# CONTRIBUTIONS TO THE FINITE ELEMENT MODELING OF LINEAR ULTRASONIC MOTORS

**Oana** CHIVU[2], **Dan** DOBROTA[3], **Claudiu** BABIS [3]

***Abstract****. The present paper is concerned with the main modeling elements as produced by means of the finite element method of linear ultrasonic motors. Hence, first the model is designed and then a modal and harmonic analysis are carried out in view of outlining the main outcomes.*

**Key words:** rotary ultrasonic motor; piezoceramic disk; turret disk

## 1. INTRODUCTION

There are several types of analyses to be performed: static; modal; harmonic and transient.

The active elements of the linear ultrasonic motor are part of the active assembly and they are the piezoceramic sector disk as well as the turret drive actuator.

Geometrical models have been designed to this purpose and modal and harmonic analyses have been performed.

First, in order to launch the modeling process, the **Preprocessor** must be actuated, the geometry designed and the finite elements generated. Second, the properties of each finite element are defined. Each of the elements and their corresponding properties help depict the type of material, the type of finite element and the real parameters.

By actuating the ***Solution*** processor, the model is charged, the type of analysis is selected and the analysis is initialized.

Once the solution obtained, the deformation in the **General Postprocessor** is displayed and the nodal values are listed.

The model directives and its corresponding analysis are as follows:

- *construction of the model consists of:*
    - name of the file (/FILNAME);
    - name of the analysis (/TITLE);
    - unit system (/UNITS);
    - running the preprocessor (/PREP7);
    - type of element and control directives (ET and etc.);
    - parameters to be used for the characteristics of the elements (R and etc.);
    - material properties (MP);
    - the design of the geometric prototype includes:
        - prototype geometry;
        - parameters used to generate the finite elements;
        - finite elements (xMESH).
        - leaving out the preprocessor (FINISH).

- *charging and solution generation will involve:*
  - running the program to obtain the solution (/SOLU);
  - type of analysis and control directives (ANTIPE etc.);
  - the charging process is concerned with:
    - DOF parameters for the finite element nodes (D, DK);
    - force parameters (F, FK);
    - surface parameters (S, SFL);
    - internal parameters (ACEL, OMEGA etc.).
    - solution generation (SOLVE);
    - ending the solution processing.
- *results analysis will entail the following:*
  - running the postprocessor program (/POST1);
  - construction of results database (REZUME);
  - appropriate results generation (PDISP, PLNSOL etc.);
  - list of results (PRNSOL, PRESOL etc.);
  - ending the postprocessor program.
- *exit from **ANSYS** (/EXIT).*

**1.1 Modal Analysis**

The modal analysis addresses the vibration characteristics of the structures (natural frequencies and modal shapes). Likewise, it may act as a springboard for further dynamic analyses such as: transient dynamic analyses, harmonic or spectral analyses. A modal analysis of a pre-grip structure is also possible. With regard to the ANSYS products, the modal analysis represents, in fact, a linear analysis (any discontinuities such as the contact element plasticity are overlooked even if previously defined). Hence, there are several extraction methods: Subspaces (Subspații), Block Lanczos, PowerDinamics, Reduced (Redusă) etc.

The stages of the modal analysis are as follows:

- model elaboration;
- charging and solution generation;
- bloating;
- results generation.

The linear behavior is typical of the modal analysis. The material properties are linear, isotropic or orthotropic, constant or temperature dependent.

For a classic modal analysis, the equation reads:

$$[K]\{\Phi_i\} = \varpi_i^2 [M]\{\Phi_i\} \quad \varpi_i \qquad (1)$$

where: $[K]$ is the stiffness matrix; $\Phi_i$ - modal vector of module I; $\varpi_i$ - modal frequency; $[M]$ - mass matrix.

In stricto sensu, the term “expansion” refers to the bloating of the solutions reduced to the whole DOF set (degrees of freedom). In the case of the modal analysis, the term “expansion” signifies the transferring of the modal shapes into a results file.

