Piezoelectricity in a Layered Shell

Introduction

This tutorial is intended as a simple example showing how to model piezoelectric devices using the layered shell functionality.

Two cases of material orientation are investigated. In the first case, the pole axis is normal to the shell surface, which results in a change in thickness of the deformed shell. In the second case, the pole axis is tangential to the shell, which leads to the shell bending.
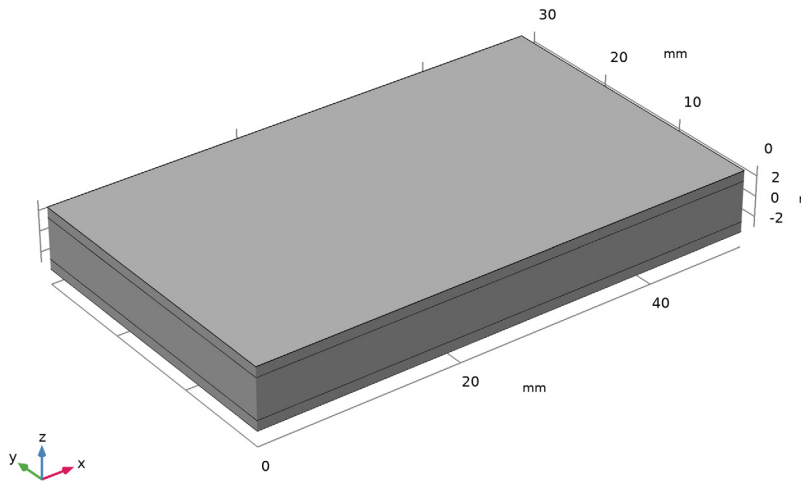
Note: This model requires either MEMS or AC/DC Module.

Model Definition

In this tutorial, you model a thin structure with three layers. The top and bottom layers are electric conductors, and the middle one is a piezoelectric material. The geometry of the structure is shown in Figure 1.


Figure 1: Layered shell geometry representation in 3D.

The modeling approach in this tutorial is based on the layered shell technology available in COMSOL Multiphysics. Thus, you represent the geometry by a surface in 3D space, which gives the base of the layered structure; see Figure 2.
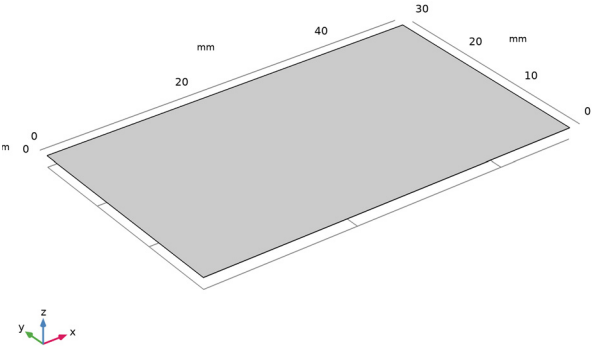
Figure 2: Model geometry representation as a surface.

You also use the surface for meshing, which gives the in-plane part of the discretized representation of the model; see Figure 3.
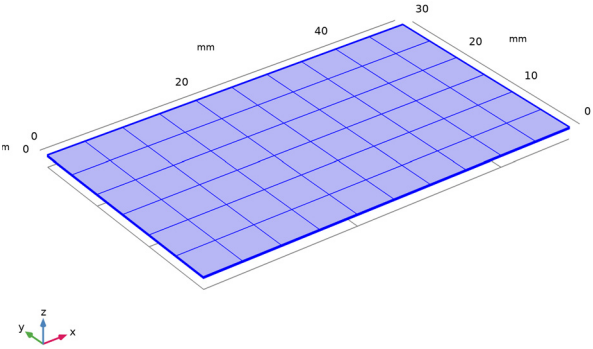
Figure 3: Meshed base selection.

The out-of-plane structure of the shell is modeled using an extra dimension attached to the base selection. The extra dimension together with the base selection span a local 3-dimensional space in which the equations are solved.

