Excerpt from the Proceedings of the 2011 COMSOL Conference in Bangalore

Heat Generation from Dielectric Loss and Vibration using COMSOL Multiphysics

Rajesh Jeba.T, Benjamin V. Vins, Ramamoorthy V
CAE&ID, Engineering Service Group, HCL Technologies LTD
ELCOT – Special Economic Zone, SDB1- Right Wing, 602/3, No. 138, Shollinganallur Village, Tambaram Taluk, Kancheepuram District, Chennai-600119 Tamilnadu, India.
rajeshjeba.t@hcl.com, benjamin.vins@hcl.com, ramamoorthy-v@hcl.com

Abstract: This paper presents a FEA approach to estimate temperature rise experienced in a typical ultrasonic transducer due to dielectric loss and vibration. The voltage is varied to study the heat generated for the various loading conditions. The heat generation across the PZT varies from disc to disc with respect to location from the node (zero displacement location) of the structure. Similarly the variation of strain along the transducer is also based on location of the nodes and acoustic gain.

A 3-dimensional finite element model was developed in COMSOL to predict the temperature distribution of the transducer. A typical transducer is modeled to operate at 24 kHz frequency. Eigen value analysis is carried out to calculate the exact operating frequency of the transducer. A frequency response analysis is carried out at the operating frequency by giving the designed voltage as the input to the PZT's. The strain from this analysis is extracted, which is used for thermal analysis. The heat generation due to dielectric loss and strain are calculated from Equ (1) and (2). Most of heat is generated due to dielectric loss. Transient analysis is performed using the above inputs for various loading conditions.

Keywords: Dielectric heating, strain rate heating, Heat in Ultrasonic Transducer.

1. Introduction

Heat generation due to high electric potential and vibration is a concern in PZT/Solid structure as it increases the temperature significantly. This paper will educate heat generation in an ultrasonic transducer due to electric potential and vibration in the model. While converting electrical energy into mechanical energy over a PZT/solid structure, a part of energy lost into heat. When a structure vibrates, a loss of energy happens due to its internal damping characteristic of the material.

A 3-Dimensional ultrasonic transducer was developed for this study purpose. The model is designed to operate at longitudinal frequency.

2. Background

An ultrasonic transducer operated at higher than 20 kHz. This ultrasonic transducer is coupled with various devices like precision cutting tools, ultrasonic drillers, which are designed to operate on the same frequency. These devises are used for various applications like diamond cutting, drilling, space applications for its lightweight and in medical devices. It works on the basic principle of exciting the longitudinal mode of the device at the same frequency, where the excitation is induced by converting the electrical signal into mechanical movements of the PZT.

3. Problem Definition

The common problem of the ultrasonic transducer are upon the continuous usage of the device, the device gets heated up more than the comfortable operating temperature range of the user. The performance of PZT deteriorates with the rise in temperature; also it causes failures of the components. Some of the heat in a typical transducer mainly because of the losses due to supplied power. The various losses are dielectric loss and mechanical loss.

3.1 Dielectric Loss

To determine the internal heat dissipation in PZT, an electromechanical model was developed. Part of the supplied electric energy is converted into mechanical energy and the remaining portion lost as heat. A loss of energy which eventually produces a rise in temperature of a dielectric placed in an alternating electrical field.

A dielectric is an electrical insulator that may be polarized by an applied electric field. When a dielectric is placed in an electric field, electric charges do not flow through the material. Polarization is the phenomenon in which electric field is restricted in direction of vibration. As a result slightly shift from their average equilibrium positions causing dielectric loss.

The power density P_1 (W/m^3) by dielectric loss is given by:

$$P_1 = \frac{1}{2}\omega E^2 \varepsilon \eta_e \qquad \text{Equ. (1)}$$

where:

ω = 2πf is the angular excitation frequency

E = Electric field amplitude vector

ε = Relative permittivity vector

η_e = Dielectric loss factor

3.2 Mechanical Loss

The amount of energy lost by vibration is a function of strain energy and material damping, and is expressed in Equ (2).

The axial displacement of the transducer is sinusoidal along the length of the transducer. The locations where zero displacement are seen, called as Node. PZT's are usually placed in this location for maximum displacement. External provision can be attached at this node location without affecting the performance of transducer.

Maximum strain energy observed at the node location and zero strain energy will be observed at antinode location.

