Created in COMSOL Multiphysics 6.1

Electrostrictive Disc

Introduction

The electrostrictive effect describes the electric field-induced strain within a material. It is particularly important for a class of materials known as relaxor ferroelectrics. Such materials are composed of ferroelectric domains that are randomly oriented in the absence of an electric field in the material. In the presence of an electric field, these domains rotate, thereby producing strain in the material. The material extends in the direction of electric field and contracts in the direction perpendicular to the field. On applying an electric field in the reverse direction, the ferroelectric domains orient in the same direction as the field, but the material still elongates and hence electrostriction produces unidirectional strain. To produce a bidirectional strain the material can be subjected to an AC electric field superimposed on a DC bias. At very large electric fields, the electrostrictive strain saturates as all ferroelectric domains in the material align along the direction of the electric field. One of the popular electrostrictive ceramic materials is PMN-PT (lead magnesium niobate-lead titanate), which is often used with some dopant, for example, with barium titanate (BT).

Relaxor ferroelectrics have very high dielectric constants, consequently large polarizations are generated by these materials as a result of an applied electric field. There is very little hysteresis in the response of these materials to electric fields, which makes them attractive for micropositioning devices. A consequence of the absence of hysteresis is that there is negligible self-heating (dielectric heating) in dynamic applications and these materials are therefore used in sonars and ultrasonic motors. Unlike piezoelectric materials, they do not need to be poled. Since there is no residual polarization in the absence of an electric field, a mechanical stress does not change the electric field in the material. Hence these materials are in general not used in sensors.

In this tutorial, you learn how to model isotropic ferroelectroelastic materials using COMSOL's Ferroelectroelasticity interface. This application requires either the MEMS Module or the AC/DC Module and Structural Mechanics Module.

Model Definition

This tutorial shows how to model an electrostrictive cylindrical disc surrounded by air. The geometry is axisymmetric, and consequently 2D axisymmetry is used. The geometry is shown in Figure 1.

The upper end of the disc is at electrical ground whereas the voltage at the lower end is quasi-statically varied from zero to $+10^3$ V volts by using the continuation parameter approach in the study settings.

Because of the symmetry, it is sufficient to model only the upper half of the disc using the symmetry boundary conditions.

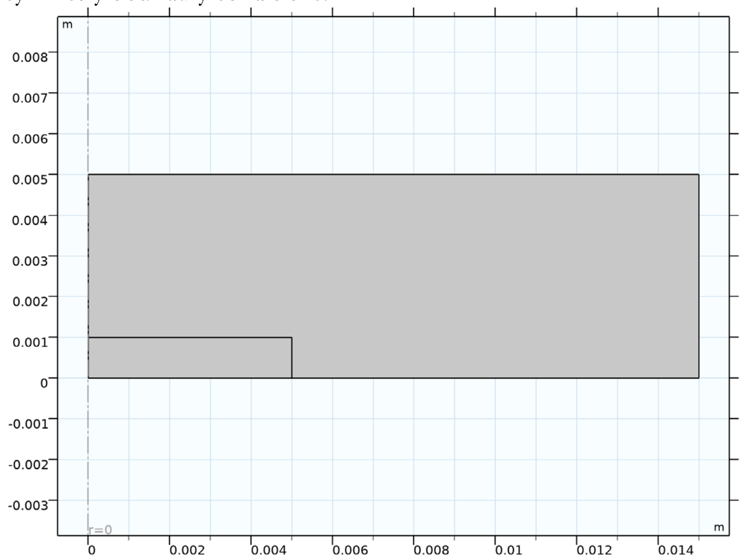


Figure 1: Axisymmetric model geometry. The ferroelectroelastic disc is in the center of the geometry, the remain of the domain shows the surrounding air.

Electrostriction for a material of arbitrary symmetry can be represented as the following additive contribution to the strain:

$$\varepsilon_{\mathrm{em},\,ij} = Q_{ijkl}P^kP^l \tag{1}$$

which is quadratic in polarization $\mathbf{P}$. However, for isotropic materials, the fourth order tensor Q has only two independent components, which are usually denoted as Q_{11} and Q_{12}.

