Model created in COMSOL Multiphysics 6.4

Surface Acoustic Wave Velocity Calculations from a Unit Cell

Introduction

FEM simulations can be used to obtain the surface acoustic wave (SAW) velocity and related parameters, such as the squared electromechanical coupling coefficient, or k^2, and reflectivity, or k_p, for different configurations. Such parameters are used as input for various analytical and semi-analytical methods of SAW device design, for example, the Delta-Function model and the COM method. The typical setup for FEM simulations is a unit-cell model encompassing one wavelength with periodic boundary conditions and an **Eigenfrequency** study. This tutorial investigates unit cells of 128 YX-cut LiNbO3 crystal with the size chosen for a center frequency of about 433 MHz. The first unit cell studied has a free surface, that is, a surface without IDT or electric terminals. The second unit cell has IDT, that is, two grounded aluminum electrodes. The third unit cell studied has a fully grounded top surface.

Model Definition

To model the unit cell, a **Piezoelectricity** interface with a **Periodic Condition** feature are used. To define the crystal cut, a **Rotated System** feature is used. In addition, a **Perfectly Matched Layer** is used to efficiently model SAWs propagating in thick substrates. To simulate the SAW phenomenon, the **Eigenfrequency** study is performed to solve for two eigenfrequencies around 433 MHz.

The geometric parameters are summarized in the first table in the Modeling Instructions section. The parameterized geometry facilitates design optimization. The model geometry includes three unit cells with free surface, IDT, and a grounded bottom surface, as shown in Figure 1.

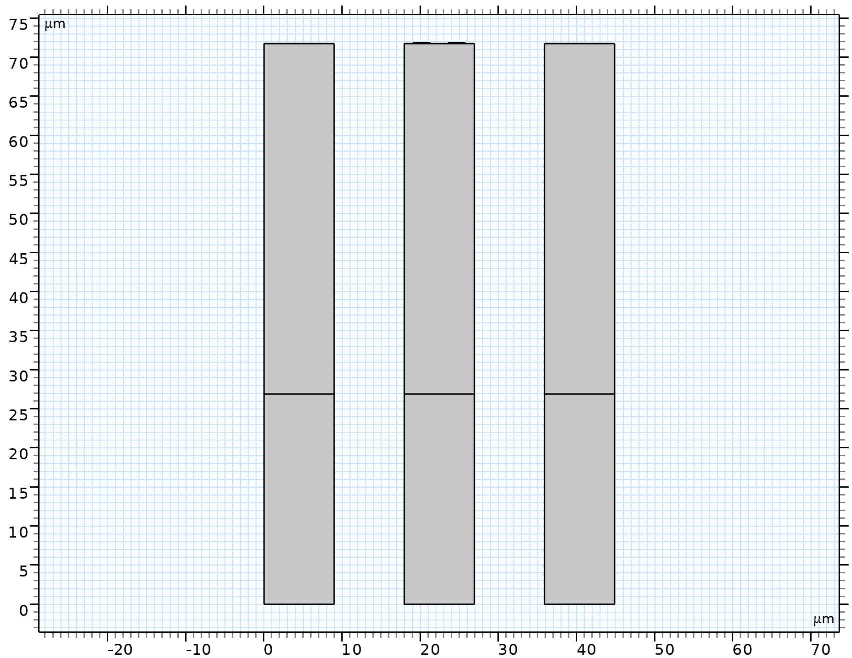


Figure 1: The model geometry includes three unit cells with free surface (left), IDT (center), and a grounded bottom surface (right).

Note that it is necessary to enable the Out-of-plane mode extension (time-harmonic) in the **Solid Mechanics** interface for the 2D component to obtain results for all three displacement components. Otherwise, one displacement component and its derivatives would be zero.

From an **Eigenfrequency** study, f_1 (resonance) and f_2 (anti-resonance) are obtained. In the case of free surface or grounded, f_1 and f_2 should be equal. The related parameters c_{free}, c_{IDT}, c_{grounded}, k^2, and k_{p} can be calculated in results processing using a **Global Evaluation** feature. Simply define the parameter and the expressions to be evaluated. The SAW velocity for the free surface is defined as

$$c_{\mathrm{free}} = \lambda\frac{(f_{1,\,\mathrm{free}} + f_{2,\,\mathrm{free}})}{2}$$

with $f_{1,\mathrm{free}}$ and $f_{2,\mathrm{free}}$ denoting the resonance and anti-resonance frequencies, respectively.

