Created in COMSOL Multiphysics 5.4

Spherical Piezoacoustic Transducer

This model is licensed under the COMSOL Software License Agreement 5.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The piezoelectric effect describes an induced strain caused by an electric field in certain ferroelectric materials that are poled along a specific direction. Piezoelectric materials are composed of ferroelectric domains which are initially randomly oriented. The poling process creates a preferred orientation of these domains. In the presence of an electric field, these domains rotate from the poled direction thereby producing strain in the material.

Piezoelectric materials are widely used as actuator cores in acoustic transducers. In such devices, the piezoelectric material is excited with an electrical input, typically at high frequencies (kHz to MHz range). The harmonic electrical excitation produces structural vibrations in the material which in turn set up acoustic waves in the surrounding fluid media. This principle is used in several applications such as hydrophones, ultrasound imaging and non-destructive testing.

In this tutorial, you learn how to model the acoustic waves generated in air by a hollow spherical piezoelectric material which is poled along the radial direction of the sphere. Since the direction of poling imparts anisotropy to the material response, it is critical to incorporate it correctly in the simulation.

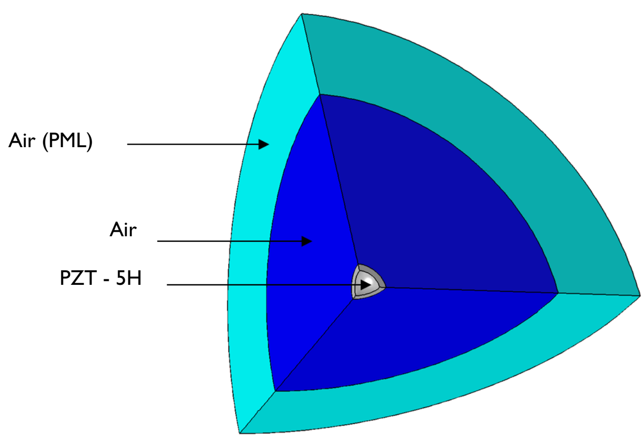


Figure 1: Pictorial representation of a 1/8 th symmetric section of a hollow, spherical, piezoelectric domain made of PZT-5H surrounded by a region of air that is infinitely extended. An additional geometric layer is used to set up a perfectly matched layer (PML). PMLs are used to efficiently absorb outgoing waves.

Model Definition

In this model, the geometry and mesh are parametrized with respect to the excitation frequency and speed of sound in the fluid medium. The model assumes the fluid to be air and the speed of sound to be 343 m/s. The inner radius and thickness of the hollow sphere are also parametrized; the model considers their values to be 2.5 mm and 1 mm, respectively. The inner air domain region is parametrized to be twice the acoustic wavelength, while the thickness of the PML is set equal to the wavelength. These settings always allow you to capture two stationary waves in the fluid domain irrespective of the dimension of the transducer, excitation frequency, and speed of sound in the fluid.

PHYSICS AND BOUNDARY CONDITIONS

This tutorial shows how to set up an acoustic-structure interaction model. The structure is a piezoelectric material (PZT-5H) and the surrounding fluid medium is air. The model uses the built-in Acoustic-Piezoelectric Interaction, Frequency Domain multiphysics interface. Here, you solve the Solid Mechanics and Electrostatics equations in the piezoelectric material, which are coupled via the constitutive equations for the piezo material. This coupling is taken care of by the Piezoelectric Effect coupling feature that is located under the Multiphysics node.

The Pressure Acoustics equation is solved in the fluid domain only. Once you demarcate the solid and fluid regions, the boundary condition at the common boundary between the fluid and the piezoelectric solid material is taken care of automatically by the Acoustic-Structure Boundary cooling feature located under the Multiphysics node. At this common boundary, the normal component of the acceleration of the piezo material acts as a sound source for the fluid, while the fluid pressure acts as a boundary load on the piezo material.

The inner surface of the hollow spherical piezo region is assumed to be at electrical ground, while a 100 V (zero-to-peak) potential is applied to its outer surface. The excitation frequency is 25 kHz. A spherical PML is used as the outer layer of the air domain to model absorption of outgoing waves as they propagate infinitely far away from the sound source. For more details on PMLs and the Acoustic-Piezoelectric Interaction, Frequency Domain interface, refer to the *Acoustics Module User's Guide*.