For ferroelectroelastic materials, the polarization vector is nonlinear function of the electric field and possible stress in the material. One possible choice of the polarization shape is hyperbolic tangent (Ref. 1):

$$\mathbf{P} = P_s\tanh(|\mathbf{E}_{\mathrm{eff}}|/a)\frac{\mathbf{E}_{\mathrm{eff}}}{|\mathbf{E}_{\mathrm{eff}}|} \tag{2}$$

where P_s is the saturation polarization, a is a material parameter called the domain wall density, and the effective electric field is given by

$$E_{\text{eff},\,l} = E_l + 2\sigma^{ij}Q_{ijkl}P^k \tag{3}$$

where $\mathbf{E}$ is the applied electric field, and the mechanics stress is computed assuming mechanically linear material as

$$\sigma^{ij} = \mathsf{C}^{ijkl}(\varepsilon_{kl} - \varepsilon_{\text{em},\,kl}) \tag{4}$$

where the strain is computed from the mechanical displacement

$$\varepsilon_{kl} = \frac{1}{2}\left(\frac{\partial u_k}{\partial X^l} + \frac{\partial u_l}{\partial X^k}\right)$$

Again, for isotropic materials, the fourth-order elasticity tensor C has only two independent components. Most common choice to represent those are by specifying the Young's modulus E_{YM} and Poisson's ratio ν for the material.

The effective tangential piezoelectric coupling coefficients can be computed as

$$d^n_{ij} = \frac{\partial\varepsilon_{\text{em},\,ij}}{\partial E_n} = 2\varepsilon_{0,\,\text{vac}}Q_{ijkl}P^k\chi^{ln}$$

where $\varepsilon_{0,\text{vac}}$ is the electric permittivity of free space, and

$$\chi^{ln} = \frac{1}{\varepsilon_{0,\,\text{vac}}}\frac{\partial P^l}{\partial E_n}$$

is the tangent electric susceptibility. An important observation from the above formula is that the piezoelectric coefficients should reach their maximum (or minimum) at certain strength of the applied bias field. This is because $\mathbf{P}$ is zero at zero applied field, while χ tends to zero at large applied field magnitudes because of saturation. The piezoelectric coupling tensor $\mathbf{d}$ is a third-order tensor. Due to the symmetry, it can be conventionally represented by a 3-by-6 matrix d_{ET} with only few nonzero components.

The following material data has been measured (Ref. 1) for a PMT-PT-BT relaxor-ferroelectric material that presents a ternary mixture of lead magnesium niobate with 7.7% lead titanate and 2.5% barium titanate:

TABLE 1: MATERIAL PROPERTIES OF PMT-PT-BT.

MATERIAL PROPERTY	VALUE	DESCRIPTION
E_{YM}	105 GPa	Young's modulus
ν	0.4	Poisson's ratio

TABLE 1: MATERIAL PROPERTIES OF PMT-PT-BT.

MATERIAL PROPERTY	VALUE	DESCRIPTION
ρ	7900 kg/m^3	Density
P_s	0.2589 C/m^2	Saturation polarization
a	0.86207 MV/m	Domain wall density
Q_{11}	0.0133 m^4/C^2	Electrostrictive coupling coefficient
Q_{12}	-0.00606 m^4/C^2	Electrostrictive coupling coefficient

The air domain is modeled with an additional layer on its periphery.

The voltage applied at the disk lower boundary is gradually varying from zero to 4 kV. The upper boundary of the disc is electrically grounded, and it can be loaded mechanically to study the inverse electrostrictive effect. Three cases are analyzed: one without mechanical loading and two others using a vertical boundary load of –40 GPa and –80 GPa, respectively.

Results and Discussion

Figure 2 shows the distribution of the electric potential in the material and in air around it. In Figure 3, the displacement magnitude is visualized in 3D in case of no mechanical loading.

