Model created in COMSOL Multiphysics 6.4

Phononic Crystal

Introduction

Phononic crystals are periodic arrangements of macroscopic unit cells that are designed to manipulate elastic-wave propagation. Wave guides, wave filters, and negative refraction lenses are common examples of applications. Unit cells can usually comprise inhomogeneous material properties, arrangements of voids, or resonating elements. When the wavelength of the traveling wave in the crystal approaches twice the characteristic size of the unit cell, Bragg scattering occurs and the crystal behaves as a wave filter in the corresponding frequency range, which is then called the band gap. A similar effect can be obtained with local resonating elements that open band gaps even in the subwavelength regime. Since S and P waves have different equivalent wave speeds, band gaps will open at different frequency ranges for longitudinal and transverse waves.

This model shows how to use the scattered-field formulation to compute the transmission coefficient for impinging plane elastic S and P waves onto a finite-size phononic crystal. The transmission tends to zero in the frequency ranges corresponding to the S- and P-wave band gaps, as predicted by a preliminary study aimed at computing the dispersion relation.

Model Definition

The model comprises two different components:

- The first component analyzes the wave-propagation properties in an infinite repetition of unit cells. This is done by computing the modes of a single unit cell equipped with Floquet-periodic conditions.
- The second component is an assembly of unit cells surrounded by a homogeneous background material. It simulates what happens when waves impinge on the separation surface between the crystal and the background.

The crystal is obtained by cutting a square pattern of square holes whose centers are separated by a distance equal to twice the side of the hole itself. The material used is aluminum, with material properties listed in Table 1.

TABLE 1: MATERIAL PARAMETERS.

PARAMETER	VALUE
E	69.9 GPa
ρ	2700 kg/m^3
ν	0.3

Different equivalent choices can be made for the geometry of the unit cell. Figure 1 shows the one used in the model.

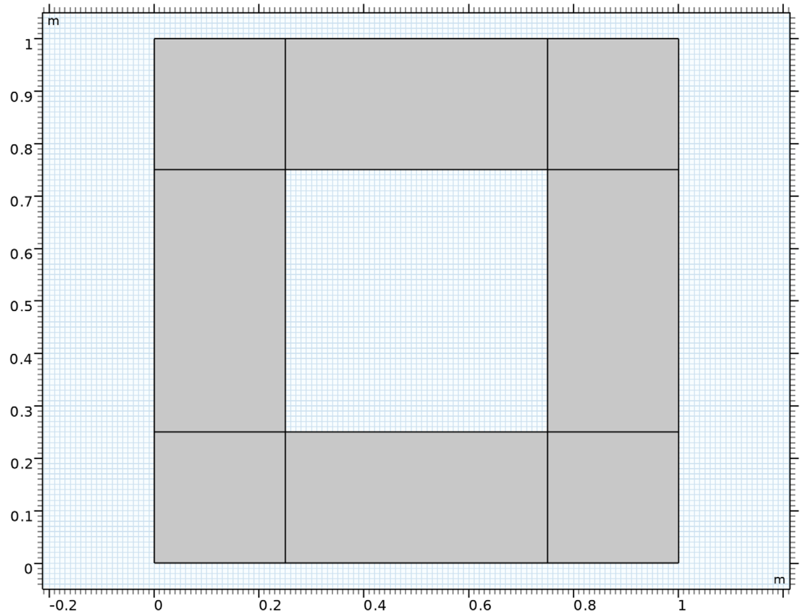

Figure 1: Unit cell used in the analysis of the dispersion relation.

The assembly of unit cells is thought to be finite in the horizontal direction but infinite in the vertical one. This situation can be simulated numerically by using a monodimensional array of cells in the direction along which the crystal is finite, and periodic conditions in the direction along which the crystal is infinite (see Figure 2). Knowing the desired incident field for which the analysis of the scattering from the crystal is to be performed, the problem can be formulated using the scattered-field formulation, thus solving for the scattered field only.

