COMSOL MULTIPHYSICS®

Piezoceramic Tube

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Piezoceramic Tube

Introduction

This example involves a static 2D axisymmetric analysis of a piezoelectric actuator using the 2D Piezo Axial Symmetry application mode. It models a radially polarized piezoelectric tube, as described by S. Peelamedu and others (Ref. 1). A application area where radially polarized tubes are employed is in nozzles for fluid control in inkjet printers.

Model Definition

GEOMETRY

The tube has a height of 0.62 mm and an inner and outer radius of 0.38 mm and 0.62 mm, respectively.

MATERIAL

The material properties for the piezoceramic (PZT-5H) are:

$$c = \begin{bmatrix} 127 & 80{,}2 & 84{,}7 & 0 & 0 & 0 \\ & 127 & 84{,}7 & 0 & 0 & 0 \\ & & 117 & 0 & 0 & 0 \\ & & & 23{,}0 & 0 & 0 \\ & \text{sym} & & & 23{,}0 & 0 \\ & & & & & 23{,}5 \end{bmatrix} \text{GPa}$$

$$e = \begin{bmatrix} 0 & 0 & 0 & 0 & 17{,}03448 & 0 \\ 0 & 0 & 0 & 17{,}0345 & 0 & 0 \\ -6{,}62812 & -6{,}62812 & 23{,}24031 & 0 & 0 & 0 \end{bmatrix} \text{C/m}^2$$

$$\varepsilon = \begin{bmatrix} 1{,}5 & 0 & 0 \\ 0 & 1{,}5 & 0 \\ 0 & 0 & 1{,}3 \end{bmatrix} \cdot 10^{-8} \text{ F/m}$$

BOUNDARY CONDITIONS

The model studies two cases, distinguished by the boundary conditions. Case 1 represents the inverse piezoelectric effect, and Case 2 represents the direct piezoelectric effect.

Case 1:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z-direction).
- Electrostatics boundary condition—apply a 1 V potential difference between the tube's inner and outer surfaces.

Case 2:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z direction), but also add an internal fluid pressure of 0.1 MPa.
- Electrostatics boundary condition—ground the inner and outer surfaces.

Results and Discussion

The image in Figure 1 shows the deformation due to the applied voltage difference in Case 1. For the same case, Figure 2 shows the radial displacement as a function of the tube thickness at the top boundary.

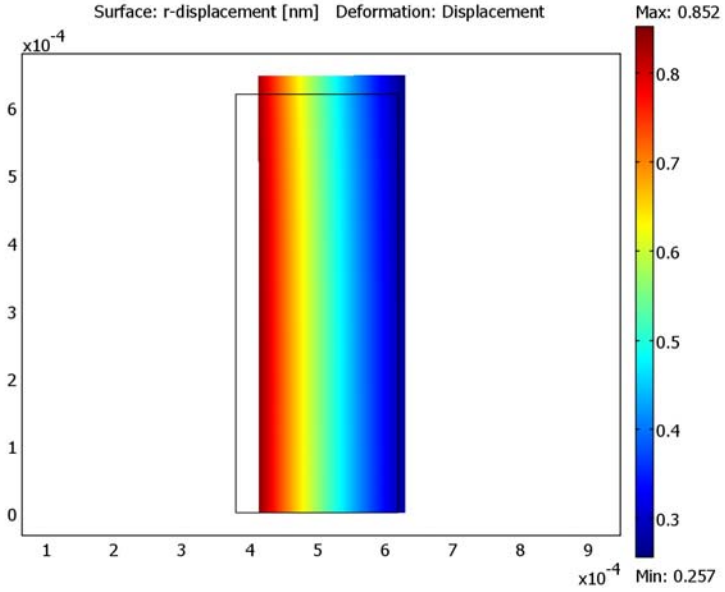


Figure 1: Deformed shape and radial displacement of the piezoceramic-tube actuator due to the radial electric field (Case 1).

