Lehrstuhl für Sensorik
Universität Erlangen-Nürnberg
Daniel Kiefer

Assignment 2: Piezoelectric Coupling – Bimorph Actuator

Piezoelectric materials allow a transformation between electrical and mechanical energy. This effect is utilized in several sensor and actuator applications. Thereby, actuators use the inverse piezoelectric effect, i.e., the transformation of electrical energy (in form of applied voltages or charges) into mechanical deformations and stresses. To obtain significant deformations, however, one has to apply very large electric fields which leads to non-linear and hysteretic effects inside the piezoelectric material. Due to the complicated material behavior it is desirable to avoid this non-linear regime. Another possibility to increase the range of deformations is to combine multiple piezoelectric actuators either in form of stacks or so-called bimorphs. In this assignment, the later setup shall be investigated.

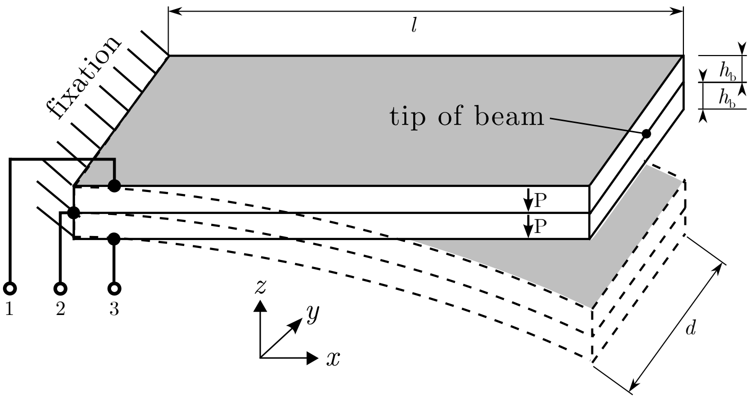

Figure 1: Parallel piezoelectric bimorph actuator

Figure 1 shows a parallel piezoelectric bimorph actuator. It consists of two piezoelectric plates which are polarized in the same direction and which are glued together along one common electrode. Driving the two glued plates with appropriate electric fields will lead to a bending of the whole actuator as depicted in the figure. This effect is used in injection valves, acoustic generators (buzzers), etc. For the given assignment, both plates shall be made of PZT-5H and have a height h_b, a length l and a depth d. The overall assembly is fixed at the left hand side (over the whole surface). Furthermore, the top, bottom and gluing surfaces are metalized and shall be considered as ideal electrodes.

General hints and remarks

- You should always correctly **label** the axes of the plots and add a **legend** if necessary. Format plots so that they can be **read easily**.
- **Save plots** as a `.png`-file before proceeding to the next task, as you will need to submit these.
- Write your **short answers** to all questions to a `Results.pdf` file (label each answer with its task number). Include your plots in this file.
- If you are unsure on how to perform the tasks, refer to
  - COMSOL Tutorial
  - COMSOL Multiphysics Reference Manual

1 Modeling

1.1 Geometry setup

1.1. Use the model wizard to create a 3D geometry and use the *Structural Mechanics→Piezoelectricity* physics. This Comsol template will add (1) a *Solid Mechanics* physics node, (2) a *Electrostatics* physics node and (3) an *Piezoelectric Effect* Multiphysics node. The latter node models the coupling between two different physical domains. Instead of using the model wizard, you could add the three nodes manually. In the last step of the model wizard, add a *Stationary* study.

1.2. Model the geometry similar to Fig. 1 but with only half the depth d. You will use a symmetry boundary condition in your physics to represent the other half. Define all required dimensions as `Parameters` and use the following values:

- $h_\mathrm{b} = 0.25\,\mathrm{mm}$
- $l = 25\,\mathrm{mm}$
- $d = 2.5\,\mathrm{mm}$

Make sure to align the center electrode (the mid-thickness of the bimorph) with $z = 0$.

1.3. Add *PZT-5H* as piezoelectric material and assign it to the whole domain.

1.4. Make sure that the *Solid Mechanics*, *Electrostatics* and *Piezoelectric Effect* physics are applied to the whole geometry. Also make sure, that the correct physics are being coupled in the *Piezoelectric Effect* node.