Similarly, in the case with IDT, the velocity is defined as

$$c_{\mathrm{IDT}} = \lambda\frac{(f_{1,\,\mathrm{IDT}} + f_{2,\,\mathrm{IDT}})}{2}$$

with $f_{1,\mathrm{IDT}}$ and $f_{2,\mathrm{IDT}}$ denoting the resonance and anti-resonance frequencies, respectively. Lastly, in the grounded case, the velocity is defined as

$$c_{\mathrm{grounded}} = \lambda\frac{(f_{1,\,\mathrm{grounded}} + f_{2,\,\mathrm{grounded}})}{2}$$

with $f_{1,\mathrm{IDT}}$ and $f_{2,\mathrm{IDT}}$ denoting the resonance and anti-resonance frequencies, respectively. The squared electromechanical coupling coefficient, k^2, is defined by

$$k^2 = 2\frac{(c_{\mathrm{free}} - c_{\mathrm{grounded}})}{c_{\mathrm{free}}}$$

and the reflectivity, k_{p}, is defined by

$$k_{\mathrm{p}} = \pi\frac{(f_{2,\,\mathrm{IDT}} - f_{1,\,\mathrm{IDT}})}{f_{1,\,\mathrm{free}}}$$

Results and Discussion

Based on the results of the **Eigenfrequency** studies for each case, the parameters of interest are calculated using a **Global Evaluation** feature. They are summarized in Table 1.

TABLE 1: CALCULATED SAW PARAMETERS

Parameter	**Value**
Velocity, free (m/s)	3997.8
f1, free (MHz)	445.7
f2, free (MHz)	445.7
Velocity IDT	3905.7
f1, IDT (MHz)	433.7
f2, IDT (MHz)	437.2
Velocity, grounded	3888.0
f1, grounded (MHz)	433.4
f2, grounded (MHz)	433.4
k2 (1)	0.05494
kp (1)	0.02457

Unit-cell modeling shown in this tutorial can be performed as verification before moving forward for more complex, full device configurations. This approach can give accurate results for different configurations, including double-electrode designs, SPUDT (unidirectional IDTs), trapezoid-shaped electrodes, multilayered structures, and so on. This approach is also valid for structures other than IDTs: all kinds of reflectors, different projections, grooves, and so on. The FEM formulation solves for the SAW velocity regardless of the IDT size and grating pattern, and automatically includes parameters such as electrode mass loading, internal reflections, scattering, and BAW radiation.

Provided that the model setup is correct, all the obtained parameters are thoroughly dependent on the material data used. That said, one has to be very careful with the material properties of a piezoelectric crystal and IDT, and has to ensure they are consistent with the experimental setup. Any notable discrepancies may end up in a different phase velocity and thus a different resonance frequency. The latter may significantly affect any frequency response of a complex SAW device. This is another reason to run a unit-cell model for verification, especially when new untested set of constants are involved.

Every **Eigenfrequency** study here is designed to return two SAW modes. This is achieved via a good initial guess specified in the **Search for eigenfrequencies around shift** text field. When performing your own simulation, you might need to adjust this initial guess. It might be a good idea to manually investigate all the obtained modes to ensure that they are of the SAW class.

To evaluate the aforementioned parameters in the most automated way, one needs to reach every solution and every mode within that solution independent of the dataset used. This can be implemented via a special `withsol` operator with an additional `setind`-argument. For example, calling `withsol('sol3',real(freq),setind(lambda,2))` will return the real part of the second eigenfrequency obtained in the third study. More information can be found in the *COMSOL Multiphysics Reference Manual*; search for “Built-In Operators.”

Reference

1. K. Hashimoto, *Surface Acoustic Wave Devices in Telecommunications: Modelling and Simulation*, Springer-Verlag Berlin, Heidelberg, 2000; DOI: doi.org/10.1007/978-3-662-04223-6.

Application Library path: `MEMS_Module/Piezoelectric_Devices/saw_velocity_calculation`

Modeling Instructions

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

NEW

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

Start by creating a new 2D model with three **Piezoelectricity** multiphysics interfaces for modeling free, IDT and grounded unit cells. Also select **Eigenfrequency** for the first study.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **AC/DC** > **Electromagnetics and Mechanics** > **Piezoelectricity** > **Piezoelectricity, Solid**.
3 Click **Add**.
4 Click **Add**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Eigenfrequency**.
8 Click **Done**.

Define and specify the parameters of the model.

GLOBAL DEFINITIONS

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
lambdaD	8.97[um]	8.97E-6 m	Design period
h_el	100[nm]	1E-7 m	Electrode height
l_el	0.25*lambdaD	2.2425E-6 m	Electrode width
W	100*lambdaD	8.97E-4 m	Aperture
h_sub	8*lambdaD	7.176E-5 m	Substrate height
L_PML	3*lambdaD	2.691E-5 m	PML width
c_0	3997.8[m/s]	3997.8 m/s	Expected free SAW velocity
f_0	0.98*c_0/lambdaD	4.3677E8 1/s	Initial guess for eigenfrequency

Create the geometry for the free unit cell. Use microns as the geometry unit.