Note: You can find more information about the scattered-field formulation in the Scattered-Field Formulation for Elastic Waves model. Application Library path: **Acoustics_Module/Elastic_Waves/scattered_field_elastic_waves**

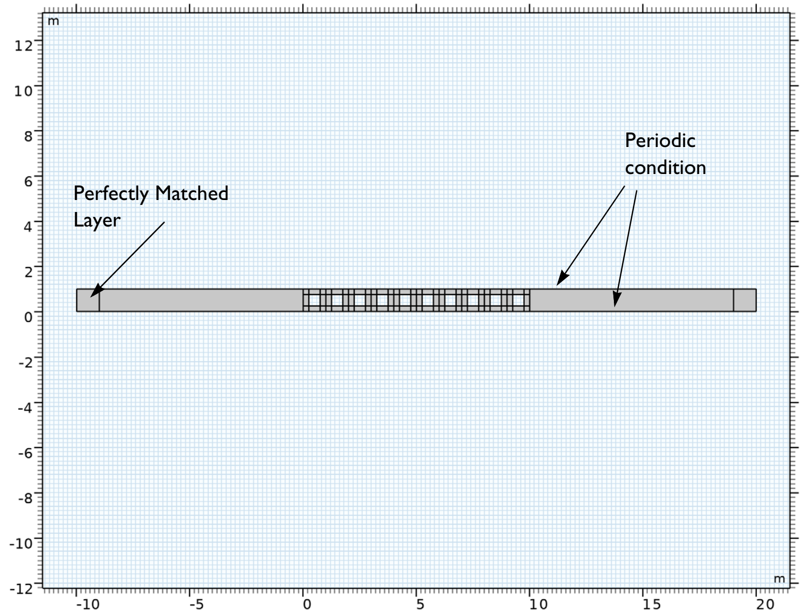

Figure 2: A strip of 1-by-10 unit cells used to compute reflection and transmission at a surface shared between the phononic crystal and the ambient background material. The top and bottom edges are connected with periodic conditions since the crystal is thought to be infinitely extended in the vertical direction.

Results and Discussion

Figure 3 shows the dispersion relation as computed from the analysis on the unit cell. The computed branches are given color values that are computed based on the longitudinal wave polarization:

$$p = \frac{\int u^2}{\int (u^2 + v^2)} \tag{1}$$

Using such a parameter, Bloch waves dominated by transverse displacements are highlighted in blue, while Bloch waves dominated by longitudinal displacements are highlighted in red. A band gap is opened by Bragg scattering between 0.8 and 1.6 kHz for S waves, while the same happens for P waves between 1.8 and 2.6 kHz. The slope of the dispersion branches in the long-wavelength limit (the wavenumber κ tending to zero) can be used to compute the wave-propagation speed in the crystal,

$$c = \frac{2\pi f}{\kappa}$$

and the quasistatic homogenized material properties as

$$\rho_{\mathrm{hom}} = \frac{\int \rho dV}{\int dV}$$

$$C_{66\mathrm{hom}} = c_{\mathrm{S}}^2 \rho_{\mathrm{hom}}$$

$$C_{11\mathrm{hom}} = c_{\mathrm{P}}^2 \rho_{\mathrm{hom}}$$

where the integrals for computing the homogenized density are performed on the whole volume of the unit cell considering the density inside the holes to be zero, and c_{S} and c_{P} are the computed speeds for S and P waves, respectively.

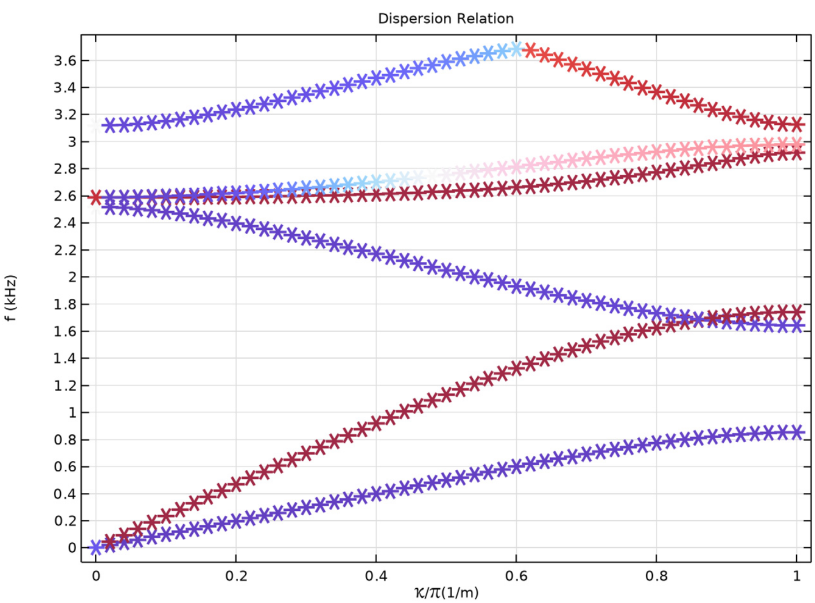

Figure 3: Dispersion relation.

