DETERMINATION OF THE STRESS AND STRAIN STATE IN METALLIC STRUCTURES USING PIEZOELECTRICAL TRANSDUCERS

D. Popovici[1] , V. Păltânea[1] , G. Păltânea[1] , G. Jiga[2]
"Politehnica" University of Bucharest,
[1] Department of Electrical Engineering, [2]Department of Strength of materials

Abstract

In this paper is presented a modern non-destructive method for the evaluation of the stress and strain state in metallic structures. This method can be successfully applied in civil engineering, installations or in structures where the determination of stress state can not be realized by conventional method.

1 The mathematical model

The piezoelectric effect implies the conversion of electrical to mechanical energy and vice–versa. It is observed in many crystalline materials, such as quartz, Rochelle salt and lead titanate zirconate ceramics, which display the phenomenon strongly enough to use it. The direct piezoelectric effect consists of an electric polarization in a fixed direction when the piezoelectric crystal is deformed. The polarization is proportional to the deformation and generates an electrical voltage over the crystal. The inverse piezoelectric effect is the opposite of the direct effect: an applied electric field induces a deformation of the crystal. In our case the cantilever beam deformation are transform in electrical voltages trough a direct piezoelectric effect [2].

The direct piezoelectric effect is described by Navier's equations for structural mechanics (mechanical stress),

$$-\nabla \mathbf{T} = \mathbf{K} \tag{1}$$

where **T** is the stress tensor field:

$$\mathbf{T} = \begin{bmatrix} T_1 & T_4 & T_6 \\ T_4 & T_2 & T_5 \\ T_6 & T_5 & T_3 \end{bmatrix} \tag{2}$$

and by Gauss law of electrostatics:

$$-\nabla \mathbf{D} = \rho \tag{3}$$

Using a compress matrix notation, the constitutive equations that describe the piezoelectric effect can be written in the stress-charge form as follow:

$$\begin{aligned} \mathbf{T} &= c_E\,\mathbf{S} - e^T\,\mathbf{E}, \\ \mathbf{D} &= e\,\mathbf{S} + \varepsilon_S\,\mathbf{E} \end{aligned} \tag{4}$$

where **S** is strain field, **E**, electric field, c_E, elasticity matrix, e, coupling matrix, ***D***, electric displacement field, ε_S, the permittivity matrix. The subscripts, E and S indicate at constant electric field and strain, respectively [1].

The stress-charge form is selected for the constitutive equation as this suits the form in which the material data is given.

2 Geometry of the model

To simulate the structure we have chosen a multiphysics problem: plane stress and piezo plane stress. The geometry used is presented in figure below:

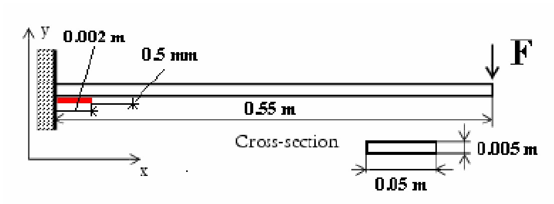

Figure 1: The geometry of the problem.

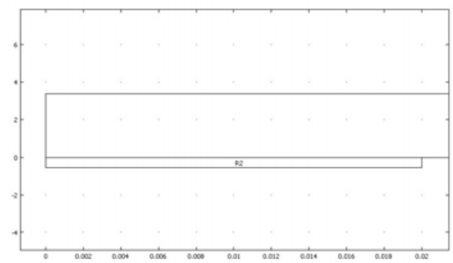

Figure 2: Zoom in the PZT cell section.

The domain R1 is an isotropic structural steel beam with a length of 550 mm, width of 50 mm and thickness of 5 mm. This material is defined in Library 1 of [COMSOL Multiphysics] [1]. The domain R2 is the PZT 5H cell which has a length 20 mm, width of 50 mm and thickness of 0,5 mm.

For the structural steel we used the following material constants: $E = 2 \cdot 10^5$ [MPa], Poisson's ratio $\nu = 0.33$ and density $\rho = 7850$ kg/m^3.