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 **µm**.

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 lambdaD.

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

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

Layer name	Thickness (µm)
Layer 1	L_PML

6 Click **Build Selected**.

Create the geometry for the unit cell with IDT.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type lambdaD*2.

5 Click **Build Selected**.

Rectangle 2 (r2)

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 l_el.

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

5 Locate the **Position** section. In the **x** text field, type lambdaD*2+lambdaD/8.

6 In the **y** text field, type h_sub.

7 Click **Build Selected**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r2** only.

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

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type 2.

6 Locate the **Displacement** section. In the **x** text field, type lambdaD/2.

7 Click **Build Selected**.

Create the geometry for the grounded unit cell.

Copy 2 (copy2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

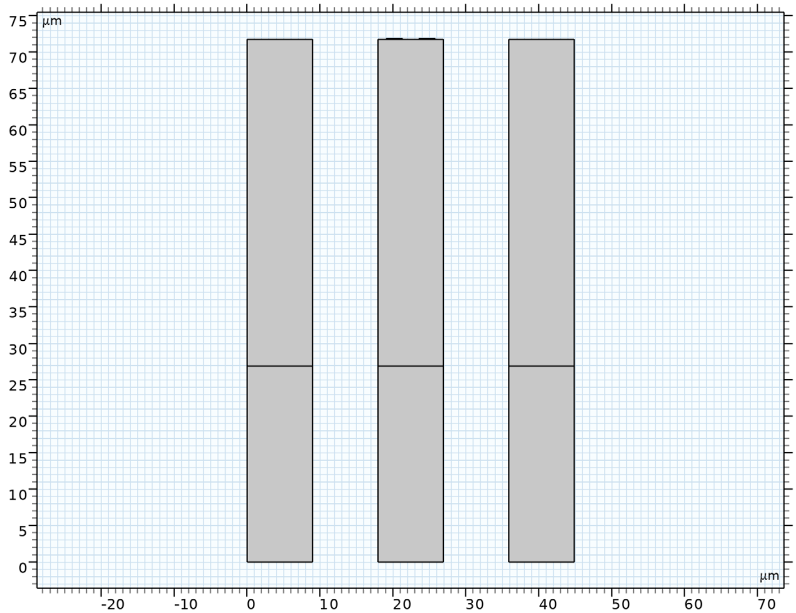
2 Click the **Select Box** button in the **Graphics** toolbar.

3 Select the object **r1** only.

4 In the **Settings** window for **Copy**, locate the **Displacement** section.

5 In the **x** text field, type lambdaD*4.

6 Click **Build Selected**.



Define selections for the various domains and boundaries.

DEFINITIONS

Substrate

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Substrate` in the **Label** text field.

3 Select Domains 1–4, 7, and 8 only.

PML

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `PML` in the **Label** text field.

3 Select Domains 1, 3, and 7 only.

IDT

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `IDT` in the **Label** text field.

3 Select Domains 5 and 6 only.

Free Unit Cell

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Free Unit Cell` in the **Label** text field.
3. Click the **Select Box** button in the **Graphics** toolbar.
4. Select Domains 1 and 2 only.

IDT Unit Cell

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `IDT Unit Cell` in the **Label** text field.
3. Click the **Select Box** button in the **Graphics** toolbar.
4. Select Domains 3–6 only.

Grounded Unit Cell

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Grounded Unit Cell` in the **Label** text field.
3. Click the **Select Box** button in the **Graphics** toolbar.
4. Select Domains 7 and 8 only.

Periodic

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, locate the **Input Entities** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. In the **Label** text field, type `Periodic`.
5. Select Boundaries 1, 3, 6–8, 10, 23–25, 27, 30, and 31 only.

Fixed

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Fixed` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 2, 9, and 26 only.

Define operators that could be useful in results processing.

Maximum 1 (maxop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.
2. In the **Settings** window for **Maximum**, locate the **Source Selection** section.
3. From the **Selection** list, choose **Free Unit Cell**.

Maximum 2 (maxop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.
2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.
3 From the **Selection** list, choose **IDT Unit Cell**.

Maximum 3 (maxop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.
2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.
3 From the **Selection** list, choose **Grounded Unit Cell**.

Define a rotated system for the 128 YX-cut lithium niobate.

Rotated System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.
2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.
3 From the **Input method** list, choose **General rotation**.
4 Find the **Euler angles** subsection. In the β text field, type `-38[deg]-90[deg]`.

Define a perfectly matched layer.

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.
2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **PML**.

Add materials and assign the domains they belong to.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **MEMS** > **Metals** > **Al - Aluminum**.
4 Click the **Add to Component** button in the window toolbar.