The “efforts” of the modal analysis do not stand for the real efforts of the structure, rather they entail a relative distribution for each vibration mode.

The results of the modal analysis are recorded in a structural results file: natural frequencies, shapes of the expanded modes, relative efforts and forces (if need be). Each mode is then stored in the results file separately.

### 1.2. Harmonic Analysis

Within a structural system, a cyclical load will produce a cyclical response (a harmonic response). Harmonic analysis brings about a dynamic behaviour of the structures and it helps check whether the modal reaches resonance and fatigue and other harmonic effects of the vibration forces.

Harmonic analysis is a technique used to determine the linear stationary reaction of the structure when the load varies sinusoidally (harmonically). A key observation is to calculate the structure reaction for several frequencies and represent it graphically according to some dimensions (displacements). By means of this analysis, vibration stationary forces can be calculated. The harmonic analysis does not take into consideration the transient vibrations of the structure, obtained at the beginning of the running process.

The harmonic reaction is linear. Any discontinuity such as plasticity and interstitial contact between elements will not be considered even if they have been defined. This analysis can also be used for a pre-tensioned structure.

The model and harmonic analysis elaboration and design requires the same set of directives like in the case of the finite element analysis. Moreover, the same options can be employed for the graphic interface (**GUI**).

The harmonic analysis consists of three methods: full, reduced and super-positioned. A fourth relatively costly method is the transient dynamic analysis based on specific frequency harmonic loads. The superposition method can be performed only through the ANSYS/LinearPlus mode.

The full method is the easiest. It is based on the full set of matrices, be they symmetrical or not. The reduced method condenses the dimension by means of degrees of freedom and reduced matrices. Once the displacements and degrees of freedom obtained, the solution can expand the full original set of degrees of freedom.

The advantages of this method are as follows:

- is quick and less costly than the full method;
- pre-tensioning effects can be included such as obliquely aligned active elements.

The present research deals with the reduced method for ultrasonic motors. The harmonic reaction analysis for the reduced method consists of:

- *model construction.* The title of analysis and digitization type of element, the real parameters and material properties as well as the model geometry are specified;
- *charging and solution generation.* Defining master DOF (master degrees of freedom) – essential or dynamic degrees of freedom that outline the dynamic behavior of the structure. The following alternatives are to be used to calculate the equations: frontal solver, Jacobi Conjugate Gradient (JCG), Incomplete Choleski Conjugate Gradient (ICCG). By definition, a harmonic analysis implies sinusoidal variation in time of any applied charge.

Three components are required: amplitude, phase and the frequency domain;

- *results***.** The results of the reduced analysis represent master DOF displacements, harmonically varying for each frequency whose solution has been determined.

## 2. FINITE ELEMENT BASED MODELING OF THE SECTOR PIEZOELEMENT FOR THE MUL-1OC LINEAR ULTRASONIC MOTOR

The active element of the stator assembly of the linear ultrasonic motor is a sector piezoelement.

Modeling of its running through ***Ansys*** is based both on a modal analysis so as to determine the frequencies of the vibration as well as on a harmonic analysis in view of determining the model's nodal displacements.