1.2 Solid Mechanics setup

1.5. Disclose the *Solid Mechanics* node. Note that the *Linear Elastic Material* node is being overwritten by the *Piezoelectric Material* node (as indicated by the red triangle in the nodes icon as well as in the *Override and Contribution* section of the settings). Inspect the equations.

1.5.1. Are the equations large- or small-signal formulations?

1.5.2. Which form of piezoelectric material behavior does Comsol use?

Go to the *Piezoelectric Material Properties* section. Under *constitutive equation* you can change the piezoelectric formulation (each form uses different material properties which you can input below).

1.5.1. How do the equations in the *Equation* section change when changing the constitutive relation? According to this, does Comsol actually use different piezoelectric formulations at the time of simulation?

1.5.2. What are the SI units of the piezoelectric coupling matrix in strain-charge form?

1.6. In the *Piezoelectric Material* node, make sure to check *force linear strains* (to simulate linear mechanics).

1.7. Add a *Symmetry* boundary condition to consider the half of the geometry that has not been modeled.

1.8. *Fix* the beam at the clamping (the left side).

1.3 Electrostatics setup

1.9. Disclose the *Electrostatics* node. Note that the *Charge Conservation* node is being overwritten by the *Charge Conservation, Piezoelectric* node. Inspect the equations in both cases.

1.9.1. How does the equation change for the piezoelectric case?

Note that all piezoelectric material parameters, including the electrical ones, have already been set in the *Solid Mechanics→Piezoelectric Material* node.

1.10. To excite the piezoelectric bimorph, ground the middle electrode and impose 500 V on the two outer electrodes.

1.4 Meshing

1.11. Mesh the 3D geometry with

- $N_x = 25$ elements in the length l
- $N_y = 5$ elements in the depth d
- $N_z = 4$ elements for each thickness h_b

1.11.1. You can do this by applying a *Mapped* mesh to the entire geometry and

1.11.2. adding three *Distribution* nodes to it. The latter are used to specify the number of elements on appropriately selected lines. Note that the *Mapped* mesh only meshes boundary elements (surfaces), not the domain/volume elements.

1.11.3. Add a *Swept* node to the mesh. This extrudes the boundary elements into domain elements.

1.12. Click on *Build All*. You will get domain, boundary and edge elements, as required.

2 Static Simulation

2.1. Perform a static/stationary simulation with the excitation as defined above.

2.1.1. Across the thickness of the beam, do you find a point where the stresses are zero? Where is this point (i.e., which x-y-plane)?

2.1.2. Inspect the electric potential across the plate's thickness, i.e., on a line along the z-direction. For this plot the electric potential along the line going through the beam's tip. For the x-data use the z-coordinate (not the arc length). Does the solution on this line satisfy the boundary conditions? Is the solution as expected?

2.1.3. Find out how the two plates elongate/shrink in x-direction. Over the same line as above, plot the displacement in x-direction. Again, plot the data over the z-coordinate – not the arc length. This way you can tell for sure, which part of the data corresponds to the two piezoelectric plates. Which one elongates? Which one shrinks?

In the following, analyze the importance of the polarization of the piezoelectric material. Remember that piezoelectric materials are polarized in 3-direction per convention. Comsol aligns this to the global z-coordinate per default.

2.2. You first need to define a rotated coordinate system. Go to *Component 1→Definitions* and add a *Rotated System*.

2.3. In the settings, make the new z-direction point towards the old $-z$-direction. The angles α, β and γ mean: rotated over the current z-axis, then rotate over the current x-axis, then again rotate over the current z-axis. If you need help, press F1 to invoke the Comsol help. As the piezoelectric material is transversely isotropic, it does not matter if the new x and y coordinates do not match the old ones.

2.4. Go to the *Piezoelectric Material 1* node and under *Coordinate System Selection* choose the newly created coordinate system. The piezoelectric material's 1, 2 and 3 axes are aligned to the x, y and z axes of the coordinate system that you select in this section.