MATERIALS

Al - Aluminum (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2 From the **Selection** list, choose **IDT**.

ADD MATERIAL

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

2 In the tree, select **Piezoelectric** > **Lithium Niobate**.

3 Click the **Add to Component** button in the window toolbar.

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Lithium Niobate (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Substrate**.

Specify the settings for the **Electrostatics** and **Solid Mechanics** interfaces in the free unit cell.

ELECTROSTATICS (ES)

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

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

3 From the **Selection** list, choose **Free Unit Cell**.

4 Locate the **Thickness** section. In the d text field, type `W`.

Periodic Condition 1

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

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

3 From the **Selection** list, choose **Periodic**.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Free Unit Cell**.

4 Locate the **2D Approximation** section. Select the **Out-of-plane mode extension (time-harmonic)** checkbox.

5 Locate the **Thickness** section. In the d text field, type `W`.

Piezoelectric Material 1

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 **Rotated System 2 (sys2)**.

Periodic Condition 1

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

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

3 From the **Selection** list, choose **Periodic**.

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Fixed**.

Specify the settings for the **Electrostatics** and **Solid Mechanics** interfaces in the IDT unit cell.

ELECTROSTATICS 2 (ES2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics 2 (es2)**.

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

3 From the **Selection** list, choose **IDT Unit Cell**.

4 Locate the **Thickness** section. In the d text field, type `W`.

Boundary Terminal 1

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

Periodic Condition 1

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

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

3 From the **Selection** list, choose **Periodic**.

Charge Conservation, Piezoelectric 1

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

2 In the **Settings** window for **Charge Conservation, Piezoelectric**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Substrate**.

Boundary Terminal 1

1 In the **Model Builder** window, click **Boundary Terminal 1**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 19 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.

7 From the **Terminal type** list, choose **Voltage**.

Ground 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 14 in the **Selection** text field.

5 Click **OK**.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **IDT Unit Cell**.

4 Locate the **2D Approximation** section. Select the **Out-of-plane mode extension (time-harmonic)** checkbox.

5 Locate the **Thickness** section. In the d text field, type W.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1) > Solid Mechanics 2 (solid2)** click **Piezoelectric Material 1**.

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

3 From the **Selection** list, choose **Substrate**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Periodic Condition 1

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

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

3 From the **Selection** list, choose **Periodic**.

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Fixed**.

Specify the settings for the **Electrostatics** and **Solid Mechanics** interfaces in the grounded unit cell.

ELECTROSTATICS 3 (ES3)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics 3 (es3)**.

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

3 From the **Selection** list, choose **Grounded Unit Cell**.

4 Locate the **Thickness** section. In the d text field, type `W`.

Periodic Condition 1

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

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

3 From the **Selection** list, choose **Periodic**.

Ground 1

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

2 Select Boundary 29 only.

SOLID MECHANICS 3 (SOLID3)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 3 (solid3)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Grounded Unit Cell**.

4 Locate the **2D Approximation** section. Select the **Out-of-plane mode extension (time-harmonic)** checkbox.

5 Locate the **Thickness** section. In the d text field, type `W`.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics 3 (solid3)** 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 **Rotated System 2 (sys2)**.

Periodic Condition 1

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

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

3 From the **Selection** list, choose **Periodic**.

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Fixed**.

Create the mesh for the free, IDT and grounded unit cells.

MESH FREE

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

2 In the **Settings** window for **Mesh**, type `Mesh Free` in the **Label** text field.

3 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Distribution 1, Free Triangular 1, Identical Mesh 1, Identical Mesh 2, Identical Mesh 3, Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh Free**, Ctrl-click to select **Identical Mesh 1**, **Identical Mesh 2**, **Identical Mesh 3**, **Distribution 1**, **Free Triangular 1**, and **Mapped 1**.

2 Right-click and choose **Delete**.

Size

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

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambdaD/20`.

4 In the **Minimum element size** text field, type `lambdaD/20`.

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Boundaries 1, 3, 6, and 7 only.

