Thickness Shear Mode Quartz Oscillator

Introduction

AT cut quartz crystals are widely employed in a range of applications, from oscillators to microbalances. One of the important properties of the AT cut is that the resonant frequency of the crystal is temperature independent to first order. This is desirable in both mass sensing and timing applications. AT cut crystals vibrate in the thickness shear mode—an applied voltage across the faces of the cut produces shear stresses inside the crystal. This example considers the vibration of an AT cut thickness shear oscillator, focusing on the mechanical response of the system in the frequency domain. The effect of a series capacitor on the mechanical resonance is also considered. Adding a series capacitance is a technique frequently employed to tune crystal oscillators.

Model Definition

The model geometry is shown in Figure 1. The oscillator consists of a single quartz disc, supported so as not to impede the motion of the vibrational mode. There are two electrodes on the top and bottom surfaces of the geometry, one of which is grounded.

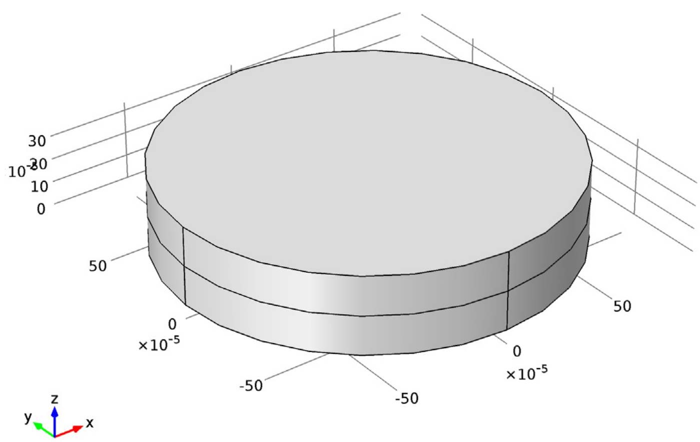

Figure 1: Model geometry.

In the first version of the model an AC voltage is applied to the top electrode. In the second version, the crystal is still driven by an AC voltage, but a capacitor is placed between the voltage source and the top electrode of the crystal.

DOMAIN LEVEL EQUATIONS

Within a piezoelectric there is a coupling between the strain and the electric field, which is determined by the constitutive relation:

$$\begin{aligned} \mathbf{T} &= c_E\mathbf{S} - e^T\mathbf{E} \\ \mathbf{D} &= e\mathbf{S} + \varepsilon_S\mathbf{E} \end{aligned} \tag{1}$$

Here, $\mathbf{S}$ is the strain, $\mathbf{T}$ is the stress, $\mathbf{E}$ is the electric field, and $\mathbf{D}$ is the electric displacement field. The material parameters c_E, e, and ε_S, correspond to the material stiffness, the coupling properties and the permittivity. These quantities are tensors of rank 4, 3, and 2 respectively, but, since the tensors are highly symmetric for physical reasons, they can be represented as matrices within an abbreviated subscript notation, which is usually more convenient. In the Piezoelectric Devices interface, the Voigt notation is used, which is standard in the literature for piezoelectricity but which differs from the defaults in the Solid Mechanics interface. In addition to Equation 1 the equations of solid mechanics and electrostatics must also be solved within the material.

MATERIAL ORIENTATION

The orientation of a piezoelectric crystal cut is frequently defined by the system introduced by the I.R.E. standard of 1949 (Ref. 1). This standard has undergone a number of subsequent revisions, with the final revision being the IEEE standard of 1989 (Ref. 2). Unfortunately the more recent versions of the standard have not been universally adopted, and significant differences exist between the 1949 and the 1987 standards. The 1987 standard was ultimately withdrawn by the IEEE. COMSOL follows the conventions used in the book by Auld (Ref. 3) and defined by the 1987 standard. While these conventions are often used for many piezoelectric materials, unfortunately practitioners in the quartz industry frequently adhere to the older 1947 standard, which results in different definitions of crystal cuts and of material properties.

The crystal axes used to define material properties in COMSOL correspond to the 1987 IEEE standard. The stiffness, compliance, coupling, and dielectric material property matrices are defined with the crystal axes aligned with the local coordinate axes. In the absence of a user defined coordinate system, the local system corresponds to the global X, Y, and Z coordinate axes. To define a particular crystal cut, a local set of rotated coordinates must be defined; this local system then corresponds to the orientation of the crystal axes within the model. Note that the orientation of the local co-ordinate system should change both when the crystal cut is changed and if the plate changes orientation with respect to COMSOL's global co-ordinate system.

