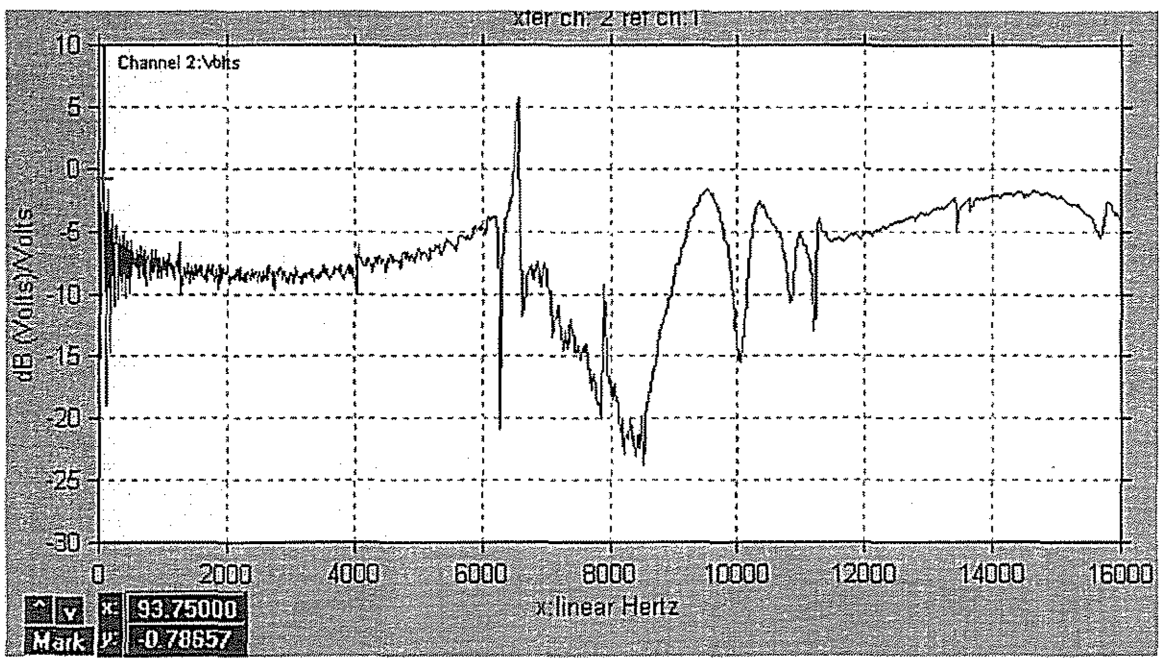

Figure 8: Frequency response of force transducer in the fixed condition (vertical impact).

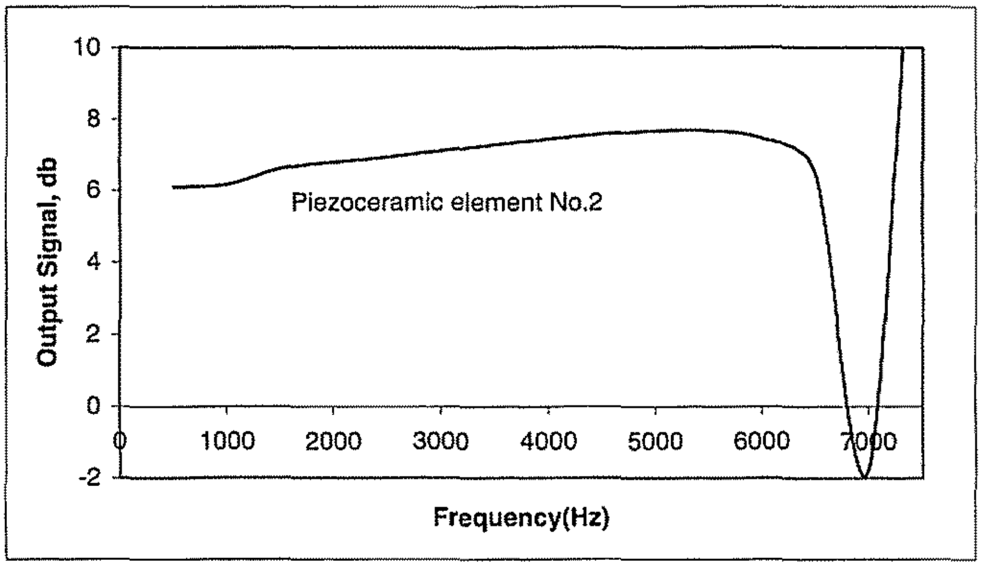

Figure 9: Piezoelectric elements outputs based on Finite Element harmonic analysis.