Due to the symmetry of both the geometry and physics set-up, only a 1/8 th section of the geometry is used for modeling purposes. In the Pressure Acoustics use the Symmetry condition and in the Electrostatics interfaces the default and Zero Charge boundary conditions work as the appropriate symmetry boundary conditions. For a Solid Mechanics interface, a Symmetry boundary condition is also used, which sets the normal component of the structural displacement to zero on each of the boundaries that align with the three different planes of symmetry.

IMPLEMENTING THE POLING DIRECTION

In COMSOL Multiphysics, the poling direction of a piezoelectric material is decided based on the choice of coordinate system, in which the material properties are evaluated. By default, the material coordinate system of the piezoelectric material is assumed to be aligned with the global coordinate system, which is Cartesian. Thus, by default, most piezo materials are assumed to be z-polarized. This also means that if you assign the piezo material properties to a user-defined orthogonal coordinate system with unit vectors **x1**, **x2**, and **x3**, then the piezo can be considered to be poled along the **x3** direction. You can use this idea to create a user-defined spherical coordinate system, where **x3** is aligned along the radial direction of the piezo sphere.

Note: COMSOL Multiphysics provides a built-in option for setting up a spherical coordinate system. However, this built-in option assumes that the radial direction is the **x1** direction of such a local coordinate system. Hence, for this model you need to create a custom spherical coordinate system.

A rectangular coordinate system is identified by three mutually perpendicular unit vectors denoted by $\mathbf{e}_x$, $\mathbf{e}_y$, and $\mathbf{e}_z$. A spherical coordinate system is similarly identified by the unit vectors $\mathbf{e}_r$, $\mathbf{e}_\theta$, and $\mathbf{e}_\varphi$, which denote the radial, polar, and azimuthal directions, respectively. The next step is to find a relationship between these two sets of unit vectors using the spatial coordinates (r, θ, φ) of the spherical coordinate system. Equation 1 shows this relationship

$$\begin{aligned}
\mathbf{e}_r &= \cos\varphi\sin\theta\mathbf{e}_x + \sin\varphi\sin\theta\mathbf{e}_y + \cos\theta\mathbf{e}_z \\
\mathbf{e}_\theta &= \cos\varphi\cos\theta\mathbf{e}_x + \sin\varphi\cos\theta\mathbf{e}_y - \sin\theta\mathbf{e}_z \\
\mathbf{e}_\varphi &= -\sin\varphi\mathbf{e}_x + \cos\varphi\mathbf{e}_y + 0\mathbf{e}_z
\end{aligned} \tag{1}$$

Using this equation, you can create a user-defined spherical coordinate system, whose unit vectors, **x1**, **x2**, and **x3**, can be related to $\mathbf{e}_x$, $\mathbf{e}_y$, and $\mathbf{e}_z$ as shown in Equation 2.

$$\begin{aligned}
\mathbf{x1} &= \cos\varphi\cos\theta\mathbf{e}_x + \sin\varphi\cos\theta\mathbf{e}_y - \sin\theta\mathbf{e}_z \\
\mathbf{x2} &= -\sin\varphi\mathbf{e}_x + \cos\varphi\mathbf{e}_y + 0\mathbf{e}_z \\
\mathbf{x3} &= \cos\varphi\sin\theta\mathbf{e}_x + \sin\varphi\sin\theta\mathbf{e}_y + \cos\theta\mathbf{e}_z
\end{aligned} \tag{2}$$

You can also use the relationship between the spatial coordinates of the spherical coordinate system, (r, θ, φ), and the spatial coordinates of the rectangular coordinate system, (x, y, z):

$$\begin{aligned} \varphi &= \operatorname{atan}\frac{y}{x} \\ \theta &= \operatorname{acos}\frac{z}{\sqrt{x^2+y^2+z^2}} \end{aligned} \tag{3}$$

Combining Equation 2 and Equation 3 gives a user-defined spherical coordinate system, whose unit vectors **x1**, **x2**, and **x3** can be represented in terms of the spatial coordinates (x, y, z). Note that this user-defined coordinate system is a cyclic permutation of the default spherical coordinate system with the property that the radial direction is aligned with the **x3** direction. This coordinate system helps you to implement a radial poling direction for the piezo material.