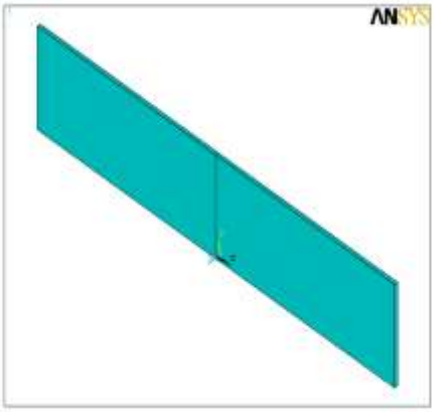
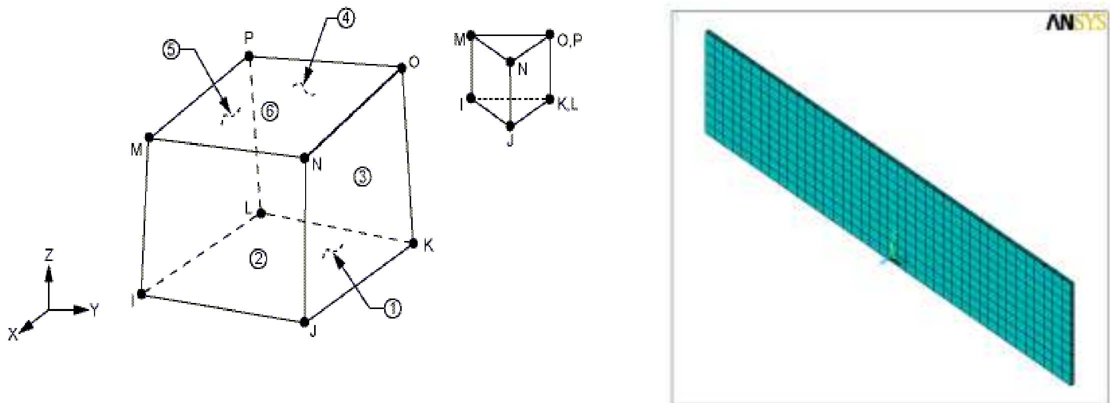


Fig. 1 Volume and areas of the laminated piezoelement.

The following phases have been performed :

**- construction of the model.** For the construction of the model based on the finite element analysis of the sector laminated piezoelement (the active element of the linear ultrasonic motor), first, its geometry must be designed. A 60 x 12 x 0.5 mm prismatic volume is generated in the Cartesian coordinate system as shown in Figure 1.

Digitization in finite elements of the structure (fig.2) is achieved by means of the ***Solid 5*** tetrahedral element, displaying the material properties of the PZT 5 piezoelectric element.



a b

Fig. 2 Finite element digitization of the laminated piezoelement:
a – Solid 5 finite element 5; b – finite element digitization of the lamella.

**- modal analysis.** The modal analysis used to determine the vibration characteristics of the structure is based on the reduced method**.** It aims at determining the frequencies of the vibration modes as well as the corresponding deformation. There have been calculated the first 30 vibration modes for the present research case. Once the solution obtained, the vibration mode is selected in the case of a sector laminated piezoelement characterized by a 24.32 KHz frequency traveling wave as shown in Figure 3.

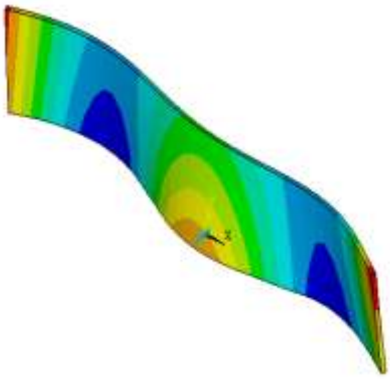


Fig. 3 Vibration mode for the 24.32 KHz.

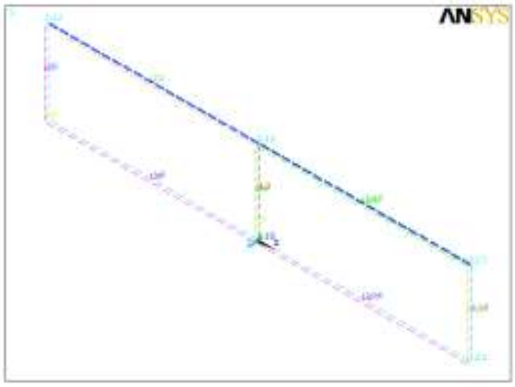


Fig. 4. Selection of corresponding lines for two sector edges.