Figure 4 shows the computed transmission coefficients for incident S and P waves. It illustrates how the crystal works as a mirror for elastic S and P waves in the frequency ranges corresponding to their respective band gaps.

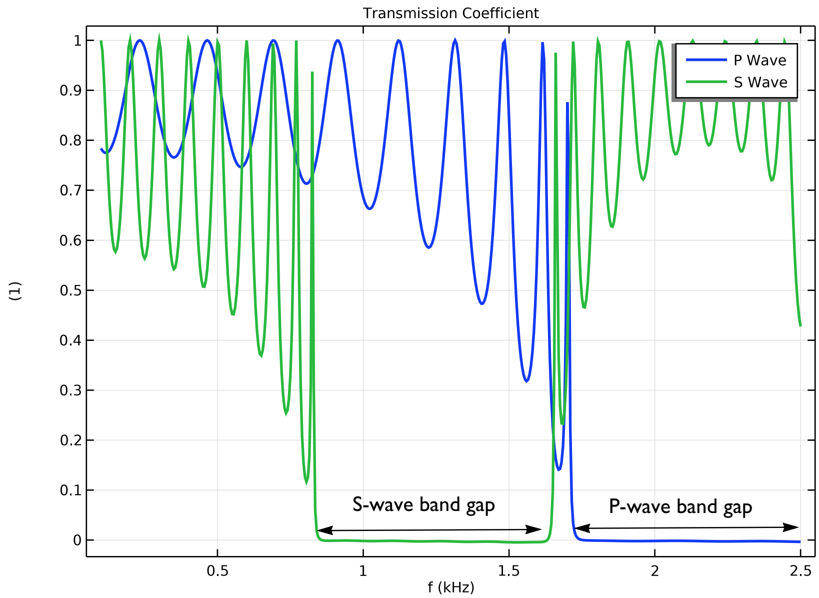

Figure 4: Transmission coefficients for S and P waves as functions of the frequency of the impinging wave.

[Figure 5](#) shows the divergence of the total displacement when a plane P wave is incident from the left on the crystal at a frequency of 2.5 kHz. This illustrates that P waves cannot penetrate into the crystal, which behaves as a mirror.

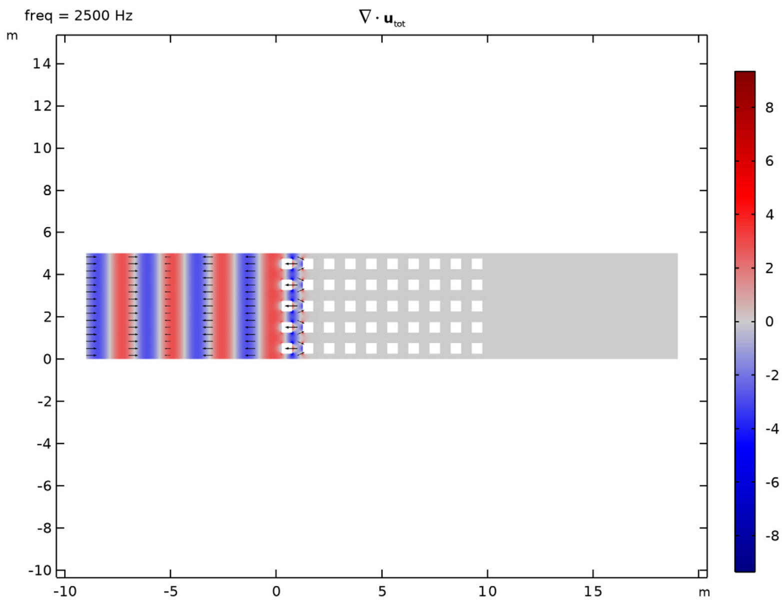

Figure 5: Total P-wave field obtained when a P wave is incident on the crystal at 2.5 kHz.