The PZT – 5H properties are those listed in [COMSOL]:

$$c = \begin{bmatrix} 126 & 80,5 & 84,1 & 0 & 0 & 0 \\ & 126 & 84,1 & 0 & 0 & 0 \\ & & 117 & 0 & 0 & 0 \\ & & & 23,3 & 0 & 0 \\ & \text{symm} & & & 23 & 0 \\ & & & & & 23 \end{bmatrix} [\text{GPa}];$$

$$e = \begin{bmatrix} 0 & 0 & 0 & 0 & 17 & 0 \\ 0 & 0 & 0 & 17 & 0 & 0 \\ -6,5 & -6,5 & 23,3 & 0 & 0 & 0 \end{bmatrix} [\frac{C}{m^2}]; \ \varepsilon_r = \begin{bmatrix} 1704 & 0 & 0 \\ 0 & 1704 & 0 \\ 0 & 0 & 1433 \end{bmatrix}$$

The boundary conditions resulted from the working conditions. For the mechanical part of the problem a constraint of zero movement on the left side of the beam and the PZT cell has been considered. The load was applied on the right end of the beam only on the y direction.

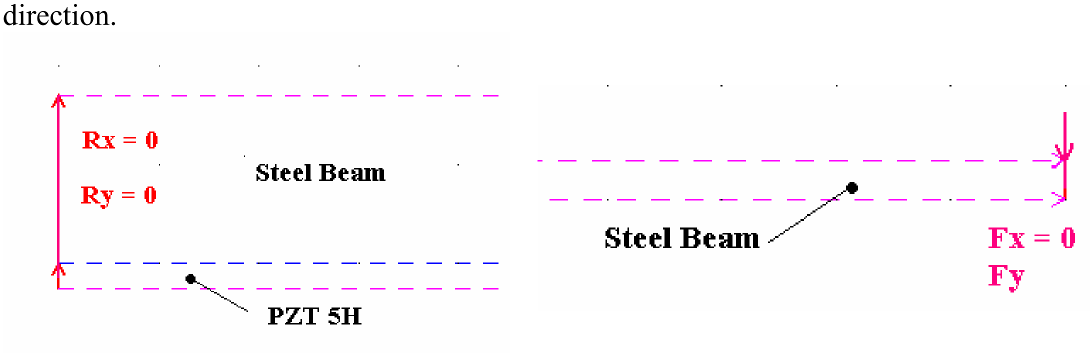

Figure 3: The mechanical boundary conditions.

In the case of the electrical aspect of the problem we set the horizontal bottom surface of the PZT cell to ground and the top surface was consider a zero charge/symmetry condition.

Applying the FEM method, the meshed model contains a number of 3516 triangular elements.

3 Results

In the paper three basic analysis types available in the Structural Mechanics Module have been taken into consideration:

- Static;
- Eigenfrequency;
- Time dependence.

At first, a statical analysis has been made, where a uniform distributed load has been applied at the right end of the beam. This force has only a vertical component $F_y = 100$ N. A stationary linear solver direct UMFPACK has been used. In Fig. 4, 5, 6 are represented the displacements along the y axis, the maximum value recorded at the end of the cantilevered beam respectively the voltage recorded for this model.