The crystal cuts are defined differently within the 1949 and 1987 standards. Both standards use a notation that defines the orientation of a virtual slice (the plate) through the crystal. The crystal axes are denoted X, Y, and Z and the plate, which is usually rectangular, is defined as having sides l, w, and t (length, width, and thickness). Initially the plate is aligned with respect to the crystal axes and then up to three rotations are defined, using a right-handed convention about axes embedded along the l, w, and t sides of the plate. The 1987 standard defines the AT cut as: $(YXl)\ -35.25°$. The first two letters in the bracketed expression always refer to the initial orientation of the thickness and the length of the plate. Subsequent bracketed letters then define up to three rotational axes, which move with the plate as it is rotated. Angles of rotation about these axes are specified after the bracketed expression in the order of the letters, using a right-handed convention. For AT cut quartz only one rotation, about the l axis, is required. This is illustrated in Figure 2.

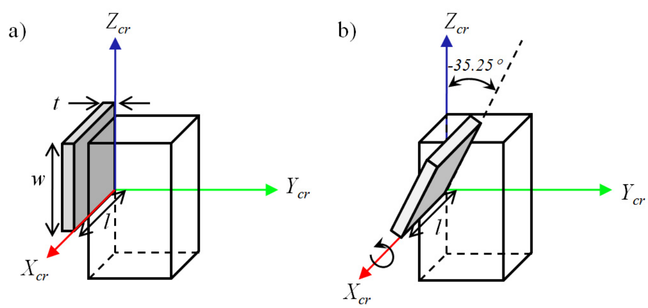

Figure 2: Definition of the AT cut of quartz within the IEEE 1987 standard. The AT cut is defined as: (YXl) −35.25°. The first two bracketed letters specify the initial orientation of the plate, with the thickness direction, t, along the crystal Y axis and the length direction, l, along the X axis. Then up to three rotations about axes that move with the plate are specified by the corresponding bracketed letters and the subsequent angles. In this case only one rotation is required about the l axis, of −35.25° (in a right-handed sense).