The extra dimension is set up using a special *Layered Material* node, which allows you to change the number of layers as well as their thickness and in-plane rotation, and to change the number of mesh elements used in each layer in the extra dimension. You can also select the materials to be used in each layer; see Figure 4.
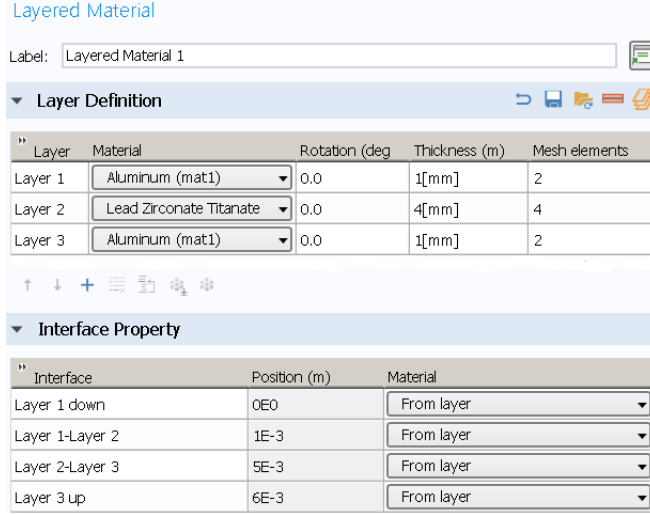

Figure 4: Layered material.

MATERIALS

The structure in this tutorial has three layers. The top and bottom ones represent 1-mm thick conductors made of aluminum. The layer in the middle has a thickness of 4 mm and is composed of a piezoelectric material called Lead Zirconate Titanate (PZT-5H).

For a piezoelectric material, the inelastic contributions to strain due to applied electric field **E** can be written in the strain-charge form as

$$\left[\varepsilon_{xx} \; \varepsilon_{yy} \; \varepsilon_{zz} \; \varepsilon_{yz} \; \varepsilon_{xz} \; \varepsilon_{xy}\right] = d^T \mathbf{E}$$

For materials with tetragonal symmetry (class 4 mm) such as PZT-5H, the coupling matrix has the following structure (Ref. 1):

$$d = \begin{bmatrix} 0 & 0 & 0 & 0 & d_{15} & 0 \\ 0 & 0 & 0 & d_{15} & 0 & 0 \\ d_{31} & d_{31} & d_{33} & 0 & 0 & 0 \end{bmatrix} \tag{1}$$

Here, the pole direction is along the third coordinate axis (the z-axis). Such orientation is assumed for all the piezoelectric material data available in COMSOL Material Library. For PZT-5H, the following values are used: $d_{31} = -2.74 \cdot 10^{-10}$ C/N, $d_{33} = 5.93 \cdot 10^{-10}$ C/N, and $d_{51} = 7.41 \cdot 10^{-10}$ C/N.

When the electric field has only one component, E_z, the only nonzero contributions to the strain are: $\varepsilon_{xx} = \varepsilon_{yy} = d_{31}E_z$ and $\varepsilon_{zz} = d_{33}E_z$.

In this example, the electric field component E_z is positive. Hence, the piezoelectric layer will tend to stretch in the z direction and shrink in the other two directions.

When the pole direction is changed to the x-axis, the first and last lines in the coupling matrix will be swapped. As a result, the only nonzero contribution to the strain will be $\varepsilon_{xz} = d_{15}E_z$ so that the piezo layer will be sheared in the xz-plane.

CONSTRAINTS

At $x = 0$, the whole cross section of the layered shell (the yz-plane) is mechanically fixed; see Figure 1. At the same cross section, the ground condition is applied to the top conducting layer, and a fixed electric potential of 20 V is applied to the bottom one. All other boundaries are free.

Results and Discussion

Two separate stationary studies are used for computing the electric potential and deformation for two different cases of piezoelectric material orientation. In both cases, the electric potential distribution is similar; see Figure 5.