Distribution 1

1 Right-click **Edge 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 10.

Mapped 1

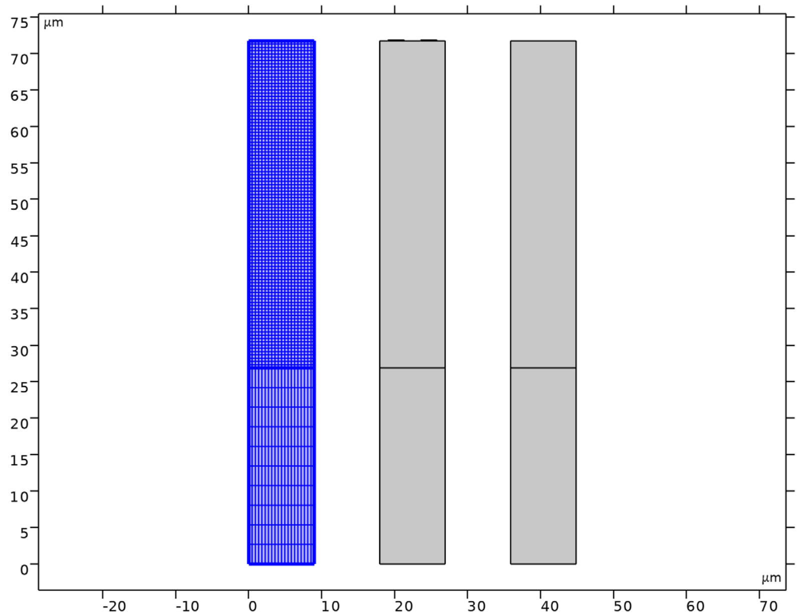
1 In the **Mesh** toolbar, click **Mapped**.

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

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

4 Select Domains 1 and 2 only.

5 Click **Build All**.



MESH IDT

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, type Mesh IDT in the **Label** text field.

3 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Distribution 1, Free Triangular 1, Identical Mesh 1, Identical Mesh 2, Identical Mesh 3, Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Meshes** > **Mesh IDT**, Ctrl-click to select **Identical Mesh 1**, **Identical Mesh 2**, **Identical Mesh 3**, **Distribution 1**, **Free Triangular 1**, and **Mapped 1**.

2 Right-click and choose **Delete**.

Size

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

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambdaD/20`.

4 In the **Minimum element size** text field, type `lambdaD/20`.

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Boundaries 8 and 10 only.

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.

2 Select Boundary 8 only.

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

4 In the **Number of elements** text field, type `10`.

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `5 6` in the **Selection** text field.

6 Click **OK**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `16 21` in the **Selection** text field.

5 Click **OK**.

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

7 In the **Number of elements** text field, type 2.

8 Click **Build Selected**.

Edge 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 12 17 22 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Edge**, click **Build Selected**.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh IDT** and choose **Copying Operations** > **Copy Edge**.

2 In the **Settings** window for **Copy Edge**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 12 14 17 19 22 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

7 Click to select the **Activate Selection** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog, type 9 11 in the **Selection** text field.

10 Click **OK**.

11 In the **Settings** window for **Copy Edge**, click **Build Selected**.

Mapped 2

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 3 4 in the **Selection** text field.

6 Click **OK**.

Distribution 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `8` in the **Selection** text field.

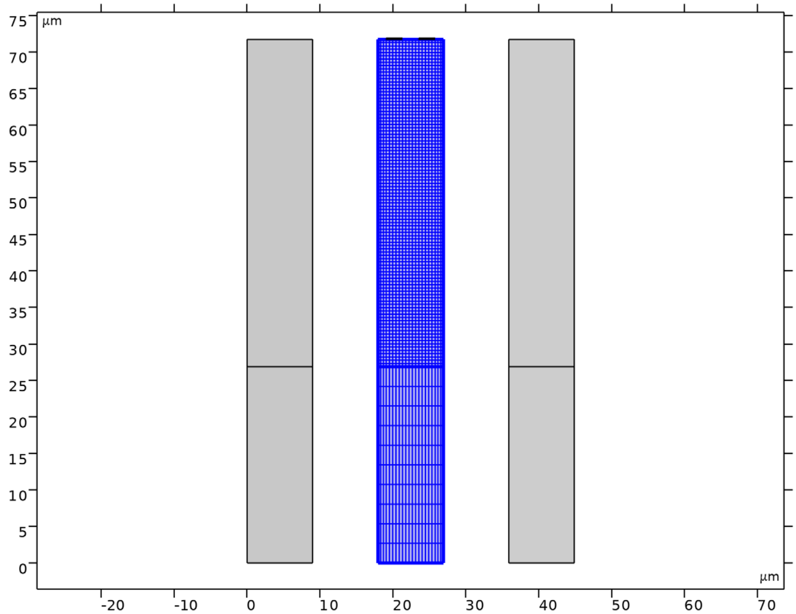
5 Click **OK**.

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

7 In the **Number of elements** text field, type `10`.

Mapped 2

In the **Model Builder** window, right-click **Mapped 2** and choose **Build All**.



MESH FREE

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

MESH GROUNDED

In the **Settings** window for **Mesh**, type `Mesh Grounded` in the **Label** text field.