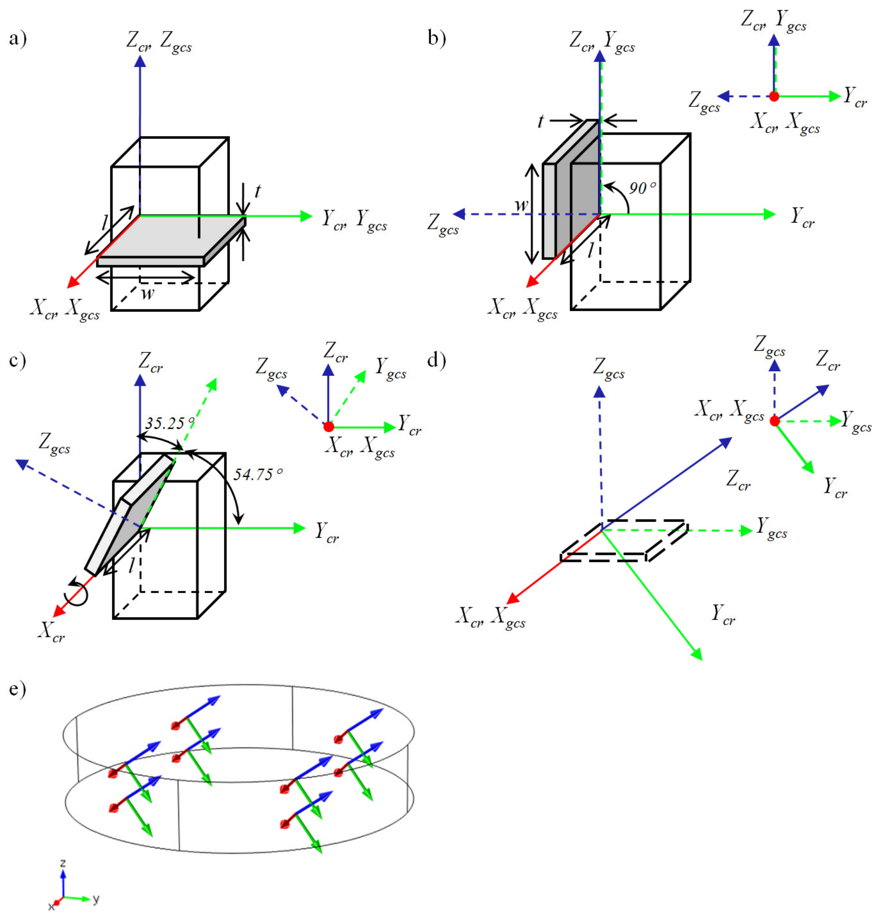

Figure 3: Defining an AT cut crystal plate within COMSOL, with normal in the global Z-direction. Within the 1987 IEEE standard the AT cut is defined as (YXl) −35.25°. Begin with the plate normal in the Z_{cr}-direction, so the crystal and global systems are coincident. Rotate the plate so that its thickness points in the Y_{cr}-direction (the starting point for the IEEE definition), the global system rotates with the plate (b). Rotate the plate −35.25° about the l axis (d). Finally rotate the entire system so that the global coordinate system is orientated as it appears in COMSOL (d). The local coordinate system should be defined with the Euler angles (ZXZ: 0, -54.75°, 0). (e) shows a coordinate system for this system in COMSOL.

Note that within the 1949 convention AT cut quartz is denoted as: (YXl) +35.25°, since the X-axis rotated by 180° in this convention and positive angles therefore correspond to the opposite direction of rotation.

Figure 3 describes in detail how to define the AT cut in a quartz crystal, with the plate orientated with its normal pointing in the Z-axis of the global co-ordinate system. This is the orientation of the device in this model. In general for defining any crystal cut, care must be taken with defining the co-ordinate system and detailed diagrams, similar to those in Figure 3, should be drawn.

ELECTRICAL CIRCUIT

In the first part of the model an AC voltage is applied directly to the top plate of the oscillator, which is grounded. In the second part of the model, a capacitor is added between the voltage source and the oscillator, as shown in Figure 4. In COMSOL the oscillator is coupled into the circuit using the **External I Terminal** feature. The terminal boundary condition within the model is set to **Circuit** and this feature then captures the charge generated by the circuit.

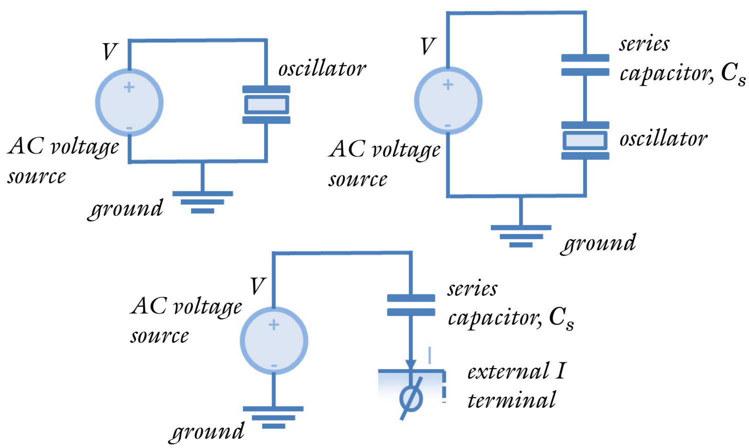

Figure 4: Top left: Electrical circuit for the first part of the model. Top right: Electrical circuit for the second part of the model. Bottom: Circuit for the second part of the model as implemented in COMSOL.

Results and Discussion

Figure 5 shows the crystal displacement at its resonant frequency of 5.11 MHz. The form of the displacement shows clearly the shear nature of the resonance. The potential on cut slices through the plate is illustrated in Figure 6. The mechanical domain frequency response of the oscillator is shown in Figure 7. A clear anti-resonance is apparent, with a resonant frequency close to 5.11 MHz.

The addition of a series capacitance between the oscillator and the voltage source is expected to pull the resonant frequency to higher values. Figure 8 shows that this effect occurs as expected, with the resonant frequency increasing the most for smaller values of the series capacitance (in the limit of very large series capacitance the impedance of the series capacitor goes to zero and we obtain the result shown in Figure 7).