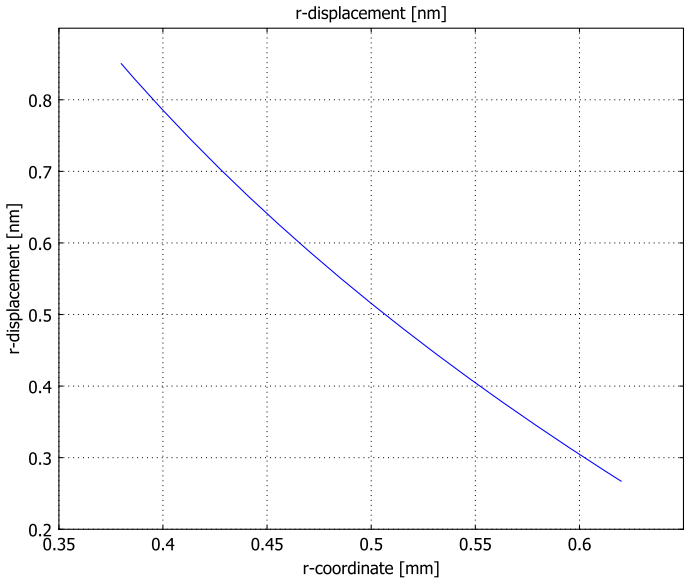


Figure 2: Radial displacement as a function of tube thickness due to the radial electric field for Case 1.

For the Case 2, Figure 3 displays the deformed shape and Figure 4 shows the radial displacement as a function of tube thickness. These results show good agreement with those from S. Peelamedu (Ref. 1).

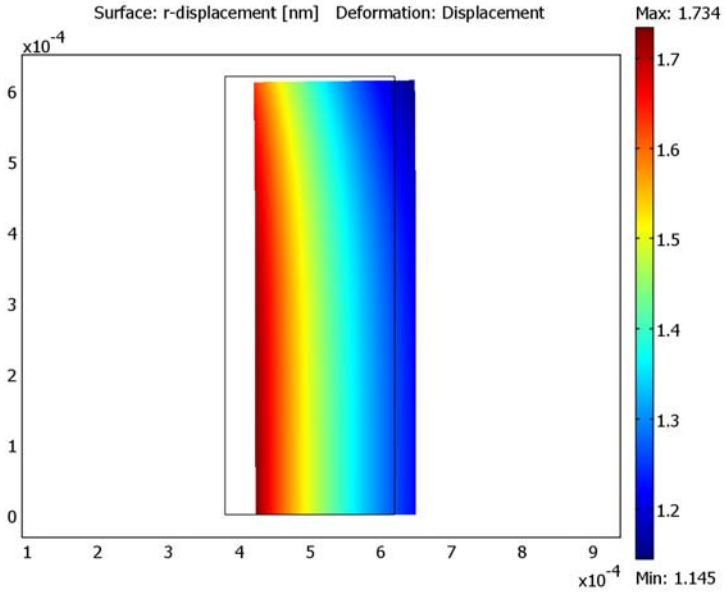


Figure 3: Deformed shape and radial displacement due to an internal pressure of 0.1 MPa (Case 2).

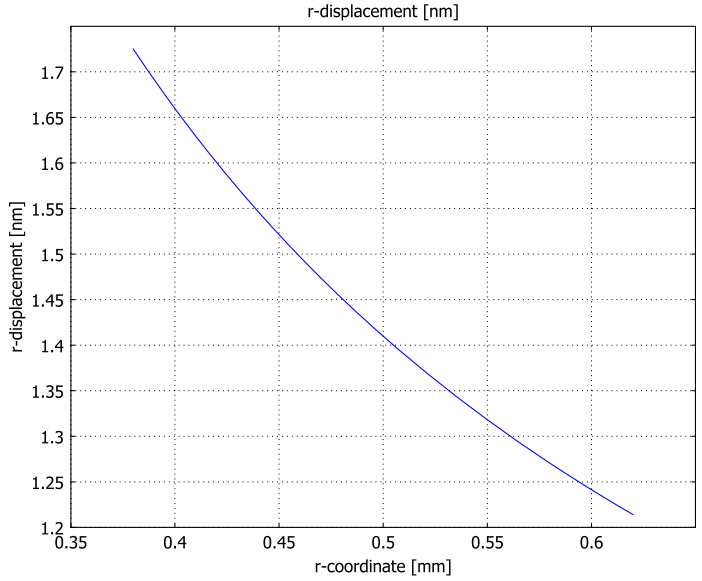


Figure 4: Radial displacement as a function of tube thickness for Case 2.