In COMSOL Multiphysics, solid mechanics problems are solved using the Lagrangian formulation, and hence one needs to make a distinction between the Spatial Coordinate System denoted by the coordinate variables (x, y and z) and the Material Coordinate System denoted by the coordinate variables (X, Y and Z). For this reason, when setting up the user-defined spherical coordinate system, use upper case variables (X, Y and Z). For more information on this topic, refer to the solid mechanics theory section in the *Acoustics Module User's Guide*.

Results and Discussion

Figure 2 shows the structural displacement of the piezoelectric material. The displacement profile corresponds to that of the "breathing mode" of the hollow spherical structure. Figure 3 shows the voltage distribution in the piezoelectric material. Figure 4 shows the acoustic pressure variation in the core air domain as well as the adjoining PML region. The color bands denote pressure waves that are propagating radially away from the transducer.

Note that as a result of having a PML on the outer layer, the pressure monotonically drops to zero within this layer.

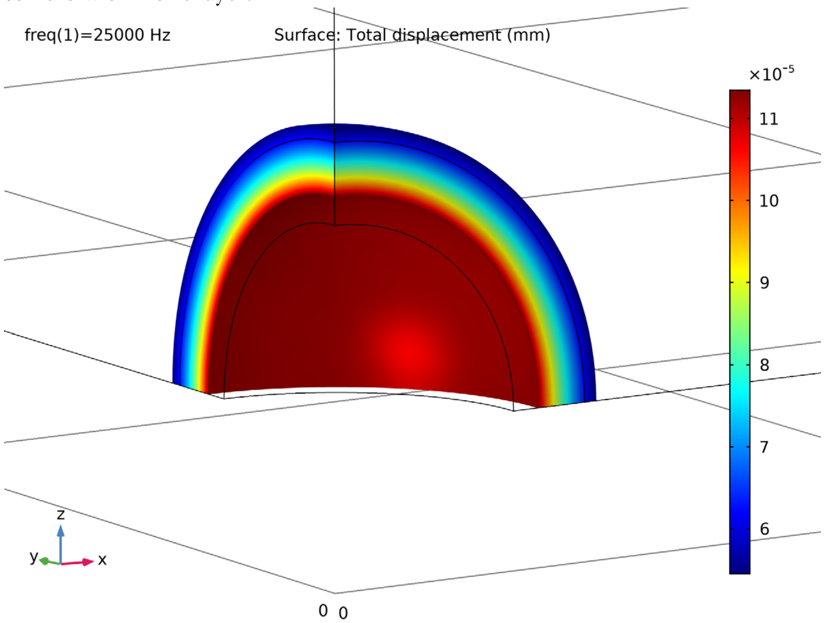


Figure 2: Surface plot of scaled deformation of the piezoelectric material when excited with 100 V at 25 kHz.