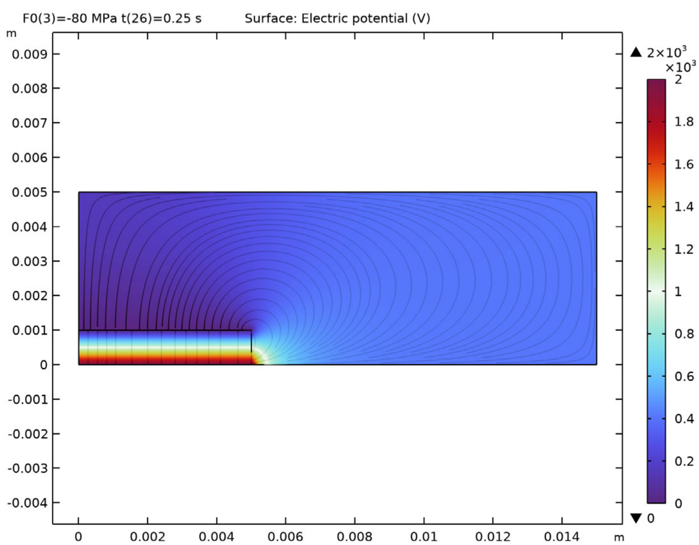


Figure 2: Surface plot of electric potential distribution in the electrostrictive material and surrounding air domain for a 4 kV applied voltage.

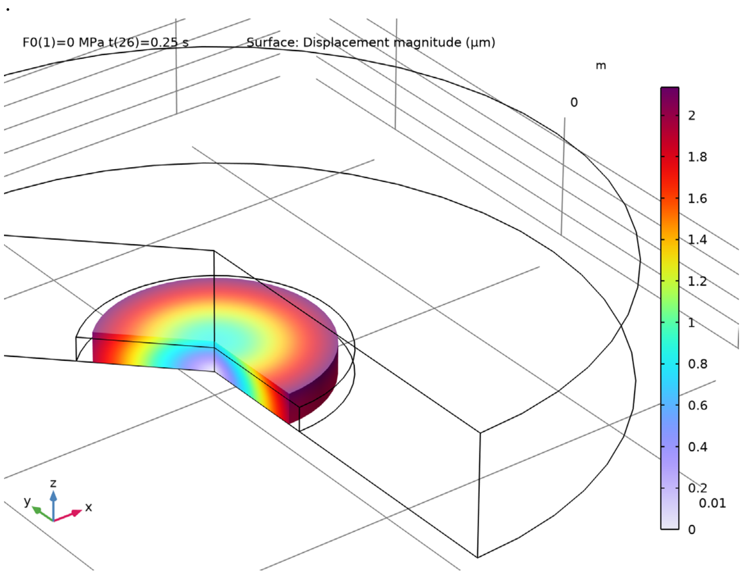


Figure 3: Surface plot of the mechanical displacement magnitude.

Figure 4 shows a plot of the z-component of polarization plotted against the Z-component of the electric field at the center of the disc. Figure 5 shows a similar plot of the ZZ-component of electrostrictive strain at the same position. A series of both negative and positive voltages applied on the top surface of the disc lead to a change in the sign and hence the direction of the electric field. However, the strain and displacement is always unidirectional.

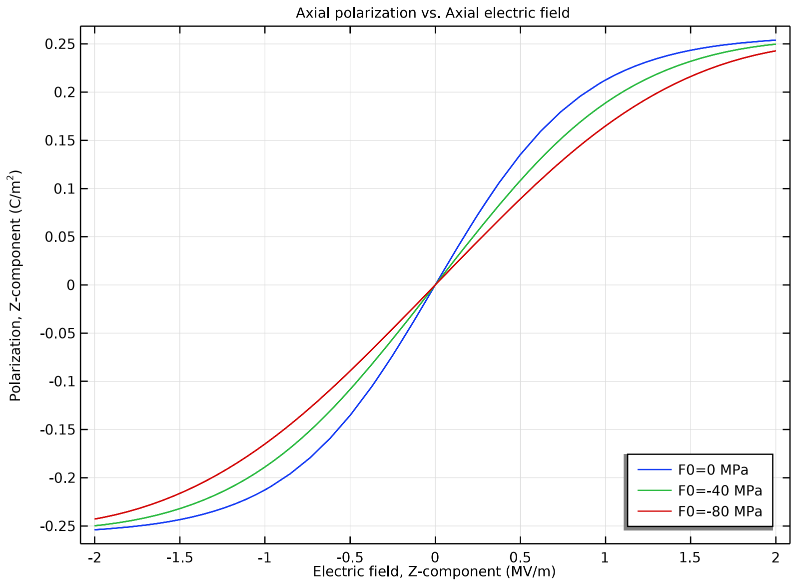


Figure 4: Axial polarization vs. axial electric field at the center of the disc.