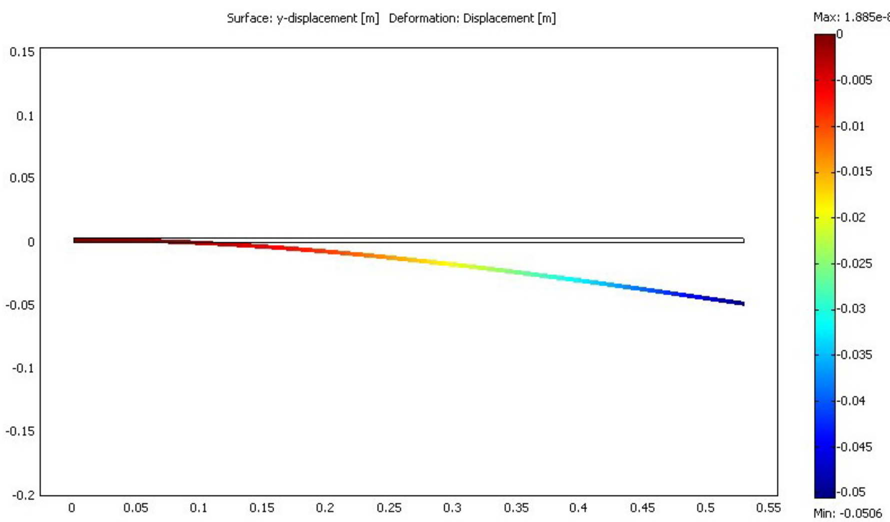

Figure 4: Displacement of the beam on the y axis for $F_y = 100$ [N].

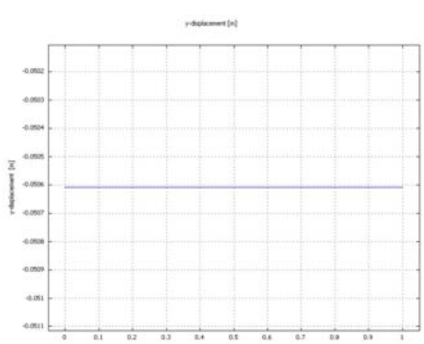

Figure 5: The displacement of the right end of the beam (y = 0,050 [m]).

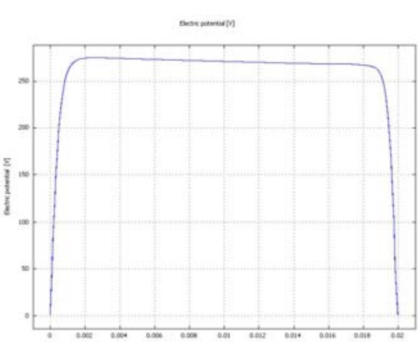

Figure 6: The voltage measured with the PZT 5H crystal (V ~ 275 [V]) .

The maximum stress calculated with Von Mises criteria has been determined in the left side of the beam (in the vicinity of the clamping side) and was equal to 294 [MPa] and the minimum value in the right side of the beam 0,168 [MPa].

The voltage response of the PZT cell at different loads ($F_y \in \{5, 10, 25, 50, 75, 100\}$ [N]) has a linear variation as one can see in Figure 7. For the same loads we determined the displacement on y axis of the right side of the beam and the maximum stress values which has the same linear variation.

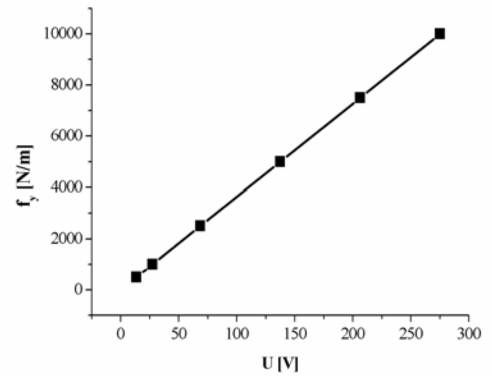

Figure 7: The linear dependence F(U).

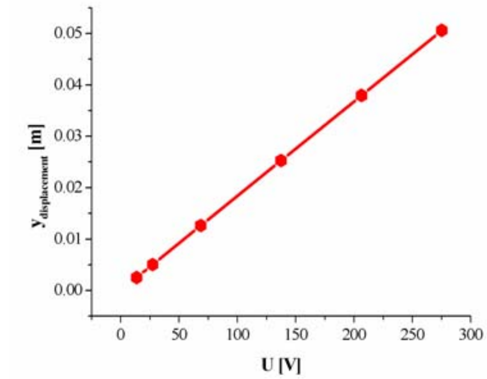

Figure 8: Displacement of the right end of the beam.

