Modeling and Simulation of THUNDER Actuators using ANSYS Finite Element Analysis

C. Kennedy*, T. Usher**, J. Mulling*** and A. Kingon****

*Think Peak, Inc., Apex, NC, USA, ckennedy@thinkpeak.com
**California State University, San Bernardino, CA, USA, tusher@wiley.csusb.edu
***North Carolina State University, Raleigh, NC, USA, jfmullin@unity.ncsu.edu
****North Carolina State University, Raleigh, NC, USA, angus_kingon@ncsu.edu

ABSTRACT

The use of a quarter-symmetry three-dimensional finite element analysis model in predicting the behavior of the laminated piezoceramic material device known as THUNDER actuators is investigated. The analyses simulating the fabrication process and subsequent applied voltage loading were investigated. The finite element model includes the metal and piezoceramic materials and also the adhesive layers. The elements within the model have very high aspect ratios and also large changes in element height between the various THUNDER layers of material. The simulation begins as stress-free at 270 C and thermal contraction affects cause the deformation of the THUNDER device into the deformed dome shape. The model geometry is then altered so it assumes the deformed shape in a stress-free state. The domed shape of the THUNDER is then subjected to an applied voltage and displacements are obtained. The results of the simulation are compared to laboratory measurements.

Keywords: Finite Element Methods, rectangular THUNDER, piezoceramic.

1 BACKGROUND

The THUNDER actuators have been modeled using analytical and numerical methods as presented in papers [1], [2], and [3]. For our investigation we required a predictable finite element model that could be used to explore THUNDER actuator design concepts without requiring a physical model. Thus we endured to obtain an ANSYS finite element model that would serve as a basis for the different THUNDER geometries that were being investigated. The ANSYS model simulates the manufacturing process and then the subsequent application of voltage.

2 FINITE ELEMENT MODEL

The ANSYS model is defined using the ANSYS Parametric Design Language (APDL)[4]. The dimensions of the actuator are defined using parameters and also the piezoceramic material description. From reference [3] it is known that the THUNDER actuator has regions of instability as it is deforming into its domed shape during the cooldown process. Therefore the options that are utilized by the solver are also parametrically defined, since it was observed that different geometries would require different solution times and possibly different solution strategies to obtain a solution. This is due to some of the investigated geometries having instabilities that are more prominent then the other THUNDER sizes. The finite element model includes a layer of elements for each layer in the THUNDER actuator. Each layer has multiple elements through the thickness. Mesh sensitivity studies were conducted to understand the relation between the accuracy of solution, computation time, and element mesh size. We use two elements through the thickness of the adhesive layers and three elements in the remaining layers. The aspect ratio of the elements varies from 212:1 over the length and 263:1 over the width of the actuator. The dimensions for the THUNDER actuator were obtained from Face International [5] and taken as room temperature values and shown in Table 1. The dimensions used at the beginning of the cooldown process were increased by the average coefficient of thermal expansion of each material times the temperature differential. The width and length of the adhesive layers were kept the same as the layer of aluminum on the top surface of the piezoceramic and the same width and length as the piezoceramic on its bottom surface. Thus the finite element model had a stepped cross-section (Figure 1).

Table 1 THUNDER Actuator Dimensions

Model	6R	7R	8R	9R	10R
Thickness (mm)	6.096	5.334	4.318	4.318	4.318
Stainless Steel Length (mm)	76.2	97.66	63.5	22.22	25.4
Stainless Steel Width (mm)	51.82	73.28	13.72	10.54	13.72

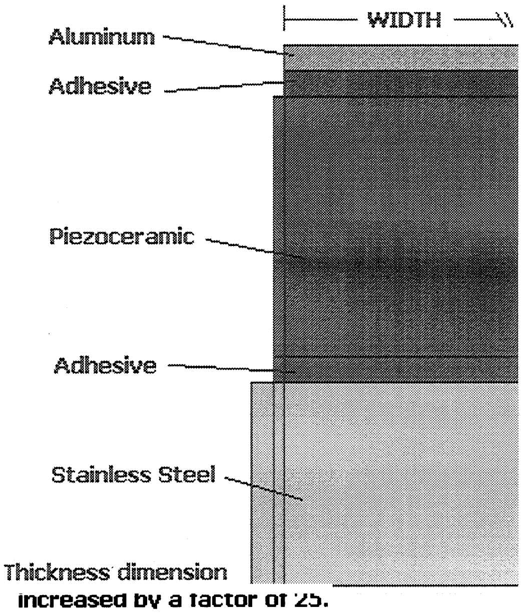

Figure 1 End View of solid geometry for finite element model showing stepped cross-section widths. Thickness increased for figure, partial width shown.