Notes About the COMSOL Implementation

- You can add a **Periodic Condition** boundary condition to the unit cell to simulate an infinite lattice at the computational cost of one cell only. In a similar fashion you can use a **Periodic Condition** boundary condition even when the crystal is finite in one direction, if you want to consider it infinite in the other directions.
- When studying the reflection and transmission properties of the finite crystal, you can turn the scattering problem into a radiation problem. On the surfaces of the holes in each unit cell, the free boundary condition applies to the total field:

$$\sigma_t \cdot \mathbf{n} = (\sigma_b + \sigma_s) \cdot \mathbf{n} = 0$$

where the total field is seen as the sum of the scattered and background fields. You can instead add a **Boundary Load** according to

$$\sigma_s \cdot \mathbf{n} = -(\sigma_b \cdot \mathbf{n})$$

and solve for the scattered field only.

- A **Perfectly Matched Layer** is added to each side of the assembly along the wave propagation direction to truncate the computational domain.

Application Library path: `Acoustics_Module/Elastic_Waves/phononic_crystal`

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**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

PART 1

In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts** > **2D Part**.

Square 1 (sq1)

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

2 In the **Settings** window for **Square**, click to expand the **Layers** section.

3 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.25

4 Select the **Layers to the left** checkbox.

5 Select the **Layers to the right** checkbox.

6 Select the **Layers on top** checkbox.

7 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Part 1** and choose **Delete Entities**.

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

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

4 On the object **sq1**, select Domain 5 only.

5 In the **Geometry** toolbar, click **Build All**.

GEOMETRY 1

Part Instance 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Part 1**.

2 In the **Settings** window for **Part Instance**, click **Build All Objects**.

GLOBAL DEFINITIONS

Aluminum

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

2 In the **Settings** window for **Material**, type `Aluminum` in the **Label** text field.

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Basic Properties** > **Density**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties** > **Poisson's Ratio**.

6 Click **Add to Material**.

7 In the **Material properties** tree, select **Basic Properties** > **Young's Modulus**.

8 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Density	rho	`2700[kg/m^3]`	kg/m³	Basic

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.3	1	Basic
Young's modulus	E	69.9[GPa]	Pa	Basic

MATERIALS

Material Link 1 (matlnk1)

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

GLOBAL DEFINITIONS

Parameters 1

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

2 In the table, enter the following settings:

Name	Expression	Value	Description
P	1	1	Sweep parameter

SOLID MECHANICS (SOLID)

Periodic Condition 1

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

2 Select Boundaries 1, 3, 5, and 22–24 only.

3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 Specify the $\mathbf{k}_F$ vector as

P*pi	X

Periodic Condition 2

1 Right-click **Periodic Condition 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 2, 7, 9, 14, 16, and 21 only.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

MESH 1

Mapped 1

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

Size

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

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

3 From the **Predefined** list, choose **Coarse**.

INFINITE CRYSTAL

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

2 In the **Settings** window for **Study**, type `Infinite Crystal` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Infinite Crystal** click **Step 1: Eigenfrequency**.

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

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
P (Sweep parameter)	`range(0,0.02,1)`	

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

RESULTS

Array 2D 1

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

2 Right-click **Results** > **Datasets** and choose **More 2D Datasets** > **Array 2D**.

3 In the **Settings** window for **Array 2D**, locate the **Array Size** section.

4 In the **X size** text field, type 20.

5 In the **Y size** text field, type 20.

6 Click to expand the **Advanced** section. Select the **Floquet–Bloch periodicity** checkbox.

7 Find the **Wave vector** subsection. In the **X** text field, type P*pi.

Mode Shape (solid)

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

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

3 From the **Dataset** list, choose **Array 2D 1**.

4 From the **Parameter value (P)** list, choose **0.1**.

5 From the **Eigenfrequency (Hz)** list, choose **98.541**.

6 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

7 Find the **Solution** subsection. Clear the **Solution** checkbox.

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Solution Array 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

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

4 In the **Parameter values (P)** list box, select **0.1**.

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

6 In the **Eigenfrequency indices (1-6)** text field, type 1 2.

7 In the **Mode Shape (solid)** toolbar, click **Plot**.

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

Dispersion Relation

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

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** checkbox. In the associated text field, type \kappa/\pi (1/m).