Surface: Electric potential (V)
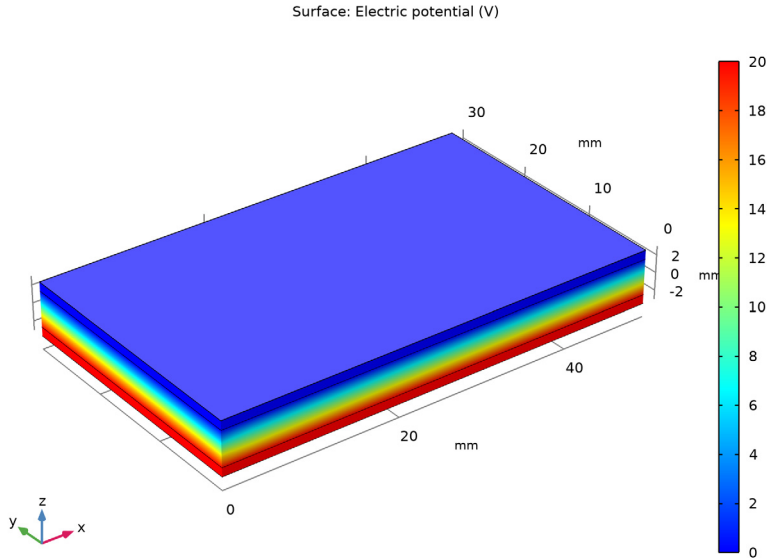

Figure 5: Electric potential distribution.

In the first study, the material has a default orientation with the z-axis as the pole direction, which is normal to the shell. The structure deformation is shown in Figure 6.

In the second study, the material orientation is changed so that the pole direction is in-plane along the x-axis. The nature of resulting structural deformation is significantly different; see Figure 7.

The results show that using a correct choice of the material orientation is essential when modeling piezoelectric applications.

Surface: Displacement field, Z component (mm)
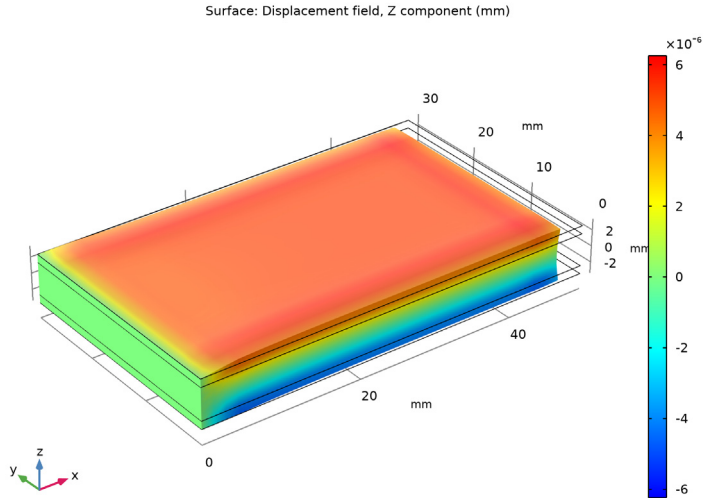
Figure 6: Vertical displacement for the case where the z-axis is the pole direction in the piezoelectric layer.

Surface: Displacement field, Z component (mm)
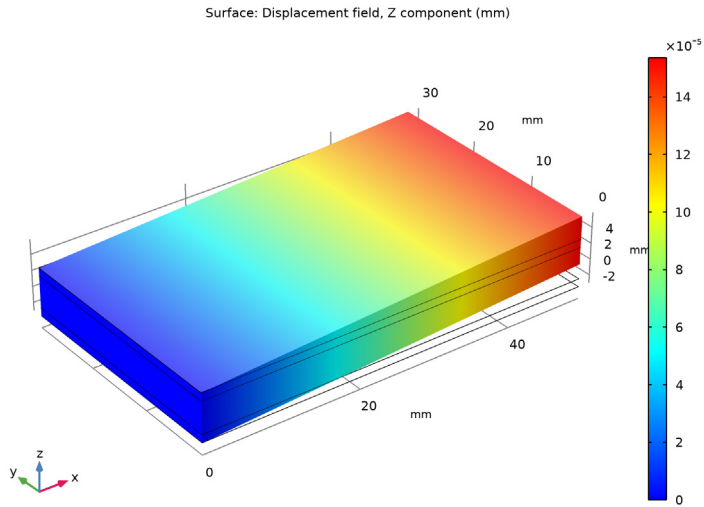
Figure 7: Vertical displacement for the case where the x-axis is the pole direction in the piezoelectric layer.