The finite element model is a quarter-symmetry idealization with the Y direction being the thickness direction and symmetry planes along the XY and YZ planes. Displacement constraints are applied in each symmetry plane's normal direction. A single node is constrained in the Y direction. This allows the actuator to assume the two primary bending modes and thus the domed shape (Figure 2).

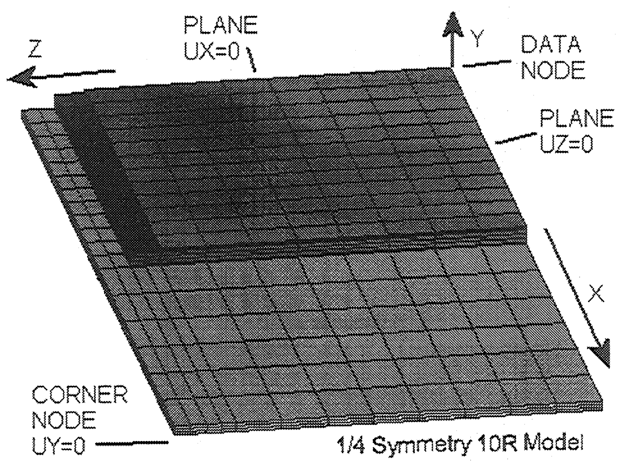

Figure 2 ¼ Symmetry Actuator Model

2.1 Material Models

The THUNDER actuator consists of metallic layers, the piezoceramic layer and two adhesive layers. For the ANSYS model the metallic layers were modeled as linear isotropic materials. For this study the variation of modulus of elasticity due to the manufacturing of the thin sheet was not represented (i.e., typically the elasticity varies in the longitudinal and transverse directions). The coefficient of thermal expansion was the only material property that included temperature dependency. The piezoceramic material was represented with the ANSYS anisotropic elastic matrix and the piezoelectric material model. The adhesive material LaRC-SI has a step-change in its coefficient of thermal expansion (CTE) at 150 C.

3 CURE SIMULATION DESCRIPTION

The simulation includes the Cure process, where the rectangular planar actuator is heated to a uniform temperature and then cooled inside a vacuum chamber. During cooling the difference in the materials' coefficient of the thermal expansion causes internal stresses, and the THUNDER actuator distorts into a domed configuration. The actuator has several different potential equilibrium states and some of the THUNDER geometries can be easily altered using a snap-through action [3]. To influence the shape that would be arrived at by the finite element solution a small force was also applied to bias the shape of the actuator into the desired equilibrium configuration. Forces were applied along the YZ symmetry plane in the positive Y direction and compressive forces applied along the width of the actuator in the negative X direction. The force loading induces the desired shape and keeps the nonlinear solution in a stable region as the effects of the temperature differential are considered. The forces are deleted and the thermal contraction forces remain to load the system and the shape of the actuator stays in the desired equilibrium state. At the end of the cooldown process the actuator is in a stressed state.

The cooldown was conducted as a nonlinear, large deformation solution using the ANSYS frontal solver. It was processed using several load steps in order to efficiently traverse the step-change in the adhesive's coefficient of thermal expansion. Since the SOLID5 [4] piezoceramic element within ANSYS does not support large deflection, the SOLID64 [4] element was utilized. This allowed the same structural anisotropic material definition but it did not provide a voltage degree of freedom. For the cooldown simulation the voltage response of the piezoceramic was not required. The ANSYS parametric model description also includes parameters that influence the solver process. We initially used the iterative solvers (PCG and Sparse) within ANSYS [4] and these required a substantial amount of computer time. Changing to the frontal solver resulted in a much reduced solution time. This was attributed to the large internal force discontinuities that occur at the layer interfaces which required a substantial number of iterations for the iterative solvers to converge. The frontal solver uses direct Gaussian methods whereas the iterative solvers utilize non-direct techniques [6].

4 APPLIED VOLTAGE SIMULATION DESCRIPTION

The deformation of the model after the cooldown simulation was then used to update the geometry of the THUNDER actuator using the ANSYS UPGEOM function [4]. The UPGEOM function takes the displacements from the bonding process and updates the node coordinates to include the displacement. This in effect relieved all stresses from the simulation model. Thus the pre-stressed condition of the THUNDER actuator due to the thermal contraction during cooldown was removed. The materials in the layers are represented using a linear representation and the pre-stress or lack thereof does not change the amount of deflection caused by the voltage application. The second portion of the simulation calculated the deformation of the domed shape actuator under an applied voltage from an initial stress-free and voltage free state. The piezoceramic layer elements were changed to ANSYS element SOLID5, the piezoceramic materials defined and the voltage applied. This portion used a linear solution with the frontal solver. The metallic layers were modeled using a linear material model; we did not consider plasticity occurring in the metallic layers. No thermal effects were considered as the reference and uniform temperatures were set to 25 C. A voltage differential of 480 volts was applied across the piezoceramic layer.