5 Select the **y-axis label** checkbox. In the associated text field, type f (kHz).

Global 1

1 Right-click **Dispersion Relation** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.freq`	`kHz`	`Frequency`

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

5 In the **Expression** text field, type `P`.

6 In the **Dispersion Relation** toolbar, click **Plot**.

Dispersion Relation

1 In the **Model Builder** window, click **Dispersion Relation**.

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

3 Clear the **Show legends** checkbox.

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

Global 1

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

2 In the **Settings** window for **Global**, click to expand the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

5 From the **Width** list, choose **2**.

Color Expression 1

1 Right-click **Global 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `intop1(real(u)^2)/ intop1(real(solid.disp)^2)`.

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

5 In the **Dispersion Relation** toolbar, click **Plot**.

Long Wavelength Homogenized Properties

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Long Wavelength Homogenized Properties` in the **Label** text field.

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

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

S Waves

1 Right-click **Long Wavelength Homogenized Properties** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `S Waves` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Infinite Crystal/Solution 1 (sol1)**.

4 From the **Parameter selection (P)** list, choose **From list**.

5 In the **Parameter values (P)** list box, select **0.02**.

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

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

Expression	Unit	Description
2*pi*solid.freq/(P*pi/1[m])	m/s	Wave Speed, S
(2*pi*solid.freq/(P*pi/1[m]))^2*2700[kg/m^2]*0.75	N/m	Homogenized C66

P Waves

1 Right-click **S Waves** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `P Waves` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency selection** list, choose **Manual**.

4 In the **Eigenfrequency indices (1-6)** text field, type `2`.

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

Expression	Unit	Description
2*pi*solid.freq/(P*pi/1[m])	m/s	Wave Speed, P
(2*pi*solid.freq/(P*pi/1[m]))^2*2700[kg/m^2]*0.75	N/m	Homogenized C11
2700[kg/m^3]*0.75	kg/m^3	Homogenized density

6 In the **Long Wavelength Homogenized Properties** toolbar, click **= Evaluate**.

ADD COMPONENT

Right-click **P Waves** and choose **2D**.

UNIT CELL

In the **Settings** window for **Component**, type `Unit Cell` in the **Label** text field.

FINITE CRYSTAL

1 In the **Model Builder** window, click **Component 2 (comp2)**.

2 In the **Settings** window for **Component**, type `Finite Crystal` in the **Label** text field.

GEOMETRY 2

Part Instance 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Part 1**.

2 In the **Settings** window for **Part Instance**, click **Build Selected**.

ADD PHYSICS

1 In the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Recently Used** > **Solid Mechanics (solid)**.

4 Click the **Add to Finite Crystal** button in the window toolbar.

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

DEFINITIONS (COMP2)

Incident P Wave

1 In the **Model Builder** window, under **Finite Crystal (comp2)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Incident P Wave` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
Omega	2*pi*freq	Hz	Angular frequency
kP	Omega/solid2.cp	1/m	P Wave wavenumber
uP	exp(-1i*kP*x+1i*phase)[m]	m	P Wave: u field
vP	0[m]	m	P Wave: v field
eps11P	d(uP, x)		Incident P wave: strain tensor, 11 component
eps22P	d(vP, y)		Incident P wave: strain tensor, 22 component
eps12P	0.5*(d(uP,y)+d(vP,x))		Incident P wave: strain tensor, 12 component
s11P	(solid2.lambLame+2*solid2.muLame)*eps11P+solid2.lambLame*eps22P	N/m²	Incident P wave: stress tensor, 11 component

Name	Expression	Unit	Description
s22P	solid2.lambLame*eps11P+(solid2.lambLame+2*solid2.muLame)*eps22P	N/m²	Incident P wave: stress tensor, 22 component
s12P	2*solid2.muLame*eps12P	N/m²	Incident P wave: stress tensor, 12 component
I1P	0.5*real(-s11P*conj(1i*Omega*uP)-s12P*conj(1i*Omega*vP))	W/m²	Incident P wave: mechanical energy flux, 1 component
I2P	0.5*real(-s12P*conj(1i*Omega*uP)-s22P*conj(1i*Omega*vP))	W/m²	Incident P wave: mechanical energy flux, 2 component

SOLID MECHANICS 2 (SOLID2)

Periodic Condition 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.
2. Select Boundaries 2, 7, 9, 14, 16, and 21 only.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundaries 10, 11, 13, and 17 only.
3. In the **Settings** window for **Boundary Load**, locate the **Force** section.
4. Specify the $\mathbf{f}_A$ vector as