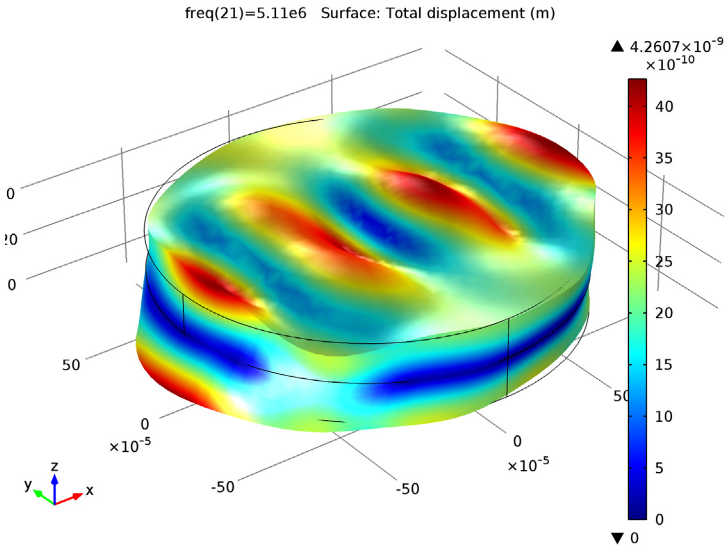

Figure 5: Displacement of the crystal at resonance.

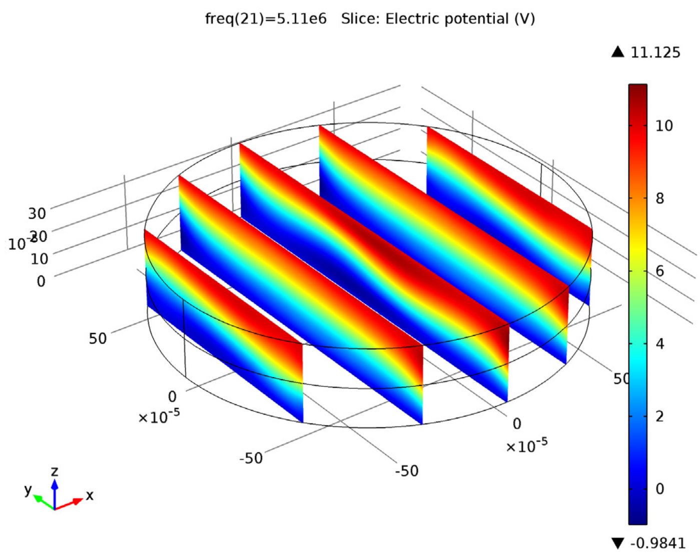

Figure 6: Electric potential inside the crystal at resonance.

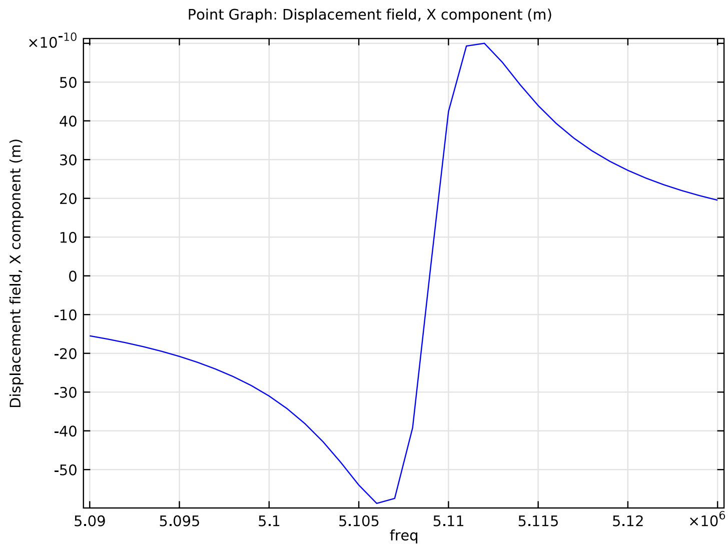

Figure 7: Mechanical response of the structure with no series capacitance.

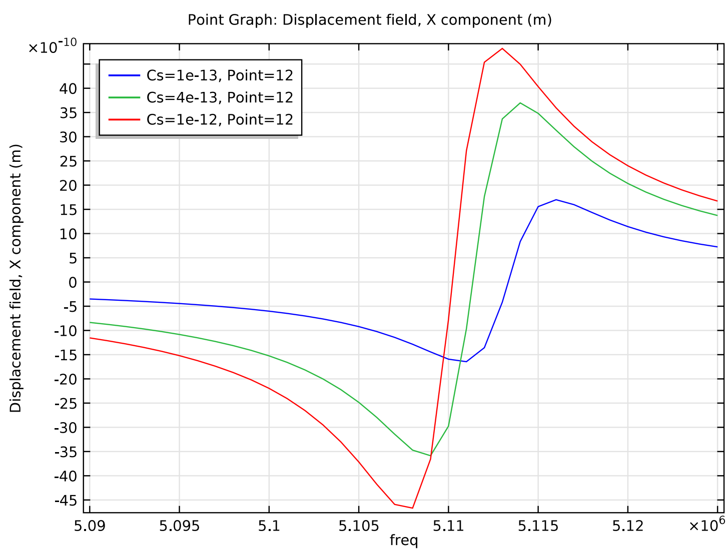

Figure 8: Mechanical response of the structure with different series capacitances.