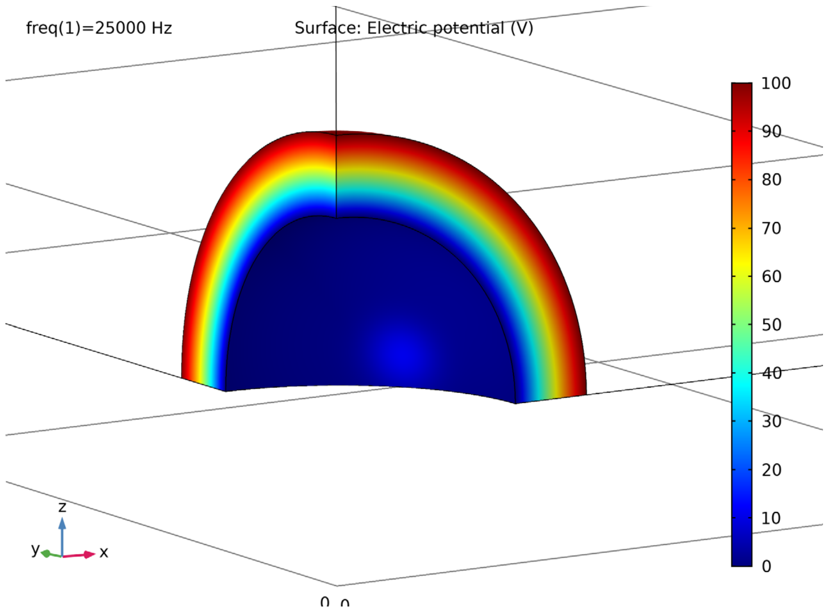


Figure 3: The electric potential, V, in the piezoelectric material.

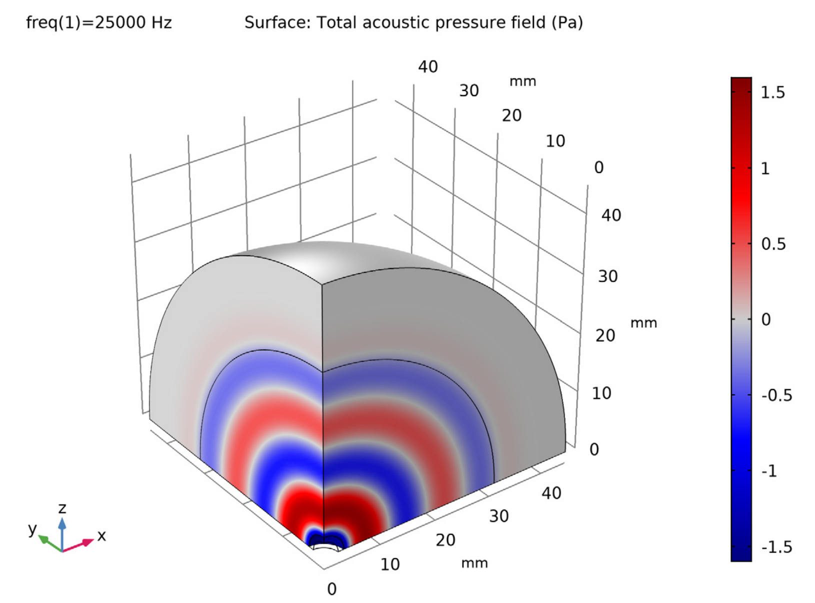


Figure 4: The acoustic pressure in air surrounding the piezoelectric material.

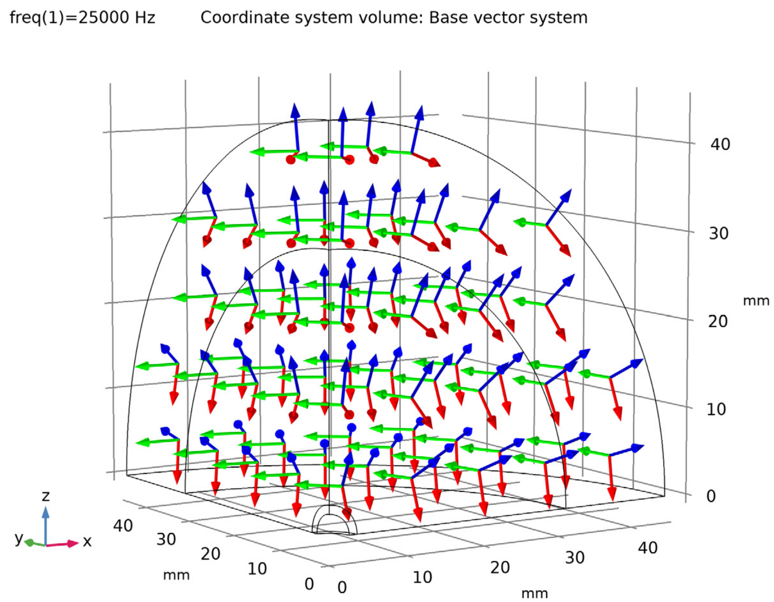


Figure 5: Arrow plot depicting the spherical coordinate system used to set up the poling direction in PZT-5H.