-(s11P*solid2.nx+s12P*solid2.ny)	x
-(s12P*solid2.nx+s22P*solid2.ny)	y

GEOMETRY 2

Array 1 (arr1)

1. In the **Geometry** toolbar, click **Transforms** and choose **Array**.
2. In the **Settings** window for **Array**, locate the **Size** section.
3. In the **x size** text field, type 10.
4. Locate the **Displacement** section. In the **x** text field, type 1.
5. Click the **Select All** button in the **Graphics** toolbar.
6. Click **Build Selected**.
7. Click the **Zoom Extents** button in the **Graphics** toolbar.

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 `10`.

4 Locate the **Position** section. In the **x** text field, type `-10`.

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

Layer name	Thickness (m)
`Layer 1`	`1`

6 Select the **Layers to the left** checkbox.

7 Clear the **Layers on bottom** checkbox.

8 Click **Build Selected**.

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

Rectangle 2 (r2)

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

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **x** text field, type `10`.

4 Locate the **Layers** section. Clear the **Layers to the left** checkbox.

5 Select the **Layers to the right** checkbox.

6 Click **Build Selected**.

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

DEFINITIONS (COMP2)

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 84 only.

Scattered Intensity

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

4 Select Boundaries 7, 9, and 11 only.

5 In the **Label** text field, type `Scattered Intensity`.

MATERIALS

Material Link 2 (matlnk2)

In the **Model Builder** window, under **Finite Crystal (comp2)** right-click **Materials** and choose **More Materials** > **Material Link**.

SOLID MECHANICS 2 (SOLID2)

Periodic Condition 1

Select Boundaries 2, 3, 5, 6, 8, 13, 15, 20, 22, 27, 29, 34, 36, 41, 43, 48, 50, 55, 57, 62, 64, 69, 71, 76, 78, 83, 85, 90, 92, 97, 99, 104, 106, 111, 113, 118, 120, 125, 127, 132, 134, 139, 141, 146, 148, 153, 155, 160, 162, 167, 169, 174, 176, 181, 183, 188, 190, 195, 197, 202, 204, 209, 211, 216, 218, 221, 223, and 224 only.

MESH 2

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 84 only.

Free Triangular 1

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

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Extremely fine**.
4 Click to expand the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.1`.
5 Click **Build All**.

ADD STUDY

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.
4 Click the **Add Study** button in the window toolbar.

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

INFINITE CRYSTAL

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 In the tree, select **Finite Crystal (comp2)** > **Solid Mechanics 2 (solid2)**.

4 Click **Disable in Model**.

INFINITE CRYSTAL

In the **Model Builder** window, collapse the **Infinite Crystal** node.

FINITE CRYSTAL: P WAVE

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Finite Crystal: P Wave` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Finite Crystal: P Wave** 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 `range(100, 5, 2500 )`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Unit Cell (comp1)** > **Solid Mechanics (solid)**.

6 Click **Disable in Model**.

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

RESULTS

Array 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Array 2D**.

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

3 In the **Y size** text field, type `5`.

4 Locate the **Data** section. From the **Dataset** list, choose **Finite Crystal: P Wave/ Solution 2 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Click the **Select All** button in the **Graphics** toolbar.

5 Select Domains 2–83 only.

P Wave Incident: P Wave Scattered

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `P Wave Incident: P Wave Scattered` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Array 2D 2**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `\$ \nabla \cdot \mathbf{u}_\mathrm{tot} \$`.

6 In the **Parameter indicator** text field, type `freq = eval(freq) Hz`.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **P Wave Incident: P Wave Scattered** and choose **Surface**.

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

3 In the **Expression** text field, type `d(u2+uP,x)+d(v2+vP,y)`.

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

5 From the **Scale** list, choose **Linear symmetric**.

6 In the **P Wave Incident: P Wave Scattered** toolbar, click **Plot**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **P Wave Incident: P Wave Scattered** and choose **Arrow Surface**.