Edge 1

1 In the **Model Builder** window, expand the **Mesh Grounded** node, then click **Edge 1**.

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

3 Click **Clear Selection**.

4 Select Boundaries 25, 27, 30, and 31 only.

Distribution 1

1 In the **Model Builder** window, expand the **Edge 1** node, then click **Distribution 1**.

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

3 Click **Clear Selection**.

4 Select Boundaries 25 and 30 only.

Mapped 1

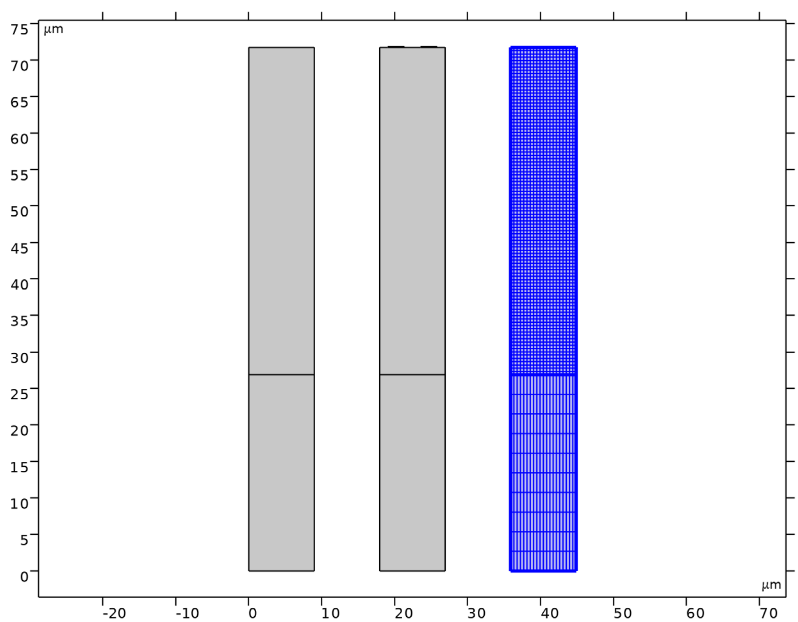
1 In the **Model Builder** window, under **Component 1 (comp1)** > **Meshes** > **Mesh Grounded** click **Mapped 1**.

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

3 Click **Clear Selection**.

4 Select Domains 7 and 8 only.

5 Click **Build All**.



FREE

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Free` in the **Label** text field.

Set up **Eigenfrequency** study to search around `f_0` MHz.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Free** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `2`.
4 From the **Unit** list, choose **MHz**.
5 In the **Search for eigenfrequencies around shift** text field, type `f_0`.
6 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Electrostatics 2 (es2)**, **Solid Mechanics 2 (solid2)**, **Electrostatics 3 (es3)**, and **Solid Mechanics 3 (solid3)**.
7 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkboxes for **Piezoelectricity 2 (pze2)** and **Piezoelectricity 3 (pze3)**.

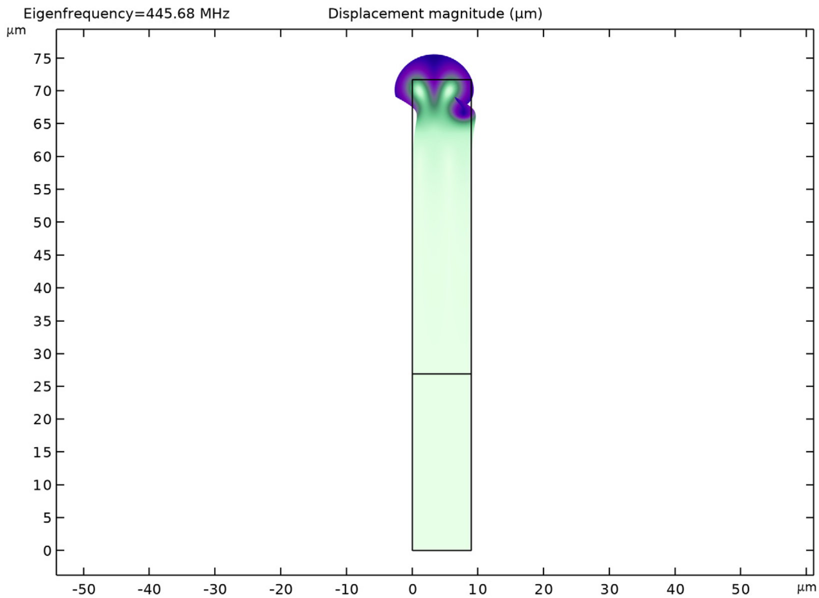
Instead of using all geometric entities, select only Free Unit Cell.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Compile Equations: Eigenfrequency**.
3 In the **Settings** window for **Compile Equations**, locate the **Geometric Entity Selection** section.
4 From the **Use entities** list, choose **Selected**.
5 Under **Selections**, click **Add**.
6 In the **Add** dialog, select **Free Unit Cell** in the **Selections** list.
7 Click **OK**.
8 In the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)



1 From the **Home** menu, choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Eigenfrequency**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Electrostatics (es)**, **Solid Mechanics (solid)**, **Electrostatics 3 (es3)**, and **Solid Mechanics 3 (solid3)**.