Figure 5 shows an arrow plot of the user-defined spherical coordinate system used to set up the poling direction in the piezoelectric material. Although this coordinate system is only active in the piezo domain, for visual clarity the arrows are plotted in the entire modeling geometry. The red, green and blue arrows correspond to the **x1**, **x2**, and **x3** directions, respectively, in the user-defined local coordinate system. In this case, **x1**, **x2**, and **x3** correspond to the polar, azimuthal, and radial directions, respectively. Because the **x3** direction corresponds to the radial direction, as indicated by the blue arrow, this setting helps to implement the idea that the piezo material is radially poled.

Figure 6 shows a plot of the acoustic pressure variation along the radius of the spherical air region including the PML. The pressure reaches a maximum at the piezo-air interface. The magnitude of the sinusoidally varying pressure decays with increasing distance from the piezo domain. Within the PML region, the pressure monotonically decays to zero, thereby demonstrating perfect absorption. Note that the pressure is continuous at the air-PML interface.

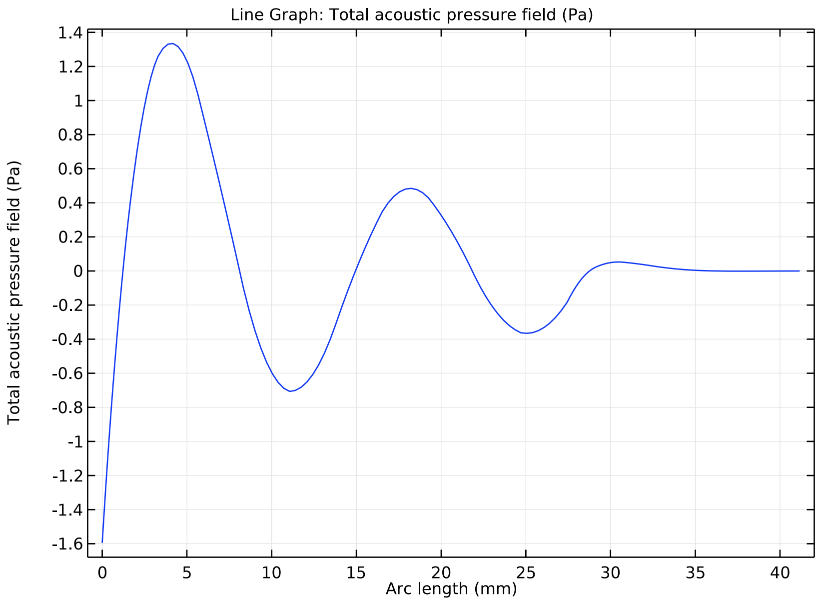


Figure 6: Pressure versus distance along the radius of the spherical air region.

Application Library path: `Acoustics_Module/Piezoelectric_Devices/piezoacoustic_spherical`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

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
f0	25[kHz]	25000 Hz	Excitation frequency
c_fluid	343[m/s]	343 m/s	Speed of sound in fluid
lambda0	c_fluid/f0	0.01372 m	Wavelength
t_piezo	1[mm]	0.001 m	Thickness of piezo layer
r_piezo_inner	2.5[mm]	0.0025 m	Inner radius of piezo
r_tot	3*lambda0+ r_piezo_inner+ t_piezo	0.04466 m	Total radius of geometry
r_PML	lambda0	0.01372 m	Width of PML layer

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)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

4 Click **Show Work Plane**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `r_tot`.

4 In the **Sector angle** text field, type `90`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	r_PML
Layer 2	r_tot-r_PML-r_piezo_inner-t_piezo
Layer 3	t_piezo

6 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Circle 1 (c1)** and choose **Build Selected**.

Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 In the **Work Plane** toolbar, click **Delete**.