The power density P_2 (W/m^3) by strain is given by:

$$P_2 = \frac{1}{2}\omega \eta_m \operatorname{Real}[\varepsilon Conj(D\varepsilon)] \qquad \text{Equ (2)}$$

where:

ω = 2πf is the angular excitation frequency

D = Elasticity matrix

ε = Strain vector

η_m = Mechanical loss factor

4. Geometry

A typical transducer consists of PZT or PZT stacks assembled and preloaded using bolts, it contains PZT stacks, bolts and acoustic horn to amplify the magnitude of vibration. Any cutting tool which is designed to vibrate at the same frequency of the transducer can be attached to the horn. The PZT's are polarized to be actuated in the axial direction of the transducer. A typical transducer wave length (λ) can be represented as below. $\lambda = C/f$, where, C is the velocity of sound in the material and f is the operating frequency. For this study, a transducer is designed to operate at 24 kHz frequency. All solid structure as Titanium material and geometry representation as shown in Figure 1.

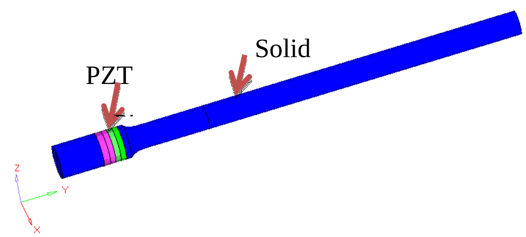

Figure 1. Geometry representation of Ultrasonic Transducer

5. Analysis Steps

An Eigen value analysis for the transducer is performed to predict the exact frequency at which longitudinal mode is occurring. A frequency response analysis is performed in this frequency by exciting the transducer in the axial direction by applying the voltage V to the PZT. This voltage is varied for various loading conditions. From this analysis, the strain energy density is extracted; it is used to calculate the heat generation due to mechanical losses using Equ (2). The dielectric losses in the PZT are calculated using Equ (1). This heat is applied to the transducer and a thermal analysis is performed to find the temperature distribution of the transducer, while finite air domain around the transducer is also considered. Analysis is performed for different electric potential. Temperature at this potential difference is also recorded. A transient dynamics analysis is performed for various loading conditions for 60 seconds.

6. Boundary Conditions

6.1 Modal Analysis

Free-free modal analysis is performed to get the Eigen frequencies and vectors.

6.2 Frequency Response Study

A frequency response analysis is performed by applying the potential difference to the surfaces of the PZT at the fundamental model of the transducer.

6.3 Conjugate Heat Transfer

From the strain energy measured from the response analysis, the power is calculated using Equ (2); this power is applied as a heat source (for mechanical loss in the metals). The heat due to dielectric loss is calculated using the Equ (1) and applied as heat on the PZT's. Fluid domain is shown in Figure 2. '*Heat transfer in Fluids*' is incorporated in the analysis for accounting convective losses to the ambient.

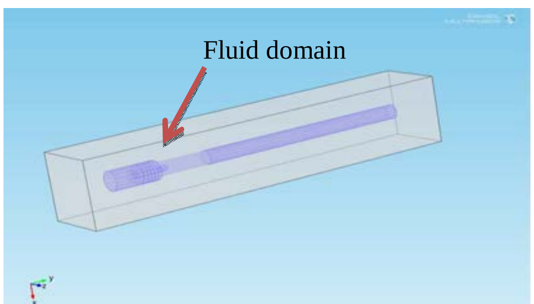

Figure 2. Fluid Domain Representation

7. Results and Discussions

A detailed analysis of the transducer is completed using coupled multiphysics modules in COMSOL. The fundamental frequency obtained through the Eigen value analysis is excited in the frequency response analysis, using '*Piezoelectric*' module. The voltage to the PZT is varied for various loading conditions. The strain energy from the response analysis coupled through Equ (2) is the input to '*Heat Transfer*' module as heat source for mechanical losses. The heat source also for dielectric losses are calculated through Equ (1) is used as an input of '*Heat Transfer*' module. The temperature distribution from the heat transfer analysis are measured and plotted as a line graph and contour and results are compared ate various conditions shown below.

7.1 Natural Frequency Analysis

The transducer was designed to operate at 24 kHz longitudinal frequency. Due to geometric changes and due to presence of PZT material, the transducer operating frequency is offset from the designed value of 24 kHz. To evaluate this, a natural frequency analysis is performed. The location of nodes and anti nodes is measured. These locations can be used for optimization of PZT locations and external mechanism to hold the transducer. The external tools can be attached to the Anti node locations, as stress at this locations are minimal.

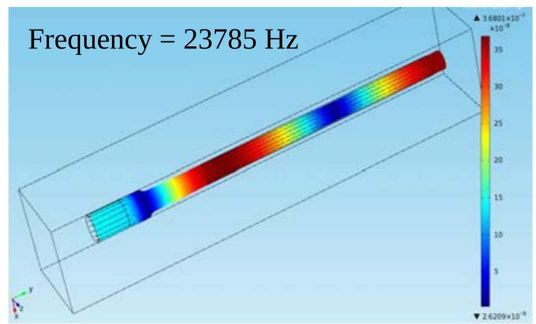

Figure 3. Total displacement plot at 24 kHz longitudinal frequency

7.2 Response Analysis

Displacements are extracted from the response analysis and to achieve a required tip displacement, the voltage is varied. The displacement along the length is plotted below. The strain energy density is extracted and is also plotted.