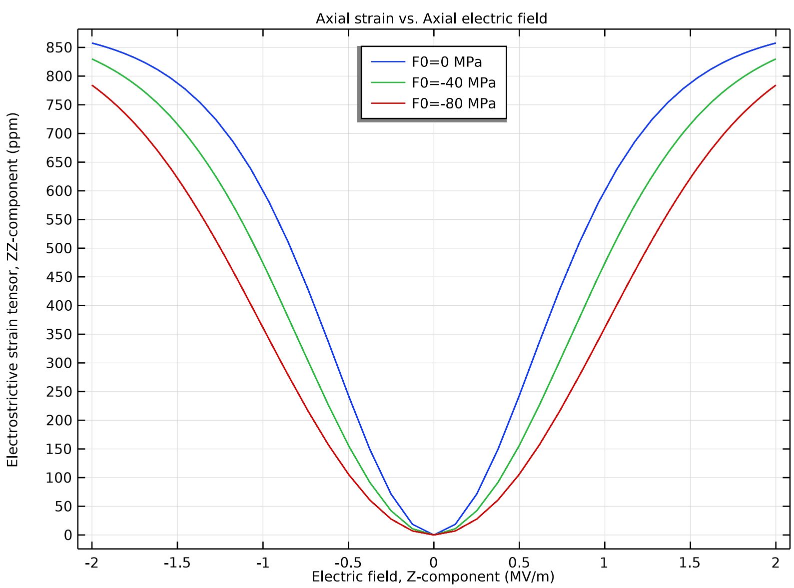


Figure 5: Axial electrostrictive strain vs. axial electric field at the center of the disc.

Finally, Figure 6 shows the effective tangent piezoelectric coefficients computed at a given value of the applied electric field.

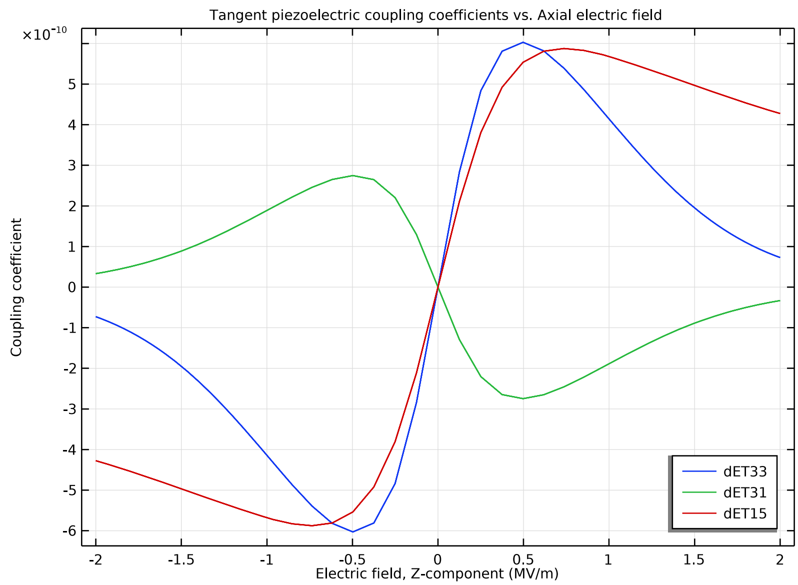


Figure 6: Tangent piezoelectric coupling coefficients vs. axial electric field at the center of the disc.

Reference

1. C.L. Horn and N. Shankar, "A finite element method for electrostrictive ceramic devices," *Int. J. Solids Structures*, vol. 33, pp. 1757–1779, 1995.