5 SOLUTION PROCEDURE

The cooldown process was simulated using nonlinear geometry options. The piezoceramic was represented with the SOLID64 element, and the other layers used the SOLID45 element. The temperature differential was 245 C and was applied in a single load step subdivided into 50 substeps and used line search, the predictor, stress stiffening, and multiple equilibrium iterations [4]. The forces were then deleted and the solution allowed to stabilize into the deformed shape caused by the thermal contraction over one additional load step.

The solution module was exited and the preprocessor entered to revise the finite element model using the UPGEOM command so it would start from the deformed shape for the voltage application. The element type and material for the piezoceramic were also revised.

The voltage application simulation was done using linear geometry options since the SOLID5 multi-field element does not support large deformation. A voltage load of 480 volts was applied across the piezoceramic layer and was applied over one load step [6].

6 RESULTS PROCESSING

A unique ANSYS database and result file were created for the Cure and applied voltage simulation. The post-processing was automated to generate a report for each analyzed Thunder model.

The center node from the top surface of the actuator is used to obtain the displacement information from the Cure process and voltage application. This is the "DATA NODE" shown in Figure 2.

7 COMPARISONS

The THUNDER models are listed in Table 2 with their published and calculated Cure deformations. The published data was obtained from Face International [5]. The accuracy of the finite element model varied from less then 1 percent to about 40 percent.

The voltage application deflection was obtained experimentally by taking numerous measurements and using the average. The applied voltage deformation accuracy varied between 10 and 55 percent accuracy.

Table 2 Cure Bonding Process 245 C

Model	Published Bonding Process Deflection (mm)	FEA Bonding Process Deflection (mm)	% Accuracy
6R	3.45	4.72	36.8
7R	8.99	9.03	0.5
8R	3.35	3.12	6.9
9R	0.61	0.36	40.8
10R	0.64	0.78	21.2

Table 3 Voltage Application 480 V

Model	Experimental Voltage Deflection (um)	FEA Applied Voltage Deflection (um)	% Accuracy
6R	-620	-282	54.6
7R	-1370	-942	31.2
8R	-400	-357	10.6
9R	-58.5	-43.1	26.3
10R	-84.7	-59.0	30.4

8 SUMMARY

The analysis model has shown that it can accurately predict the deformation due to the Cure bonding process and also due to the voltage application for some of the THUNDER actuator sizes. The variation in accuracy can be attributed to several potential influencers. However additional investigation and studies should be conducted to fully understand the actuator behavior. Preliminary sensitivity studies were conducted on the thickness variation of the actuator layers versus the bonding deformation and voltage deformation. The bonding deformation is most dependent on the stainless steel substrate thickness; the piezoceramic and aluminum thickness are approximately equivalent on their effect on the bonding deformation but about half as strong as the stainless steel layer.

For the voltage application the piezoceramic thickness has the greatest effect on the resulting deformation. The voltage deformation has a very weak dependency on the aluminum and stainless steel thickness.

Bonding process simulation accuracy is very dependent on metal thickness. Doing a study of the minimum and maximum thicknesses versus the accuracy would be appropriate. Using the Probabilistic Design System available in ANSYS 5.7 [7] would be useful. The PDS system should have all the parameters of the model as inputs including geometry, material properties, etc. It would point out dependencies and relationships that would be difficult to determine by manually making changes and processing the data.

REFERENCES

[1] Soderkvist, "Piezoelectric Simulations Validated on Beams", ANSYS Conference Proceedings, pg. 2-255,1996.

[2] Capozzoli, Gopalakrishman, Hogan, Massad, Tokarchik, Wilmarth, Banks, Mossi, Smith, "Modeling Aspects Concerning THUNDER Actuators", Industrial Mathematics Modeling Workshop, North Carolina State University, 1998.

[3] Hyer and Jilani, "Predicting the deformation characteristics of rectangular unsymmetrically laminated piezeoelectric materials", Smart Material Structures, 7 (1998) 784-791.

[4] ANSYS, Inc., "ANSYS 5.6 User's Manual on-line", 2000.

[5] Face International Corporation, 427 W. 35th St. Norfolk, VA 23508 757.624.2121.

[6] ANSYS, Inc., "ANSYS 5.6 Theoretical Manual on-line", 2000.

[7] ANSYS, Inc. "ANSYS 5.7 Product Release Webcast", http://www.ansys.com/57/index.htm, 2000.