References

1. Standards on Piezoelectric Crystals, 1949", *Proceedings of the I. R. E.*,vol. 37, no.12, pp. 1378 - 1395, 1949.

2. IEEE Standard on Piezoelectricity, ANSI/IEEE Standard 176-1987, 1987.

3. B. A. Auld, *Acoustic Fields and Waves in Solids*, Krieger Publishing Company, 1990.

Model Library path: `MEMS_Module/Piezoelectric_Devices/thickness_shear_quartz_oscillator`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Piezoelectric Devices (pzd)**.

4 Click **Add Selected**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Add parameters for the model geometry and series capacitance.

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
`Cs`	`1[pF]`	`Series Capacitance`
`R0`	`0.835[mm]`	`Oscillator Radius`
`H0`	`334[um]`	`Oscillator Thickness`

Create the geometry.

GEOMETRY 1

Cylinder 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `R0`.

4 In the **Height** edit field, type `H0`.

Cylinder 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `R0`.

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

5 Click the **Build All** button.

Set up a rotated system appropriate for AT cut Quartz.

DEFINITIONS

Rotated System 2

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Coordinate Systems>Rotated System**.

2 In the **Rotated System** settings window, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the β edit field, type `-54.75[deg]`.

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Piezoelectric>Quartz**.

3 Click **Add Material to Model**.

Quartz

Use the rotated system to define the orientation of the crystal.

PIEZOELECTRIC DEVICES

Piezoelectric Material 1

1 In the **Model Builder** window, expand the **Model 1>Piezoelectric Devices** node, then click **Piezoelectric Material 1**.

2 In the **Piezoelectric Material** settings window, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 2**.

Add damping to the model.

Damping and Loss 1

1 Right-click **Model 1>Piezoelectric Devices>Piezoelectric Material 1** and choose **Damping and Loss**.

2 In the **Damping and Loss** settings window, locate the **Damping Settings** section.

3 From the **Damping type** list, choose **Isotropic loss factor**.

4 From the η_s list, choose **User defined**. In the associated edit field, type `1e-3`.

Set up point constraints to prevent rotational and translational modes.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Piezoelectric Devices** and choose **Points>Fixed Constraint**.

2 Select Point 5 only.

Prescribed Displacement 1

1 Right-click **Piezoelectric Devices** and choose **Points>Prescribed Displacement**.

2 Select Point 8 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1. Right-click **Piezoelectric Devices** and choose **Points>Prescribed Displacement**.
2. Select Point 2 only.
3. In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.
4. Select the **Prescribed in z direction** check box.

   Add electrical boundary conditions to the model.

Terminal 1

1. Right-click **Piezoelectric Devices** and choose the boundary condition **Electrical>Terminal**.
2. Select Boundary 7 only.
3. In the **Terminal** settings window, locate the **Terminal** section.
4. From the **Terminal type** list, choose **Voltage**.
5. In the V_0 edit field, type 10.

Ground 1

1. Right-click **Piezoelectric Devices** and choose the boundary condition **Electrical>Ground**.
2. Select Boundary 3 only.

   Create a swept triangular mesh.

MESH 1

Free Triangular 1

1. In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.
2. Select Boundary 7 only.

Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1. In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.
2. In the **Distribution** settings window, locate the **Distribution** section.
3. In the **Number of elements** edit field, type 2.

4 Click the **Build All** button.

Set up and solve a frequency dependent study.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency Domain**.
2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
3 Click the **Range** button.
4 Go to the **Range** dialog box.
5 In the **Start** edit field, type `5.09e6`.
6 In the **Step** edit field, type `1e3`.
7 In the **Stop** edit field, type `5.125e6`.
8 Click the **Replace** button.
9 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

Visualize the mode shape of the device at resonance.