2.5. Perform the *Stationary* simulation again.

2.5.1. How do the displacements change?

2.5.2. Does the electric potential along the line plot of task 2.1.2. change at all? Why?

2.5.3. Are you using the direct or the inverse piezoelectric effect?

3 Harmonic Simulation

For the following harmonic simulations, use 30 frequency points per decade between 100 Hz and 1 kHz. Use the same excitation as for the static simulation (the voltages are now interpreted as amplitudes of harmonic signals).

First, determine the frequency response for the undamped case.

3.1. Set up a *Frequency Domain* study and perform the computation. This may take a couple of minutes.

3.2. Plot the absolute value of the z-displacement of the beam's tip at the middle line over frequency. Make the plot double logarithmic. At what frequency is the resonance located?

3.3. Go back to the *Stress (solid) 1* plot created by default by Comsol and visualize the displacement at the closest frequency to the resonance peak. How are displacements different below the resonance and above the resonance? Which case would you think is in-phase with the excitation and which one out-of-phase?

Now, compute the frequency response including Rayleigh damping.

3.4. Define a new *Parameter* named `xi` for the desired damping ratio and set it to 0.0001.

3.5. Go the the *Piezoelectric Material 1* node and add *Mechanical Damping*. As Damping type choose *Rayleigh damping* and as input parameters choose *Damping ratios*. Set the frequencies `f1` and `f2` to 300 kHz and 500 kHz, respectively. Specify the damping ratio at those frequencies as the parameter `xi` you defined in task 3.4. Sketch qualitatively, how the damping ratio over frequency plot looks for some non-zero value of `xi`.

3.6. Add a *Parametric Sweep* to your frequency dependent *Study*. Use it to sweep `xi` over the values: 0, 0.0001, 0.0005 and 0.002.

3.7. Run the simulation. This might take about 15 min. to 20 min.

3.8. Make a double-logarithmic plot of the tip's $|u_z|$ displacement over frequency for all damping cases. Add a legend to distinguish the curves.

3.8.1. How does the resonant frequency change with increased damping? Does it change a lot?

3.8.2. How does the maximum at resonance change with increased damping?

3.8.3. How does the width of the resonance peak change?

3.8.4. The parameter `xi` only specifies the damping ratio at the two frequency points `f1` and `f2`. The overall damping might be different. According to the plot, which values of `xi` result in an overall damping below "critical damping"? Which ones above?

4 Transient simulation

Analyze how the piezoelectric bimorph responds to an electric impulse/spike:

4.1. Add a transient voltage excitation to the simulation with the same amplitudes as before but with a rectangular shaped pulse. The duration of the pulse shall be approximately one-fifth of the period of the first resonance.

4.2. Add a *Time Dependent* study and use the following values for the simulation:

- $T_{\max} = 20\,\mathrm{ms}$
- $\Delta t = 25\,\mathrm{\mu s}$
- `manual` time stepping (Right click on *Solver Configurations* and select *Show default solver*. Then go to *solution x→Time-Dependent Solver 1* in the section *Time Stepping* specify `dt` as *manual* time step for the solver).

4.3. Add a *Parametric Sweep* to the study and set the values of `xi` to 0 and 0.0001.

4.4. Perform the simulation. This may take about 15 min.

4.5. Create a plot showing the u_z-displacement of the tip over time (both linear) for both damping ratio values. Add an appropriate legend.

4.6. Visualize the transient deformation of the beam by creating an animation of the default Comsol plot called *Stress (solid) x*.

Submitting your assignment

Hand in your

- Comsol `.mph` file and
- a `Results.pdf` file with short answers to all questions within this assignment sheet (label each answer with its task number). All plots you are asked to create should also be contained in this file at the position of the corresponding task. Animations should be reduced to just one frame at a "representative" time. You can, for example, use Word or LaTeX to produce the PDF file.

Upload the above files to the corresponding Assignment section on StudOn:
https://www.studon.fau.de/exc2992793.html.

Important: Make sure to delete all results and meshes from the Comsol file before handing in. This reduces the file size. Refer to the Comsol Tutorial on how to do this.