**- harmonic analysis.** The harmonic analysis is performed for one frequency only. This is one of the frequencies obtained through the modal analysis and it produces a traveling wave type of vibration.

In this case, dephased harmonic tensions are applied to the electrodes of the sector laminated ceramic piezoelement.

The harmonic analysis was carried out for the 24.32 KHz. For this frequency, the vibration of the sector laminated piezoelement triggers a traveling wave of rather high amplitude so as to drive the actuator.

Once the harmonic analysis completed, the corresponding lines are selected for the edges of two sectors as shown in Figure 4.

**- results.** The corresponding nodes for these lines are selected and the displacement values on the OZ axis are listed as shown in Table 1.

**Table 1.** *Displacement values in nodes on the OZ axis.*

| **No. Node** | **Displacement [µm]** | **No. Node** | **Displacement [µm]** |
|---|---|---|---|
| 1 | 0.047 | 21 | 0.009 |
| 2 | 0.035 | 22 | 0.008 |
| 3 | 0.023 | 23 | 0.007 |
| 4 | 0.013 | 24 | 0.004 |
| 5 | 0.004 | 25 | 0.001 |
| 6 | -0.004 | 26 | -0.002 |
| 7 | -0.010 | 27 | -0.006 |
| 8 | -0.014 | 28 | -0.010 |
| 9 | -0,017 | 29 | -0.014 |
| 10 | -0.018 | 30 | -0.017 |

| | | | |
|---|---|---|---|
| 11 | -0.018 | 31 | -0.019 |
| 12 | -0.016 | 32 | -0.02 |
| 13 | -0.014 | 33 | -0.019 |
| 14 | -0.010 | 34 | -0.016 |
| 15 | -0.006 | 35 | -0.011 |
| 16 | -0.003 | 36 | -0.004 |
| 17 | 0.001 | 37 | 0.005 |
| 18 | 0.004 | 38 | 0.016 |
| 19 | 0.007 | 39 | 0.028 |
| 20 | 0.008 | 40 | 0.040 |

These values (as illustrated in Table 1) help draft the nodal displacement graph (see Figure 5) for a 100 V driving tension of the laminated piezoelement. The harmonic displacement wave is marked. The displacements on the OZ axis display submicronic values. The turret drive assembled with the laminated piezoelement will trigger similar or even smaller displacements.

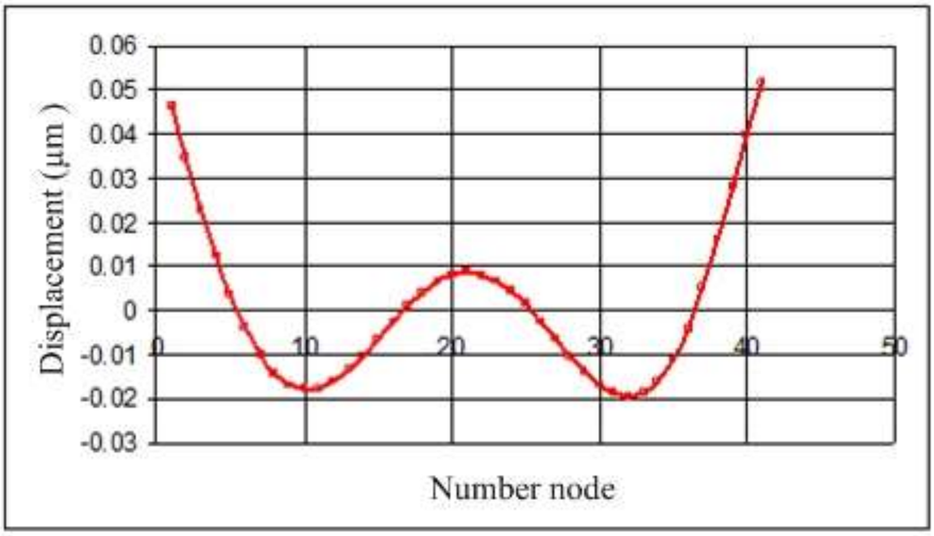



Fig. 5. The nodal displacement variation of the laminated piezoelement edges.