RESULTS

Displacement (pzd)

1 In the **3D Plot Group** settings window, locate the **Data** section.
2 From the **Parameter value (freq)** list, choose **5.11e6**.
3 Click the **Plot** button.

The second default plot shows the electric potential within the device.

Potential (pzd)

1 In the **Model Builder** window, under **Results** click **Potential (pzd)**.
2 In the **3D Plot Group** settings window, locate the **Data** section.
3 From the **Parameter value (freq)** list, choose **5.11e6**.
4 Click the **Plot** button.

Add a plot to show the mechanical response of the device.

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2 Right-click **1D Plot Group 3** and choose **Point Graph**.

3 Select Point 12 only.

4 In the **Point Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Piezoelectric Devices>Displacement field (Material)>Displacement field, X component (u)**.

5 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Rename**.

6 Go to the **Rename 1D Plot Group** dialog box and type `Mechanical Response` in the **New name** edit field.

7 Click **OK**.

8 Right-click **1D Plot Group 3** and choose **Plot**.

Now add a capacitor in series with the device.

MODEL 1

In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 In the **Add physics** tree, select **AC/DC>Electrical Circuit (cir)**.

3 Click **Add Selected**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Frequency Domain**.

6 Click **Finish**.

ELECTRICAL CIRCUIT

Features in the electric circuits interface are connected by specifying connecting node numbers for each port of the device.

Ground Node 1

A ground node is automatically added to the circuit, with the default node number of 0.

Next add a voltage source between the ground node and a (newly created) node with number 2.

Voltage Source 1

1 In the **Model Builder** window, right-click **Electrical Circuit** and choose **Voltage Source**.

2 In the **Voltage Source** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2

4 Locate the **Device Parameters** section. From the **Source type** list, choose **AC-source**.

5 In the V_{src} edit field, type `10`.

Add a capacitor between the voltage source output (node 2) and a new node, 1.

Capacitor 1

1 Right-click **Electrical Circuit** and choose **Capacitor**.

2 In the **Capacitor** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	1

4 Locate the **Device Parameters** section. In the C edit field, type `Cs`.

Connect node 1 to the terminal feature in the model using the external I-terminal feature.

External I-Terminal 1

1 Right-click **Electrical Circuit** and choose **External I-Terminal**.

The terminal node in the piezoelectric devices interface needs to be changed to a circuit type terminal.

PIEZOELECTRIC DEVICES

Terminal 1

1 In the **Model Builder** window, under **Model 1>Piezoelectric Devices** click **Terminal 1**.

2 In the **Terminal** settings window, locate the **Terminal** section.

3 From the **Terminal type** list, choose **Circuit**.

Next, couple the electric potential from the terminal node back into the model.

ELECTRICAL CIRCUIT

External I-Terminal 1

1 In the **Model Builder** window, under **Model 1>Electrical Circuit** click **External I-Terminal 1**.
2 In the **External I-Terminal** settings window, locate the **External Terminal** section.
3 From the *V* list, choose **Terminal voltage (pzd/term1)**.

Finally set up a study to compute the frequency response of the device with different capacitors added in series.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** click **Step 1: Frequency Domain**.
2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
3 Click the **Range** button.
4 Go to the **Range** dialog box.
5 In the **Start** edit field, type `5.09e6`.
6 In the **Step** edit field, type `1e3`.
7 In the **Stop** edit field, type `5.125e6`.
8 Click the **Replace** button.

Parametric Sweep

1 In the **Model Builder** window, right-click **Study 2** and choose **Parametric Sweep**.
2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter value list
`0.1e-12 0.4e-12 1e-12`

5 In the **Model Builder** window, click **Study 2**.
6 In the **Study** settings window, locate the **Study Settings** section.
7 Clear the **Generate default plots** check box.
8 Click the **Compute** button.

RESULTS

Re-plot the mechanical response with the additional series capacitance.

Mechanical Response 1

1. In the **Model Builder** window, under **Results** right-click **Mechanical Response** and choose **Duplicate**.
2. In the **1D Plot Group** settings window, locate the **Data** section.
3. From the **Data set** list, choose **Solution 3**.
4. Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.
5. In the **Model Builder** window, expand the **Mechanical Response 1** node, then click **Point Graph 1**.
6. In the **Point Graph** settings window, click to expand the **Legends** section.
7. Select the **Show legends** check box.
8. Click the **Plot** button.

   Note how the mechanical resonant frequency is 'pulled' by the series capacitance.