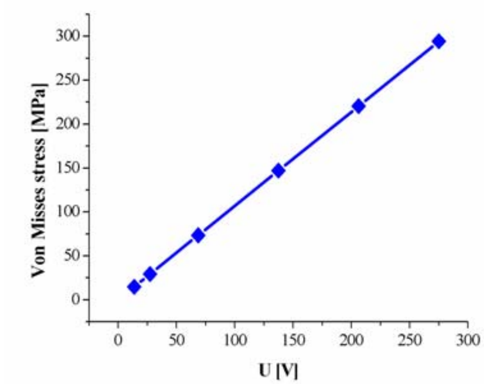

Figure 9: The maximum stress values.

Table 1: VALUES DETERMINED IN STATICAL ANALYSIS

F_y [N]	Von Mises stress [MPa]	$y_{displacement}$ [mm]	U [V]
5	14,71	2,53	13,75
10	29,42	5,06	27,51
25	73,55	12,65	68,77
50	147,1	25,30	137,55
75	220,6	37,95	206,33
100	294,2	50,61	275,10

An eigenfrequency analysis finds the eigenfrequencies and modes of deformation of the analyzed structure. The eigenfrequencies f in the structural mechanics field is related to the eigenvalues λ returned by the solvers through:

$$f = \frac{\sqrt{\lambda}}{2\pi} \tag{5}$$

The purpose of the eigenfrequency analysis is to find the six lowest eigenfrequencies and their corresponding shape modes. This model uses the same material, load and constraints as the statical analysis. A linear system solver direct Umfpack system was used and the following are presented below:

Table 2: THE FIRST SIX EIGENFREQUENCIES OF THE MODEL

f_1	f_2	f_3	f_4	f_5	f_6
9,98 Hz	64,48 Hz	174,73 Hz	341,91 Hz	564,31 Hz	841,75 Hz

A time-dependence analysis for the transient solution of the displacements and velocities as functions of time was applied. In this case, the material properties, loads and boundary conditions are function of time. The purpose of this analysis was to find the transient response from a harmonic load with the same amplitude as the static load during the first two periods. The excitation frequency has been taken of 50 Hz, which is between the first and second eigenfrequency found in the eigenfrequency analysis.

A harmonic load $F_x(t) = 0$ and $F_y(t) = 100 \sin(100 \pi t)$ [N] has been used. Damping is very important in transient analysis but difficult to model. The Structural Mechanics Module supports Rayleigh damping, specifying damping parameters proportional to the mass (α_{dM}) and stiffness (β_{dK}) in the following way:

$$C = \alpha_{dM} M + \beta_{dK} K \tag{6}$$

where C is the damping matrix, M is the mass matrix, and K is the stiffness matrix. One calculated according to [1] the mass damping parameter and the stiffness damping parameter. The damping parameters have been considered at their default values in the previous analysis, due to the fact they were only been used for transient and frequency response analysis. The structure has a constant damping ratio of 0,1. Two frequencies near the excitation frequency, 20 respectively 60 Hz, have been considered, to calculate the damping parameters, according to the FEMLAB code [1]:
$\alpha_{dM} = 18{,}849$ 1/s and $\beta_{dK} = 3{,}979 \cdot 10^{-4}$ s.

This problem was computed using a time dependent solver with the time set on the interval [0; 0,08][s] with a step of 0,001, a relative tolerance of 0,05 and an absolute tolerance of 10^{-9}.
The following waweformes for the displacement on the x and y axis are represented in Fig.10 and 11.

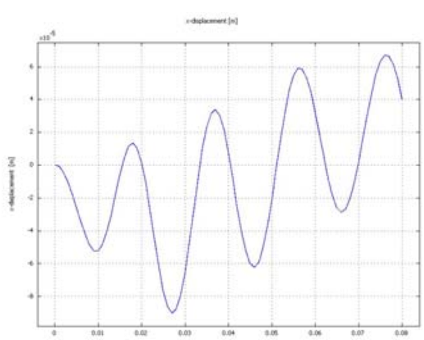

Figure 10: The horizontal (x) displacement of the right end of the beam.