CONCLUSION

The analysis of a piezoelectric transducer can be complicated due to coupling of electrical and mechanical effects. To capture both the structural and electrical effects together, a coupled 3D FE calculation of a novel pulp refiner transducer was presented by using ANSYS. With respect to piezoelectric material properties, careful attention was paid to units and meaning, specially since ANSYS admittedly doe not follow the convections followed in the typical manufacturer's piezoelectric data sheets. Modal analysis was performed for both unconstrained and constrained boundary conditions. Our study showed the type of the electrical boundary condition played an important role in the dynamic response of the designed transducer and its first calculated natural frequency. The compression of modal analysis results showed an excellent agreement with physical test measurements on the actual device. Harmonic analysis of transducer was also performed. The harmonic analysis showed that the transducer could be used to measure the dynamic forces up to 6 kHz inside the pulp refiner, successfully. The resolution of output signals of piezoelectric elements of the transducer was also investigated under dynamic cyclic loading. The simulation of the lab-scale transducer electromechanical behaviour provides enough accurate results which are sufficient to parameterize the geometry of transducer elements without manufacturing and testing new prototypes.

REFERENCES

1. Hwang, W.S and Park, H.C. "Finite Element Modeling of Piezoelectric Sensors and Actuators," AIAA Journal, Vol. 31, pp. 930-937, 1993.

2. Yen, Y. T., Yang, C. J., Chien , W. T., and Wu, D. H. "Coupled-field Analysis of Piezoelectric Beam Actuator Using FEM," Sensors and Actuators A, Vol. 118, pp. 171-176, 2005.

3. Elka, E., Elata, D., and Abramovich, H., "The Electromechanical Response of Multi-Layered Piezoelectric Structures," Technical Report ETR-2003-2, Israel Institute of Technology, 2003.

4. Lowrie, F., Cain, M., and Stewart, M. "Finite Element Modelling of Electroceramics", NPL Report CMMT (A) 150, 1999.

5. Benjeddou, A., "Advances in Piezoelectric Finite Element Modeling of Adaptive Structural Elements: A Survey," Computers and Structures, Vol. 6, pp. 347-363, 2000.

6. Barnett, A.R., Peelamedu, S.M., Dukkipati, R.V. and Nagathan, N.G., "Finite Element Approach to Model and Analyze Piezoelectric Actuators," JSME International Journal, Vol. 44, pp. 476-486, 2001.

7. Allik, H., and Hughes TJR.," Finite Element Method for Piezoelectric Vibration," International Journal of Numerical Methods in Engineering, Vol. 2, pp. 151-157, 1974.

8. Poole, G., Liu, Y., and Mandel, J., "Advancing Analysis Capabilities in ANSYS through Solver Technology," Electronic Transactions in Numerical Analysis , Vol.15, pp. 106-121, 2003.

9. Tzou, H.S. and Ye, R., "Piezothermoelasticity and Precision Control of piezoelectric Systems: Theory and Finite Element Analysis," Journal of Vibration and Acoustics, Vol. 116, pp. 489-495, 1994.

10. ANSYS Coupled-Field Analysis Guide, Release 5.5, ANSYS Inc., Sep. 1998, pp.3-6.

11. ANSYS Tip of the Week: Conversion of Piezoelectric Material Data, Collaborative Solutions Inc., An ANSYS Support Distributor, November 1999.

12. Piezo Kinetic Inc., www.piezo-kinetics.com/materials.htm, Electro Ceramics Division.

13. Hestenes, M. R., and Stiefel, E., "Methods of Conjugate Gradients for Solving Linear System," Journal of Research of the National Bureau of Standards, Vol. 49, pp. 409-436, 1952.

APPENDIX: Properties of PKI -502:

Dielectric Permittivity:

$$[\varepsilon] = 8.85 \times 10^{-12} \begin{bmatrix} 1730 & 0 & 0 \\ 0 & 1730 & 0 \\ 0 & 0 & 1700 \end{bmatrix} \text{Farad/meter}$$

Elastic Compliance Constants:

$$[s]=\begin{bmatrix}16.4 & -5.74 & -7.22 & 0 & 0 & 0\\ -5.74 & 16.4 & -7.22 & 0 & 0 & 0\\ -7.22 & -7.22 & 18.8 & 0 & 0 & 0\\ 0 & 0 & 0 & 47.5 & 0 & 0\\ 0 & 0 & 0 & 0 & 47.5 & 0\\ 0 & 0 & 0 & 0 & 0 & 11.7\end{bmatrix}\times 10^{-12} \text{ meter}^2\text{/Newton}$$

Piezoelectric Strain Constants:

$$[d]=\begin{bmatrix}0 & 0 & 0 & 0 & 580 & 0\\ 0 & 0 & 0 & 580 & 0 & 0\\ -171 & -171 & 400 & 0 & 0 & 0\end{bmatrix}\times 10^{-12} \text{ meter/Volt OR Coulomb/Newton}$$