CONSTITUTIVE EQUATION AND MATERIAL DATA

You specify the material parameters for the piezoelectric material in the **Subdomain Settings** dialog box in the piezoelectric application mode. Then select the stress-charge form for the constitutive equation because this suits the form in which you give the material data. Further, enter the elasticity-matrix elements in the c_e matrix, the piezoelectric coupling-matrix elements in the e matrix, and the relative permittivities in the ε_{rS} matrix.

You enter the material properties in this example such that the polarization is in the z direction (in a 3D Cartesian coordinate system), which is a common orientation for published material data. This orientation means that you must rotate the material so that its polarization direction is aligned with the r direction (radially polarized). To do so, use the material-orientation feature in the Piezo Axial Symmetry application mode. By selecting the material orientation as the zx-plane, you rotate the material so that its z direction is aligned with the r direction of the model, and the material's x direction is aligned with the model's z direction.

The piezoceramic material in this example (PZT-5H) is a transversely isotropic material, which is a special class of orthotropic materials. Such a material has the same properties in one plane (isotropic behavior) and different properties in the direction normal to this plane. Thus you can use either the zx-plane material orientation or the zy-plane material orientation; both give the same solution.

MESHING

The rectangular geometry is well suited for quadrilaterals, and the model uses a 6-by-6-element grid.

Reference

1. S. Peelamedu and others, *Numerical Approach for Axisymmetric Piezoceramic Geometries towards Fluid Control Applications*, Univ. Toledo, OH, Mechanical, Industrial and Manufacturing Engineering Dept., 2000.

Model Library path: `MEMS_Module/Piezo_Models/piezoceramic_tube`

MODEL NAVIGATOR

1 Start COMSOL Multiphysics. This invokes the **Model Navigator**, which you can also open from an already running COMSOL Multiphysics session by choosing **New** from the **File** menu.

2 Select **Axial symmetry (2D)** from the **Space dimension** list.

3 Select **MEMS Module>Structural Mechanics>Piezo Axial Symmetry>Static analysis** from the list of application modes.

4 Click **OK** to close the **Model Navigator**. Note that this gives second-order elements, **Lagrange - Quadratic**, by default.

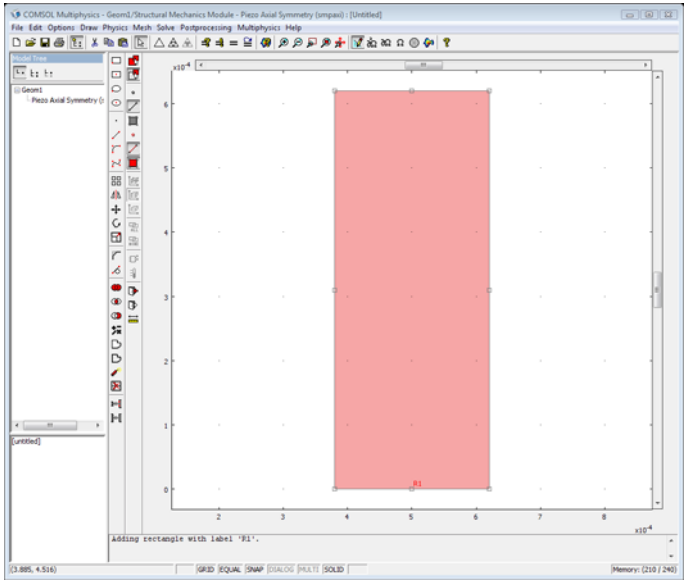
GEOMETRY MODELING

Define the model geometry in Draw mode.

1 Shift-click the **Rectangle/Square** button in the Draw toolbar to open the **Rectangle** dialog box.

2 Type `0.24e-3` in the **Width** edit field and `0.62e-3` in the **Height** edit field. In the **Position** area, type `0.38e-3` in the ***r*** edit field and `0` in the ***z*** edit field. This creates a rectangle with opposite corners at (0.38e-3, 0) and (0.62e-3, 0.62e-3). Click **OK**.

3 Click the **Zoom Extents** button on the Main toolbar.



PHYSICS SETTINGS—MODEL CASE I

Boundary Conditions

1 Open the **Boundary Settings** dialog box by choosing **Boundary Settings** from the **Physics** menu.

2 From the **Boundary selection** list, select Boundary 2.

3 On the **Constraint** page, select **Roller** from the **Constraint condition** list.

4 Select all boundaries by pressing Ctrl+A.

5 On the **Electric BC** page, select **Zero charge/Symmetry** from the **Boundary condition** list.

6 Select Boundary 1, then select **Ground** from the **Boundary condition** list.

7 Select Boundary 4, then select **Electric potential** from the **Boundary condition** list. Set the **Electric potential** to 1.