Notes About the COMSOL Implementation

The device is modeled in 3D using the predefined multiphysics interface **Piezoelectricity, Layered Shell**. Two physics interfaces, structural **Layered Shell** and **Electric Currents in Layered Shells**, will automatically be added to the model together with a multiphysics coupling feature called **Layered Piezoelectric Effect**.

The **Layered Shell** interface will contain a **Piezoelectric Material** node, where you can select geometry boundary for the base surface, and also select certain layers within the layered material. The **Electric Currents in Layered Shells** interface will contain a node **Piezoelectric Layer**, where a similar selection should be made. It is important to have the same selections under both interfaces.

All settings for the material properties and orientation can be found under the **Piezoelectric Material** node within the **Layered Shell** interface. This includes the structural, dielectric, and coupling properties.

Reference

1. A.L. Kholkin, N.A. Pertsev, and A.V. Goltsev, "Piezoelectricity and Crystal Symmetry." In A. Safari and E.K. Akdogan (eds.), *Piezoelectric and Acoustic Materials for Transducer Applications*, Springer, Boston, MA, 2008.

Application Library path: `Composite_Materials_Module/Multiphysics/piezoelectric_layered`

Modeling Instructions

From the **File** menu, choose **New**.

NEW
In the **New** window, click **Model Wizard**.

MODEL WIZARD
1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Electromagnetics-Structure Interaction>Piezoelectricity>Piezoelectricity, Layered Shell**.

3 Click **Add**.

4 Click ➡ **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click ☑ **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click 🔲 **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click ▢ **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 50.

4 In the **Height** text field, type 30.

5 In the **Model Builder** window, right-click **Geometry 1** and choose **Build All**.

6 Click the ↖ **Go to Default View** button in the **Graphics** toolbar.

This geometry represents the base surface of the layered shell.

Next, add the materials aluminum and PZT-5H from the Material Library.

ADD MATERIAL

1 In the **Home** toolbar, click 🔡 **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Aluminum**.

4 Click 🌐 **Add to Global Materials**.

5 In the tree, select **Built-in>Lead Zirconate Titanate (PZT-5H)**.

6 Click 🌐 **Add to Global Materials**.

7 In the **Home** toolbar, click 🔡 **Add Material** to close the **Add Material** window.

Add a layered material node and defined a cross section of the layered shell. Here you can also change the number of mesh elements used in the layered shell extra dimension.
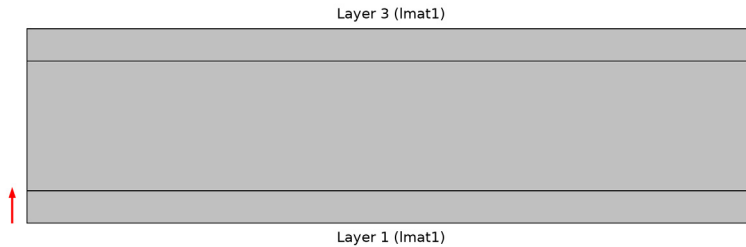
Layered Material 1 (lmat1)

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

3 Click ┼ **Add** twice.

4 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Aluminum (mat1)	0.0	1 [mm]	2
Layer 2	Lead Zirconate Titanate (PZT-5H) (mat2)	0.0	4 [mm]	4
Layer 3	Aluminum (mat1)	0.0	1 [mm]	2

You can preview the shell cross section.

5 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type 8/30.

6 Locate the **Layer Definition** section. Click **Layer Cross Section Preview** in the upper-right corner of the section.

Layer 3 (lmat1)

Layer 1 (lmat1)

Add a layered material link node to make the layered material available in **Component 1**. Using this node, you can also select a coordinate system defining the orientation. Note that only boundary coordinate systems can be selected.

MATERIALS

Layered Material Link 1 (llmat1)
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Layered Material Link**.

Next, select only the middle layer in the shell to use the piezoelectric material.

ELECTRIC CURRENTS IN LAYERED SHELLS (ECIS)

Piezoelectric Layer 1
1 In the **Settings** window for **Piezoelectric Layer**, locate the **Shell Properties** section.
2 Clear the **Use all layers** check box.
3 In the **Selection** table, clear the check boxes for **Layer 1** and **Layer 3**.