4 Find the **Multiphysics couplings in study** subsection. In the table, clear the **Solve** checkboxes for **Piezoelectricity 1 (pze1)** and **Piezoelectricity 3 (pze3)**.

5 Click the **Add Study** button in the window toolbar.

6 From the **Home** menu, choose **Add Study**.

IDT

In the **Settings** window for **Study**, type `IDT` in the **Label** text field.

Set up **Eigenfrequency** study to search around `f_0` MHz.

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Mode Shape (solid)** node, then click **IDT** > **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `2`.

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

5 In the **Search for eigenfrequencies around shift** text field, type `f_0`.

6 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh IDT

Instead of using all geometric entities, select only IDT Unit Cell.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Compile Equations: Eigenfrequency**.

3 In the **Settings** window for **Compile Equations**, locate the **Geometric Entity Selection** section.

4 From the **Use entities** list, choose **Selected**.

5 Under **Selections**, click **Add**.

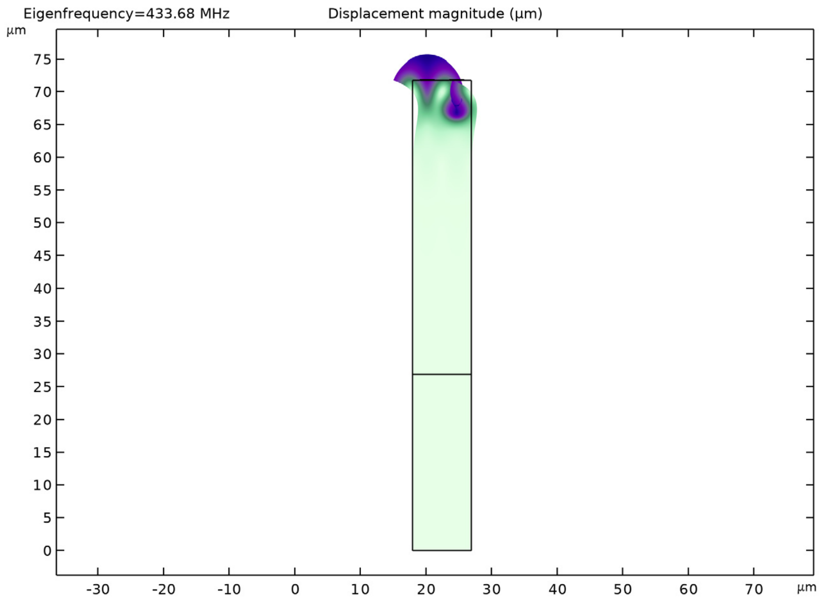
6 In the **Add** dialog, select **IDT Unit Cell** in the **Selections** list.

7 Click **OK**.

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

RESULTS

Mode Shape (solid2)



1 From the **Home** menu, choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Eigenfrequency**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Electrostatics (es)**, **Solid Mechanics (solid)**, **Electrostatics 2 (es2)**, and **Solid Mechanics 2 (solid2)**.

4 Find the **Multiphysics couplings in study** subsection. In the table, clear the **Solve** checkboxes for **Piezoelectricity 1 (pze1)** and **Piezoelectricity 2 (pze2)**.

5 Click the **Add Study** button in the window toolbar.

6 From the **Home** menu, choose **Add Study**.

GROUNDED

In the **Settings** window for **Study**, type `Grounded` in the **Label** text field.

Set up **Eigenfrequency** study to search around `f_0` MHz.

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Mode Shape (solid2)** node, then click **Grounded** > **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `2`.

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

5 In the **Search for eigenfrequencies around shift** text field, type `f_0`.

6 Locate the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh Grounded

Instead of using all geometric entities, select only Grounded Unit Cell.

Solution 3 (sol3)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Compile Equations: Eigenfrequency**.