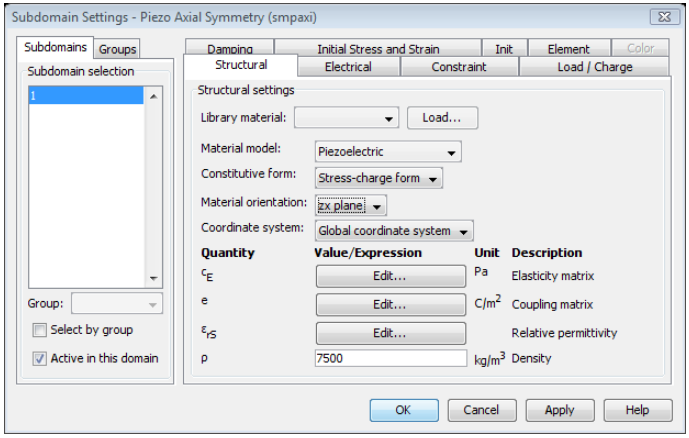
8 Click **OK** to close the dialog box.

Subdomain Settings

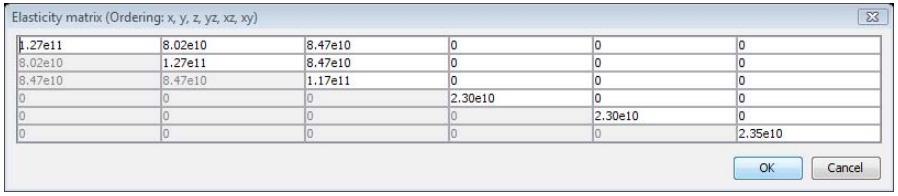
1 Open the **Subdomain Settings** dialog box by selecting **Subdomain Settings** from the **Physics** menu.

2 Select Subdomain 1 and click the **Structural** tab.

3 From the **Material orientation** list, select **zx plane**.



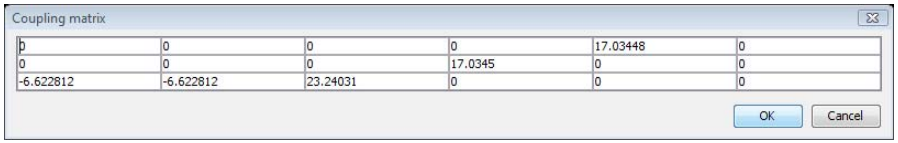
4 Click the **Edit** button associated with $\mathbf{c_E}$ and enter the following values into the **Elasticity matrix** dialog box; when done, click **OK**.



1.27e11	8.02e10	8.47e10	0	0	0
	1.27e11	8.47e10	0	0	0
		1.17e11	0	0	0
			2.30e10	0	0
				2.30e10	0
					2.35e10

Entries for the elasticity matrix.

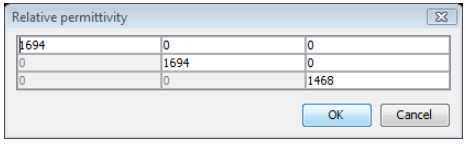
5 Click the **Edit** button associated with **e** and enter the following values into the **Coupling matrix** dialog box; when done, click **OK**.



0	0	0	0	17.03448	0
0	0	0	17.0345	0	0
-6.62812	-6.62812	23.24031	0	0	0

Entries for the coupling matrix.

6 Click the **Edit** button associated with ε_{rS} and enter the following values into the **Relative permittivity matrix** dialog box; when done, click **OK**.



1694		
	1694	
		1468

Entries for the relative permittivity matrix.