Application Library path: `MEMS_Module/Actuators/electrostrictive_disc`

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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics>Electromagnetics-Structure Interaction>Ferroelectroelasticity**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Create parameters to define the geometry, material properties, and applied loads.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
t	0[s]	0 s	Time parameter
H0	2[mm]	0.002 m	Disc thickness
R0	5[mm]	0.005 m	Disc radius
Q11	0.0133[m^4/C^2]	0.0133 m^4/C^2	Electrostrictive coupling coefficient
Q12	-0.00606[m^4/C^2]	-0.00606 m^4/C^2	Electrostrictive coupling coefficient
E1	105[GPa]	1.05E11 Pa	Young's modulus
nu1	0.4	0.4	Poisson's ratio
rho1	7900[kg/m^3]	7900 kg/m^3	Density
Ps	0.2589[C/m^2]	0.2589 C/m^2	Saturation polarization
a	0.86207 [MV/m]	8.6207E5 V/m	Domain wall density
Vmax	2[MV/m]*H0	4000 V	Maximum applied voltage
F0	-80[MPa]	-8E7 Pa	F0

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
V0	Vmax*sin(2*pi*t[1/s])	V	Applied voltage

This variation of the potential with respect to the parameter at one of the disc boundaries will cause the electric field within the material to gradually change between -Vmax and Vmax.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `R0`.

4 In the **Height** text field, type `H0/2`.

Rectangle 2 (r2)

1 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

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

3 In the **Width** text field, type `3*R0`.

4 In the **Height** text field, type `5*H0/2`.

5 Click **Build All Objects**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 1 only.

Symmetry Plane 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.

2 Select Boundary 2 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	r
F0	z

ELECTROSTATICS (ES)

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

Ground 1

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

2 Select Boundary 4 only.

Electric Potential 1

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

2 Select Boundary 2 only.

Because of the symmetry, the voltage at the horizontal symmetry plane equals to a half of that applied at the bottom surface.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `V0/2`.

Charge Conservation, Ferroelectric 1

1 In the **Model Builder** window, click **Charge Conservation, Ferroelectric 1**.

2 Select Domain 1 only.

3 In the **Settings** window for **Charge Conservation, Ferroelectric**, locate the **Ferroelectric Material Properties** section.

4 Find the **Anhysteretic polarization** subsection. From the **Anhysteretic polarization shape** list, choose **Hyperbolic tangent**.

5 Find the **Effective electric field** subsection. From the α list, choose **User defined**. In the associated text field, type `0`.

MULTIPHYSICS

Electrostriction 1 (efe1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Electrostriction 1 (efe1)**.

2 In the **Settings** window for **Electrostriction**, locate the **Coupling Type** section.

3 From the list, choose **Fully coupled**.

4 Locate the **Electrostriction** section. In the Q_{11} text field, type `Q11`.

5 In the Q_{12} text field, type `Q12`.

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>Air**.

4 Right-click and choose **Add to Component 1 (comp1)**.

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

MATERIALS

PMT-PT-BT

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `PMT-PT-BT` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	E1	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu1	1	Young's modulus and Poisson's ratio
Density	rho	rho1	kg/m³	Basic
Saturation polarization	Psat	Ps	C/m²	Ferroelectric
Domain wall density	aJAe_iso ; aJAeii = aJAe_iso, aJAeij = 0	a	V/m	Ferroelectric

MESH 1

Mapped 1

1. In the **Mesh** toolbar, click **Mapped**.
2. In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 1 only.

Distribution 1

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 1 and 6 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type 4.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 2 and 4 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type 16.