3 In the **Settings** window for **Compile Equations**, locate the **Geometric Entity Selection** section.

4 From the **Use entities** list, choose **Selected**.

5 Under **Selections**, click **Add**.

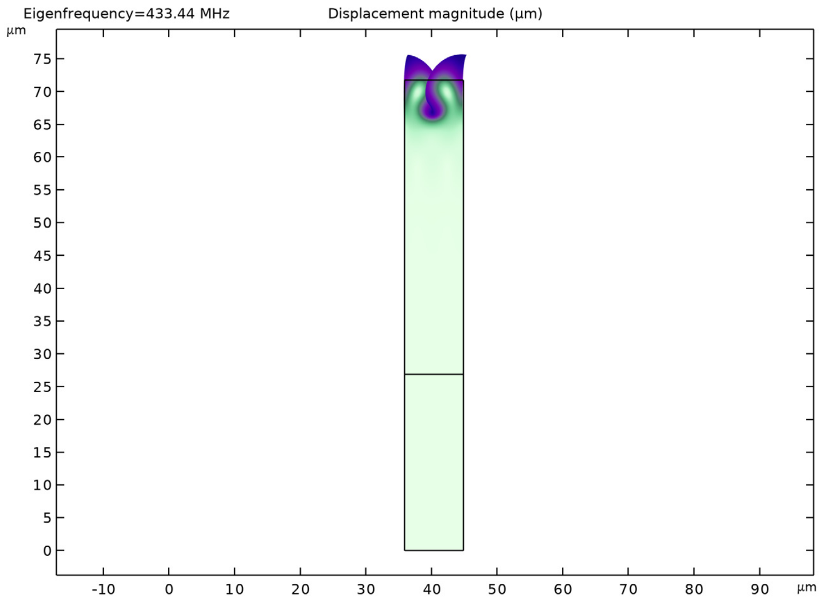
6 In the **Add** dialog, select **Grounded Unit Cell** in the **Selections** list.

7 Click **OK**.

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

RESULTS

Mode Shape (solid3)



Add an evaluation group for evaluating parameters such as velocity, eigenfrequency, k^2, and kp.

Frequencies, Velocities, and Coefficients

1 In the **Model Builder** window, expand the **Mode Shape (solid3)** node.

2 Right-click **Results** and choose **Evaluation Group**.

3 In the **Settings** window for **Evaluation Group**, type `Frequencies, Velocities, and Coefficients` in the **Label** text field.

4 Locate the **Transformation** section. Select the **Transpose** checkbox.

5 Click to expand the **Format** section. From the **Include parameters** list, choose **Off**.

Global Evaluation 1

1 Right-click **Frequencies, Velocities, and Coefficients** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`0.5*lambdaD*(withsol('sol1',real(freq),setind(lambda,1))+withsol('sol1',real(freq),setind(lambda,2)))`	`m/s`	`Velocity, free`
`withsol('sol1',real(freq),setind(lambda,1))`	`MHz`	`f1 free`
`withsol('sol1',real(freq),setind(lambda,2))`	`MHz`	`f2 free`

4 Locate the **Data** section. From the **Dataset** list, choose **Free/Solution 1 (sol1)**.

5 From the **Eigenfrequency selection** list, choose **Last**.

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

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

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

Expression	Unit	Description
`0.5*lambdaD*(withsol('sol2',real(freq),setind(lambda,1))+withsol('sol2',real(freq),setind(lambda,2)))`	`m/s`	`Velocity, IDT`
`withsol('sol2',real(freq),setind(lambda,1))`	`MHz`	`f1 IDT`
`withsol('sol2',real(freq),setind(lambda,2))`	`MHz`	`f2 IDT`

Global Evaluation 3

1 Right-click **Global Evaluation 2** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Grounded/Solution 3 (sol3)**.

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

Expression	Unit	Description
`0.5*lambdaD*(withsol('sol3',real(freq),setind(lambda,1))+withsol('sol3',real(freq),setind(lambda,2)))`	`m/s`	`Velocity, grounded`

Expression	Unit	Description
`withsol('sol3',real(freq), setind(lambda,1))`	MHz	f1 grounded
`withsol('sol3',real(freq), setind(lambda,2))`	MHz	f2 grounded

Global Evaluation 4

1 Right-click **Global Evaluation 3** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Eigenfrequency selection** list, choose **First**.

4 Locate the **Expressions** section. Click **Clear Table**.

5 In the table, enter the following settings:

Expression	Unit	Description
`2*(0.5*lambdaD*(withsol('sol1',real(freq), setind(lambda,1))+withsol('sol1',real(freq), setind(lambda,2)))-0.5*lambdaD* (withsol('sol3',real(freq),setind(lambda,1))+ withsol('sol3',real(freq),setind(lambda, 2))))/(0.5*lambdaD*(withsol('sol1', real(freq),setind(lambda,1))+withsol('sol1', real(freq),setind(lambda,2))))`	1	k^2

Global Evaluation 5

1 Right-click **Global Evaluation 4** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 Click **Clear Table**.

4 In the table, enter the following settings:

Expression	Unit	Description
`pi*abs(withsol('sol2',real(freq), setind(lambda,2))-withsol('sol2',real(freq), setind(lambda,1)))/withsol('sol1',real(freq), setind(lambda,1))`	1	kp

Frequencies, Velocities, and Coefficients

1 In the **Model Builder** window, click **Frequencies, Velocities, and Coefficients**.

2 In the **Frequencies, Velocities, and Coefficients** toolbar, click **Evaluate**.