## 3. THE FINITE ELEMENT MODELING OF THE TURRET DRIVE ACTUATOR

The second element of the drive system of the MUL-1OC linear ultrasonic motor is the turret drive actuator. This is deformed by the sector laminated piezoelement.

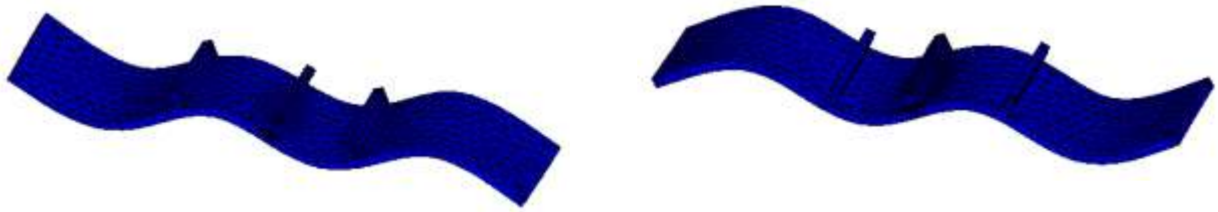
Once the volume geometry of the turret drive actuator devised (Figure 6), the digitization of the ***Solid 45*** element is initialized (Figure 6, b).

a b

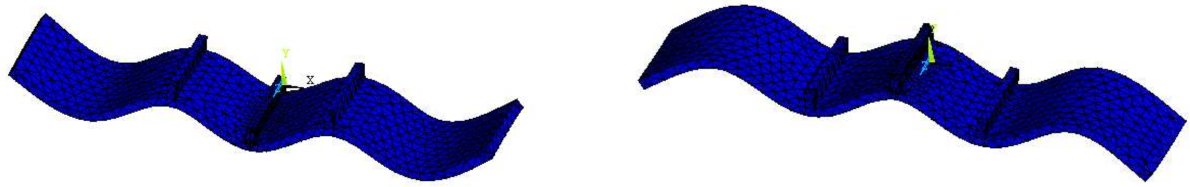
Fig. 6. The turret actuator of the drive system :
a – volume of the turret drive actuator; b – digitization of the turret drive actuator.

The modal analysis is performed so as to achieve two „traveling” wave type of vibration modes, for the 24.342 KHz and 33.862 KHz frequencies. The two vibration modes are illustrated in Figure 7. The number 13 mode of vibration of the turret drive actuator displays a frequency close to that of the sector laminated piezoelement.

The teeth of the turret drive actuator, situated on the traveling wave, have a forward and backward reciprocal movement and, therefore, they can drive, by friction, the drive actuator.



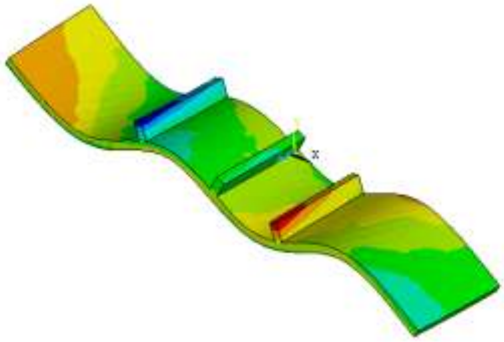
a

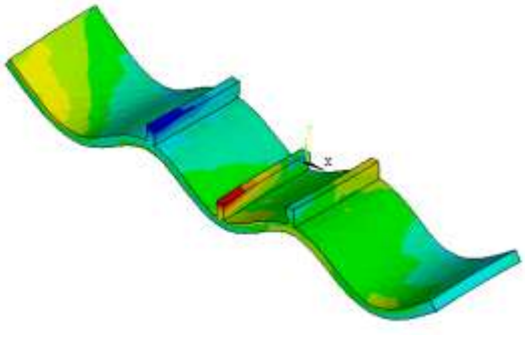


b

Fig. 7. Deformation of the turret drive actuator for the two vibration modes:
a – vibration mode 13 for 24.342KHz; b – vibration mode 15 for 33.862KHz.