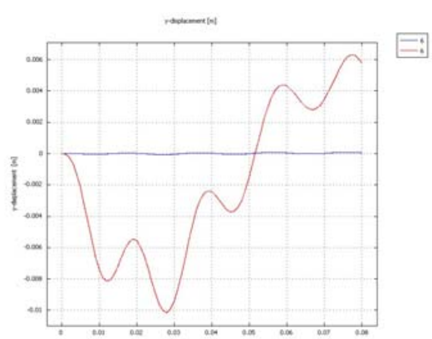

Figure 11: The horizontal (*x)* displacement and the vertical (*y*) displacement of the right end of the beam.

The voltage output measured on the PZT cristal has a sinusoidal figure with a maximum value of 182,072 [V] and a minimum value of -150,962 V.

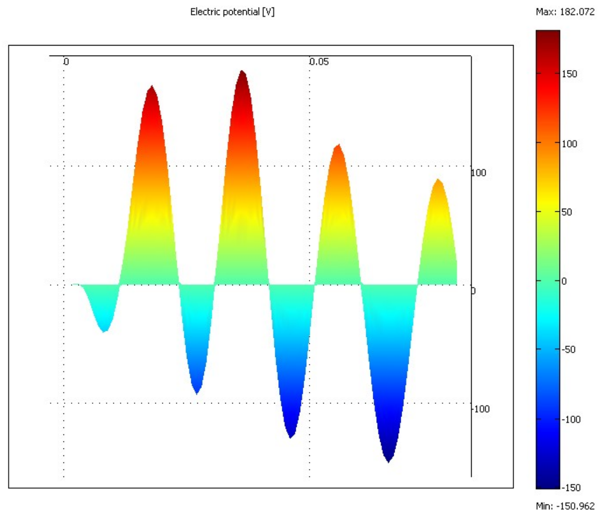

Figure 12: The waveform of the voltage (the horizontal axis is time and the vertical axis is voltage similar to electric potential)

For a more accurate solution the time interval has been increased from 0 to 0,2 [s] and in this case, the total displacement and the equivalent voltage response are represented in Fig.13 and 14.

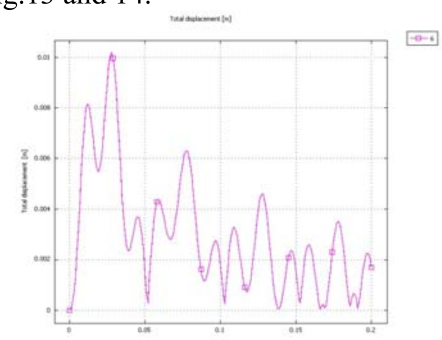

Figure 13: The total displacement of the right side of the beam.

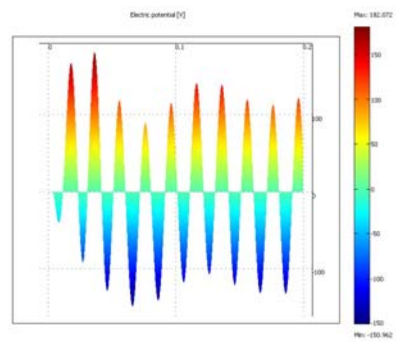

Figure 14: The voltage output on the PZT.

4 Conclusions

The paper presents a theoretical model, able to give an accurate prediction of the displacement of a steel beam after impact by intermediate of the voltage response. The theoretical model approximates very well all experimental data. Finally, the transient response from a harmonic load has been measured on a on the PZT crystal. This method could be further developed in civil engineering, installations or in big structures where the determination of stress state can not be realized easily using the non-conventional methods.

References

[1] COMSOL 3.2., Structural Mechanics, Manual, Comsol, AB, Sweden (2005).

[2] A. Morega, V. Mănescu, G. Păltânea, *A FEM Static Model for a Piezoelectric Actuator Microgripper,* Proceedings of the 4th International Workshop, Materials for Electrotechnics, p. 240 – p.245, Bucharest, Romania, 2004.

Author1
Contact information e-mail:" Dorina Popovici " karina_popovici@yahoo.com
Author2
Contact information e-mail "Veronica Manescu" m1vera2@yahoo.com