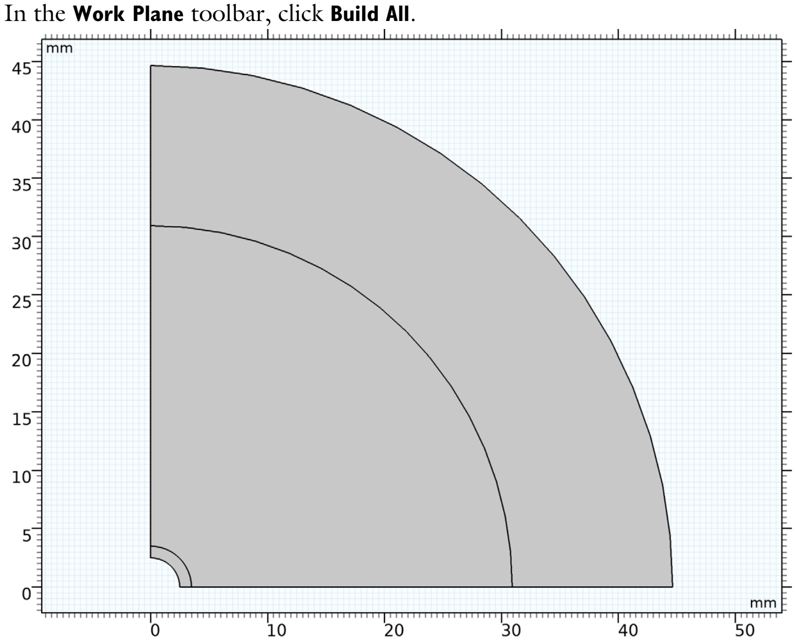
2 Select the object **c1** only.

3 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 On the object **c1**, select Domain 1 only.

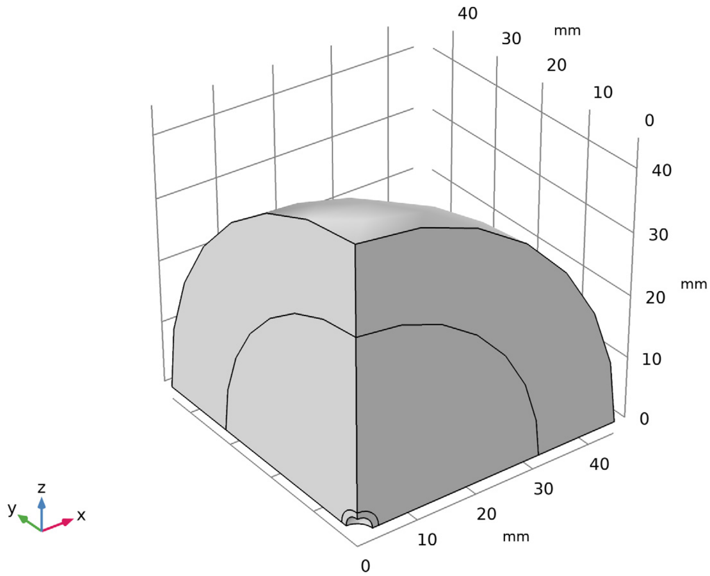
6 In the **Work Plane** toolbar, click **Build All**.



7 In the **Model Builder** window, click **Geometry 1**.

Revolve 1 (rev1)

1 In the **Geometry** toolbar, click **Revolve**.
2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.
3 Click the **Angles** button.
4 In the **End angle** text field, type `-90`.
5 Click **Build All Objects**.



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 Click **Add to Component** in the window toolbar.
5 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.
6 Click **Add to Component** in the window toolbar.
7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat2)
Select Domain 1 only.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.
2 Select Domain 3 only.
3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.
4 From the **Type** list, choose **Spherical**.

Now, add a system of coordinates that represents the spherical system defined in Equation 2.

Base Vector System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.
2 In the **Settings** window for **Base Vector System**, locate the **Settings** section.
3 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	**y**	**z**
x1	`cos(atan2(Y,X))*cos(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`sin(atan2(Y,X))*cos(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`-sin(acos(Z/sqrt(X^2+Y^2+Z^2)))`
x2	`-sin(atan2(Y,X))`	`cos(atan2(Y,X))`	`0`
x3	`cos(atan2(Y,X))*sin(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`sin(atan2(Y,X))*sin(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`cos(acos(Z/sqrt(X^2+Y^2+Z^2)))`

4 Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.
2 Select Domains 2 and 3 only.
3 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Typical Wave Speed for Perfectly Matched Layers** section.
4 In the c_{ref} text field, type `acpr.c`.