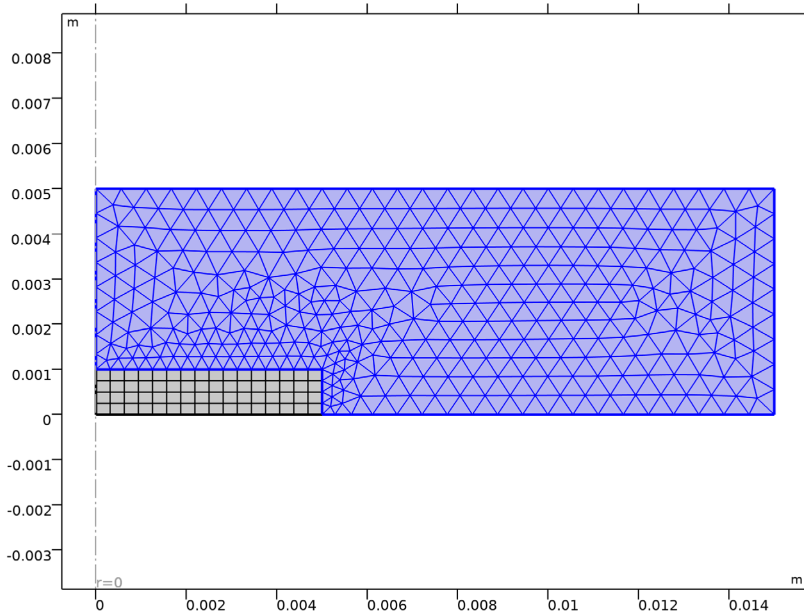
Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Finer**.

Free Triangular 1

1. In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, click **Build All**.



STUDY 1

Change the study settings to sweep over applied voltage.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t (Time parameter)	range(0,0.01,1)	s

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
F0 (F0)	0 -40 -80	MPa

5 In the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

The first and second default plots show the stress distribution in the disc, which is almost uniform.

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **MPa**.

4 In the **Stress (solid)** toolbar, click **Plot**.

Displacement, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (F0 (MPa))** list, choose **0**.

4 From the **Parameter value (t (s))** list, choose **0.25**.

5 In the **Label** text field, type `Displacement, 3D (solid)`.

Surface 1

1 In the **Model Builder** window, expand the **Displacement, 3D (solid)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `solid.disp`.

4 From the **Unit** list, choose **µm**.

5 In the **Displacement, 3D (solid)** toolbar, click **Plot**.

Electric Potential (es)

The fourth default plots shows the electric potential distribution.

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (t (s))** list, choose **0.25**.

4 In the **Electric Potential (es)** toolbar, click **Plot**.

Plot the electrostrictive strain and polarization in the middle of the disc versus the applied field.

Electrostrictive Strain

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

2 In the **Settings** window for **1D Plot Group**, type `Electrostrictive Strain` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Axial strain vs. Axial electric field`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Point Graph 1

1 Right-click **Electrostrictive Strain** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `efe1.emZZ`.

5 From the **Unit** list, choose **ppm**.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `es.EZ`.

8 From the **Unit** list, choose **MV/m**.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 Find the **Include** subsection. Clear the **Point** check box.

11 In the **Electrostrictive Strain** toolbar, click **Plot**.

Polarization

1 In the **Model Builder** window, right-click **Electrostrictive Strain** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Polarization` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Axial polarization vs. Axial electric field`.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

1 In the **Model Builder** window, expand the **Polarization** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `es.PZ`.

4 In the **Polarization** toolbar, click **Plot**.

Finally, plot the tangent piezoelectric coupling coefficients.

Tangent Piezoelectric Coupling Coefficients

1 In the **Model Builder** window, right-click **Polarization** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (F0)** list, choose **First**.

4 In the **Label** text field, type `Tangent Piezoelectric Coupling Coefficients`.

5 Locate the **Title** section. In the **Title** text area, type `Tangent piezoelectric coupling coefficients vs. Axial electric field`.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** check box. In the associated text field, type `Coupling coefficient`.

8 Locate the **Grid** section. Select the **Manual spacing** check box.

9 In the **x spacing** text field, type `0.5`.

10 In the **y spacing** text field, type `1e-10`.

Point Graph 1

1 In the **Model Builder** window, expand the **Tangent Piezoelectric Coupling Coefficients** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `efe1.dET33`.

4 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`dET33`

Point Graph 2

1 Right-click **Results>Tangent Piezoelectric Coupling Coefficients>Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `efe1.dET31`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
dET31

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `efe1.dET15`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
dET15

5 In the **Tangent Piezoelectric Coupling Coefficients** toolbar, click **Plot**.