LAYERED SHELL (LSHELL)

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Layered Shell (lshell)** click
Piezoelectric Material 1.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Shell Properties** section.

3 Clear the **Use all layers** check box.

4 In the **Selection** table, clear the check boxes for **Layer 1** and **Layer 3**.

5 Locate the **Piezoelectric Material Properties** section. From the **Constitutive relation** list,
choose **Strain-charge form**.

In the first study, the orientation of the piezoelectric layer is in the default state, that is
with the pole axis normal to the base surface.

Next, set up the boundary conditions for the structure and the charge balance. Note that
you need to operate with edges on the base surface.

Fixed Constraint 1

1 In the **Physics** toolbar, click ▢ **Edges** and choose **Fixed Constraint**.

2 Select Edge 1 only.

ELECTRIC CURRENTS IN LAYERED SHELLS (ECIS)

Conductive Shell 1

In the **Model Builder** window, under **Component 1 (comp1)>**
Electric Currents in Layered Shells (ecis) click **Conductive Shell 1**.

Ground 1

In the **Physics** toolbar, click ▢ **Boundaries** and choose **Ground**.

Electric Potential 1

1 In the **Physics** toolbar, click ▢ **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Interface Selection** section.

3 From the **Apply to** list, choose **Bottom interface**.

4 Locate the **Electric Potential** section. In the V_0 text field, type 20.

Use a mapped mesh on the base surface of the layered shell.

Mapped 1

1 In the **Mesh** toolbar, click △ **Boundary** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Click ▦ **Build All**.

5 Click the ↙ **Go to Default View** button in the **Graphics** toolbar.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click ═ **Compute**.

RESULTS

Add a special dataset that will allow a 3D representation of the layered shell extra dimension.

Layered Material 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Datasets** and choose **More Datasets>Layered Material**.

First, create a plot of the electric potential distribution.

Electric Potential

1 In the **Results** toolbar, click ▦ **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type Electric Potential in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

Surface 1

1 In the **Model Builder** window, right-click **Electric Potential** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type V.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

5 In the **Electric Potential** toolbar, click ▣ **Plot**.

6 Click the ↘ **Go to Default View** button in the **Graphics** toolbar.

Add a plot of the shell vertical displacement.

Vertical Displacement (z pole axis)

1 In the **Home** toolbar, click ▣ **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type Vertical Displacement (z pole axis) in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

Surface 1

1 Right-click **Vertical Displacement (z pole axis)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type w.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Vertical Displacement (z pole axis)** toolbar, click ▣ **Plot**.

3 Click the ↘ **Go to Default View** button in the **Graphics** toolbar.

Next, change the orientation of the piezoelectric layer.

LAYERED SHELL (LSHELL)

Piezoelectric Material 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Layered Shell (lshell)** right-click **Piezoelectric Material 1** and choose **Duplicate**.

2 In the **Settings** window for **Piezoelectric Material**, click to expand the **Out-of-Plane Material Orientation** section.

The layered material always operates with a boundary coordinate system on the base surface (laminate system). For such systems, the third base vector direction is always normal to the surface.

Use a special control if you need to change the out-of-plane orientation of the material. This is essential when the pole direction in the material data coincides with the third coordinate axis.

3 From the **Use laminate coordinate system with** list, choose **Swapped normal and 1st tangential directions**.

1 In the **Home** toolbar, click ⚲ **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click ➕ **Add Study**.

5 In the **Home** toolbar, click ⚲ **Add Study** to close the **Add Study** window.

Add one more stationary study and recompute the results after having changed the piezoelectric material orientation.

STUDY 2

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

3 In the **Home** toolbar, click ▬ **Compute**.

RESULTS

Layered Material 2

1 In the **Model Builder** window, under **Results>Datasets** right-click **Layered Material 1** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Finally, plot the new distribution of the vertical displacement in the structure.

Vertical Displacement (x pole axis)

1 In the **Model Builder** window, right-click **Vertical Displacement (z pole axis)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type Vertical Displacement (x pole axis) in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 2**.

4 In the **Vertical Displacement (x pole axis)** toolbar, click ▣ **Plot**.

5 Click the ↙ **Go to Default View** button in the **Graphics** toolbar.