Symmetry 1

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

2 Select Boundaries 4, 5, and 12 only.

SOLID MECHANICS (SOLID)

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

2 Select Domain 1 only.

Piezoelectric Material 1

Select the new base vector system you have defined `sys2` as the local system of coordinates.

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

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

3 From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

4 In the **Model Builder** window, click **Solid Mechanics (solid)**.

Symmetry 1

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

2 Select Boundaries 1, 2, and 11 only.

The symmetry planes take care of the fact that you are modeling only 1/8th of the actual structure. Note that the symmetry boundary conditions that you just added in acoustics and solid mechanics only takes care of symmetry here. The electrical symmetry boundary condition is the same as the defaults Zero Charge. Since this boundary condition is applied on these boundaries by default, you do not need to do anything more explicitly.

ELECTROSTATICS (ES)

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

2 Select Domain 1 only.

Ground 1

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

2 Select Boundary 3 only.

Electric Potential 1

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

2 Select Boundary 6 only.

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

4 In the V_0 text field, type `100`.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda0/5`.

5 In the **Minimum element size** text field, type `t_piezo/2`.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 2 only.

Add a size node that defines the size inside of the piezo shell. Define a maximum size equal to half the thickness of the shell. The structural dynamics and electrostatics need to be resolved.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.

2 Select Domain 1 only.

3 In the **Settings** window for **Size**, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `t_piezo/2`.

Distribution 1

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

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `8`.

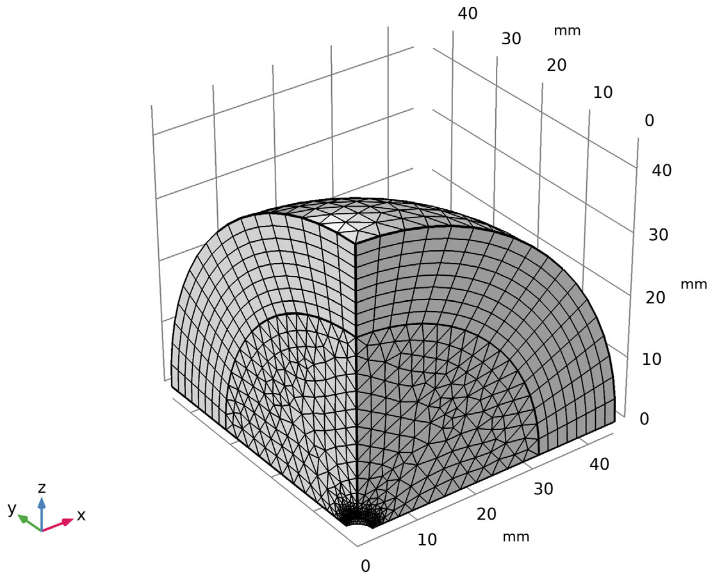
5 In the **Model Builder** window, click **Mesh 1**.

6 In the **Settings** window for **Mesh**, click **Build All**.

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

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

The meshed geometry should look as shown in the figure below.



STUDY 1

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `f0`.

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

RESULTS

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

The plot should look like Figure 4.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

Acoustic Pressure, Isosurfaces (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Potential (es)** node.