Figure 8 illustrates the nodal solutions on the OZ axis displacement for the two modes in which the turret drive actuator triggers a traveling wave.





a b

Fig. 8 Nodal solutions for OZ displacement:
a – vibration mode 13for f = 23.34KHz; b –vibration mode 15 f = 33.86 KHz.

## 4. CONCLUSIONS

1° The finite element method ensures the study of both static and dynamic behavior of the active and activated elements in the case of linear ultrasonic motors;

2° The modal analysis brings about the frequencies at which both the piezoelements and the other active elements of the ultrasonic motors display a "traveling wave" type of vibration mode;

3° The harmonic analysis deals with modal displacements in various directions, the most representative is the running direction;

4° The input data for the harmonic analysis are garnered through measurements and the results are compared to the corresponding dimensions of the theoretical results;

5° In the case of the linear ultrasonic motor, the two elements of the drive system (the sector laminated piezoelement and the turret drive actuator) are modeled according to the finite element method.

## REFERENCES


**[1]**AMZA, Gh., ş.a. - Ultrasunete de mari energii, Editura Academiei R.S.R, Bucureşti, 1984;

**[2]**AMZA, Gh., ş.a. - Tratat de tehnologia materialelor, Editura Academiei, Bucureşti, 2003;

[3] *AMZA, Gh., Aplicaţiile ultrasunetelor, vol. 3, Motoare ultrasonice, Editura Bren, Bucureşti, 2005.*

[4]. *AMZA, Gh., Ignat, M., Niţoi, D., Borda, C., - Motoare ultrasonore pe baza de materiale magnetostrictive inteligente, Construcţia de Maşini, nr. 8-9, 1997.*

[5] *AMZA, Gh., Niţoi, D., Elements for designs and modeling of an ultrasonic motor, Timişoara, 2001.*

[6]A.LANGELLA, Crivelly Visconti, Technological Characteristics of a New Optical Sensor for Smart Composites, Advanced Composite Materials, DetroitMichigan, 1991.

[7]ABRAHAM SALEHI, Andrew Tay, Strain Concentrations Arouund Embedded Optical Fibers by FEM and Moire Interferometry, Advanced Composite Materials, DetroitMichigan, 1991.

[8] ACHENBACH, J., Fang, S., Asymtotic analysis of the modes of wave propagation in solid cylinder, j. Acoust. Soc. Am. 1970.

[9]ADACHI, M., Shiosaki, T., Temperature compensated piezoelectric lithiuim tetraborate cristal for hight frequency surface acoustic wave and bulk wave device application, in Proc. 1985.

[10] BALTRUKONIS, J., Gottenberg, W., Thickness shear Vibrations of circular bars, J. Acoust. Soc. A., 1959.

[11]CHENG, M., Finite element analysis of finite axis symmetric cylinders, M.S. Thesis, Tennesse Tech., Univ., Cookeville, 1998.
[12]DANCOV, I., Tokarev, E., Elastic, piezoelectric and dielectric properties of monocristalline lithium niobiate between 78 and 800 K, Inorganic Materials, 1983.

[13]A. LANGELLA, Crivelly Visconti, Technological Characteristics of a New Optical Sensor for Smart Composites, Advanced Composite Materials, DetroitMichigan, 1991.
[14] *Hirata, H.*, and *Ueha, S*. 1995. Design of a traveling wave type ultrasonic motor. In *IEEE Transactions on Ultrasonic, Ferroelectrics, and Frequency Control*, pages 225-231, Vol. 42, No. 2, March 1995.