7 Click **OK** again to close the **Subdomain Settings** dialog box.

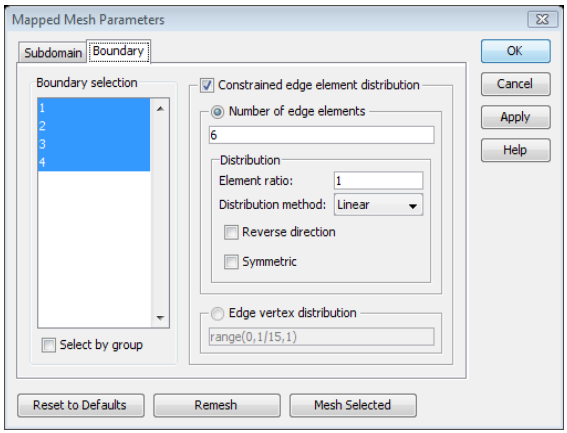
MESH GENERATION

1 From the **Mesh** menu, choose **Mapped Mesh Parameters**.

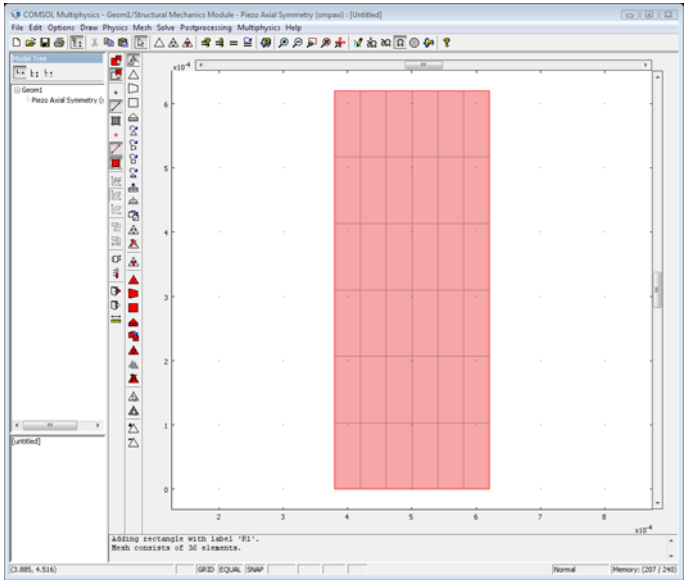
2 Click the **Boundary** tab.

3 Select Boundaries 1–4, then click the **Constrained edge element distribution** button.

4 Set the **Number of edge elements** to 6.



5 Click the **Remesh** button.



The meshed geometry for the piezoceramic-tube actuator.

6 Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1. Choose **Plot Parameters** from the **Postprocessing** menu to open the **Plot Parameters** dialog box.
2. On the **General** page, go to the **Plot type** area and select the **Deformed shape** and **Surface** check boxes.
3. Click the **Surface** tab.
4. Verify that the selection in the **Predefined quantities** list is **Piezo Axial Symmetry (smpaxi)>r-displacement**. From the **Unit** list, select **nm**.
5. Click **OK** to reproduce the plot in Figure 1.
6. From the **Postprocessing** menu, choose **Cross-Section Plot Parameters**.
7. Click the **Line/Extrusion** tab.
8. In the **y-axis data** area, select **Piezo Axial Symmetry (smpaxi)>r-displacement** from the **Predefined quantities** list.
9. In the **x-axis data** area, click first the lower option button and then the **Expression** button. In the **X-Axis Data** dialog box, type `r` in the **Expression** edit field and select **mm** from the **Unit** list. Click **OK** to close the dialog box.
10. In the **Cross-section line data** area, set **r0** to `3.8e-4`, **r1** to `6.2e-4`, and both **z0** and **z1** to `3e-4`.
11. Click **OK** to reproduce the plot in Figure 2.

PHYSICS SETTINGS—MODEL CASE 2

Boundary Conditions

1. From the **Physics** menu, select **Boundary Settings**.
2. Select Boundary 1. On the **Load** page, type `0.1e6` in the $\mathbf{F_r}$ edit field.
3. Select Boundary 4 and then select **Ground** from the **Boundary condition** list on the **Electric BC** page.
4. Click **OK** to close the dialog box.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

If you followed the postprocessing instructions for Case 1 previously, the plot in the main user interface should reproduce the one in Figure 3. If not follow Steps 1–5 in the previous “Postprocessing and Visualization” section.

To reproduce the plot in Figure 4, execute the following instructions.

1 From the **Postprocessing** menu, choose **Cross-Section Plot Parameters**.

2 Click the **Line/Extrusion** tab.

3 Verify that the settings from Steps 8–10 on page 12 are still active, then click **OK**.