2 Right-click **Multislice 1** and choose **Disable**.

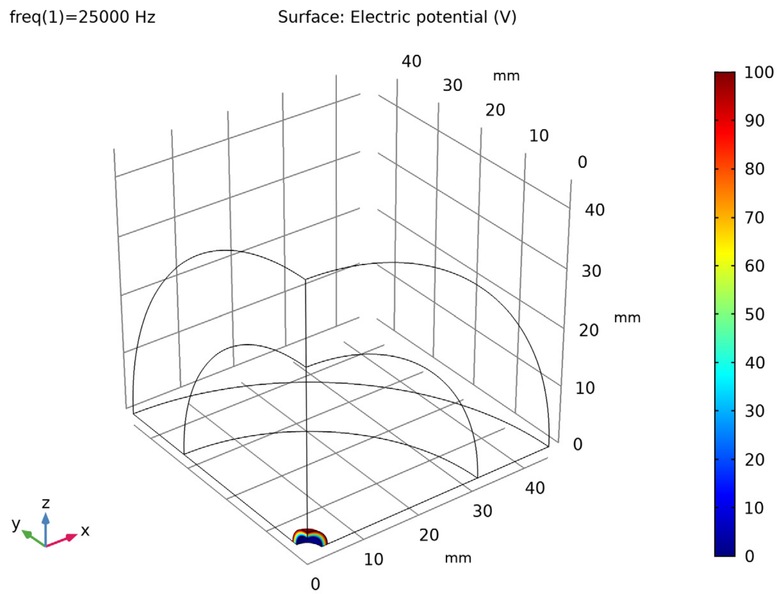
Surface 1

1 In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electrostatics>Electric>V - Electric potential - V**.

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

The plot should look like the figure below. Use the **Zoom Box** tool to zoom on the figure to study it in more detail, this can look like Figure 3.



You can save the view settings to apply them to the plots that contain the piezo domain only (for example, zoomed in on the piezo sphere). Alternatively, you simply use the **Zoom Box** tool.

DEFINITIONS

View 3

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

2 In the **Settings** window for **View**, type `Zoom View` in the **Label** text field.

3 Locate the **View** section. Select the **Lock camera** check box.

RESULTS

Electric Potential (es)

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

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

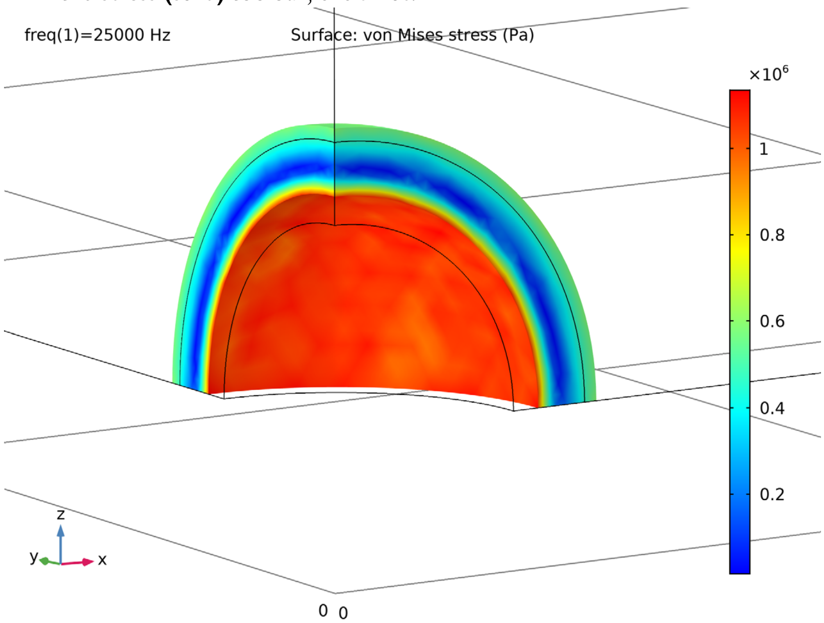
3 From the **View** list, choose **Zoom View**.

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

Stress (solid)

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

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



3D Plot Group 6

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

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

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>solid.disp - Total displacement - m**.

Deformation 1

1 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

2 In the **Displacement** toolbar, click **Plot**.

The plot should look like Figure 2.

3D Plot Group 7

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

2 In the **Settings** window for **3D Plot Group**, type `Coordinate system` in the **Label** text field.

Coordinate System Volume 1

1 In the **Coordinate system** toolbar, click **More Plots** and choose **Coordinate System Volume**.

2 In the **Settings** window for **Coordinate System Volume**, locate the **Coordinate System** section.

3 From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

4 Locate the **Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `5`.

5 Find the **y grid points** subsection. In the **Points** text field, type `5`.

6 Find the **z grid points** subsection. In the **Points** text field, type `5`.

7 In the **Coordinate system** toolbar, click **Plot**.

Go back to the default view.

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

The plot should look like Figure 5.

1D Plot Group 8

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

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

Line Graph 1

1 Right-click **1D Pressure Plot** and choose **Line Graph**.

2 Select Edges 20 and 21 only.

3 In the **1D Pressure Plot** toolbar, click **Plot**.

The plot should look like Figure 6.