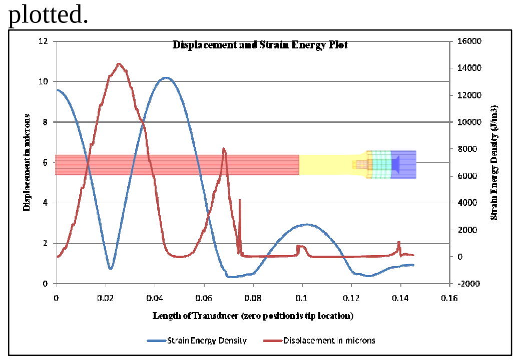

Figure 4. Displacement and Strain Energy plot of transducer

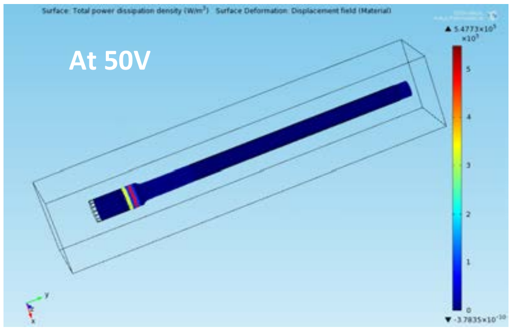

Figure 5. Power density plot at Electric potential 50V

7.3 Thermal Analysis

The temperature distribution for different electric potential are plotted to capture the rise in temperature for a given time frame. These temperatures are plotted to study the variation of temperature with the increase in electric potential. The temperature around PZT disc is measured as it is critical parameter for the performance of the PZT's.

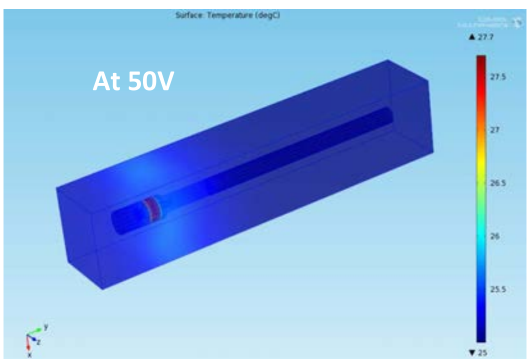

Figure 6. Temperature distribution at Electric Potential 50 V

7.4 Comparison of Result Values

Applied Voltage	Power Density (W/m3)	Maximum Temperature (deg C)
Only vibration (No applied potential)	1.03E+04	25.001
10 V	2.19E+04	25.1
50 V	5.47E+05	27.7
100 V	2.19E+06	35.8

Table 1. Power density and maximum temperature at 10V, 50V and 100V potential

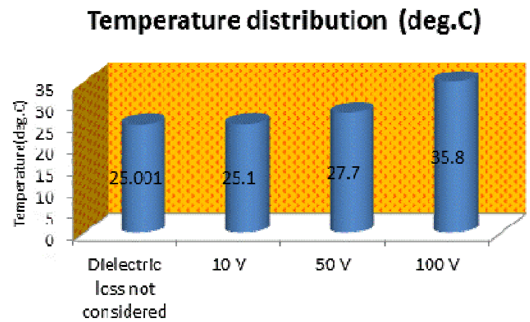

Figure 7. Temperature variation along the Electric potential.

8. Conclusions

a. The methodology to evaluate the temperature distribution of the typical transducer by coupled multiphysics such as Eigen value analysis, frequency response analysis, PZT and thermal analysis, in **COMSOL**.
b. The heat generation in a typical transducer mainly due to dielectric loss in a PZT rather than mechanical losses due to vibration.
c. The variation of heat generation Vs electric potential is non-linear and is significantly low at unloaded and high at loaded conditions.

9. References

a. COMSOL Multiphysics 4.2 user and verification manual
b. Incropera, Dewitt, *Fundamentals of Heat Transfer*, 5th Edition, Wiley Publisher, New York, 1981.

10. Future Scope of Work

a. The transducers are usually housed in shroud that can be included in future analysis to predict the actual temperature rise.

b. The transducer used in this analysis is the simple representation. Optimized transducer can be used for future studies.

c. Heat generation for different loads is analyzed in this study separately. A load curve with respect to time can be used in future studies, for the actual prediction of temperature.