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

3 In the **x-component** text field, type `u2+uP`.

4 In the **y-component** text field, type `v2+vP`.

5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

6 From the **Color** list, choose **Black**.

P Wave Incident: P Wave Scattered

1 In the **Model Builder** window, click **P Wave Incident: P Wave Scattered**.

2 In the **P Wave Incident: P Wave Scattered** toolbar, click **Plot**.

P Wave Incident: S Wave Scattered

1 Right-click **P Wave Incident: P Wave Scattered** and choose **Duplicate**.

2 In the **Model Builder** window, click **P Wave Incident: P Wave Scattered 1**.

3 In the **Settings** window for **2D Plot Group**, type `P Wave Incident: S Wave Scattered` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `\$ \nabla \times \mathbf{u}_\mathrm{tot} \$`.

Surface 1

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

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

3 In the **Expression** text field, type `-d(u2+uP,y)+d(v2+vP,x)`.

4 In the **P Wave Incident: S Wave Scattered** toolbar, click **Plot**.

Transmission Coefficient

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Finite Crystal: P Wave/ Solution 2 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `f (kHz)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `(1)`.

P Wave

1 Right-click **Transmission Coefficient** and choose **Global**.

2 In the **Settings** window for **Global**, type `P Wave` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`1-comp2.aveop1(solid2.IX*solid2.nx/I1P)`	1	`Transmission Coefficient`

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **kHz**.

5 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

6 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Solution** checkbox.

7 Clear the **Description** checkbox.

8 Select the **Label** checkbox.

9 In the **Transmission Coefficient** toolbar, click **Plot**.

FINITE CRYSTAL: P WAVE

In the **Model Builder** window, collapse the **Finite Crystal: P Wave** node.

ADD STUDY

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

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

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

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

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

FINITE CRYSTAL: S WAVE

1 In the **Settings** window for **Study**, type `Finite Crystal: S Wave` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Finite Crystal: S Wave** 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 `range(100, 5, 2500)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Unit Cell (comp1)** > **Solid Mechanics (solid)**.

6 Click **Disable in Model**.

DEFINITIONS (COMP2)

Incident S Wave

1 In the **Model Builder** window, under **Finite Crystal (comp2)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
kS	Omega/solid2.cs	1/m	S Wave wavenumber
uS	0[m]	m	S Wave: u field
vS	exp(-1i*kS*x+1i*phase)[m]	m	S Wave: v field
eps11S	d(uS, x)		Incident S wave: strain tensor, 11 component
eps22S	d(vS, y)		Incident S wave: strain tensor, 22 component
eps12S	0.5*(d(uS,y)+d(vS,x))		Incident S wave: strain tensor, 12 component
s11S	(solid2.lambLame+2*solid2.muLame)*eps11S+solid2.lambLame*eps22S	N/m²	Incident S wave: stress tensor, 11 component
s22S	solid2.lambLame*eps11S+(solid2.lambLame+2*solid2.muLame)*eps22S	N/m²	Incident S wave: stress tensor, 22 component
s12S	2*solid2.muLame*eps12S	N/m²	Incident S wave: stress tensor, 12 component
I1S	0.5*real(-s11S*conj(1i*Omega*uS)-s12S*conj(1i*Omega*vS))	W/m²	Incident S wave: mechanical energy flux, 1 component
I2S	0.5*real(-s12S*conj(1i*Omega*uS)-s22S*conj(1i*Omega*vS))	W/m²	Incident S wave: mechanical energy flux, 2 component

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Finite Crystal (comp2)** click **Solid Mechanics 2 (solid2)**.

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Typical Wave Speed for Perfectly Matched Layers** section.

3 In the c_{ref} text field, type `solid2.cs`.

Boundary Load: P Wave

1 In the **Model Builder** window, under **Finite Crystal (comp2)** > **Solid Mechanics 2 (solid2)** click **Boundary Load 1**.

2 In the **Settings** window for **Boundary Load**, type `Boundary Load: P Wave` in the **Label** text field.

Boundary Load: S Wave

1 Right-click **Boundary Load: P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Load**, type `Boundary Load: S Wave` in the **Label** text field.

3 Locate the **Force** section. Specify the $\mathbf{f}_A$ vector as

`-(s11S*solid2.nx+s12S*solid2.ny)`	x
`-(s12S*solid2.nx+s22S*solid2.ny)`	y

FINITE CRYSTAL: P WAVE

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Finite Crystal: P Wave** node, then click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Finite Crystal (comp2)** > **Solid Mechanics 2 (solid2)** > **Boundary Load: S Wave**.

4 Click **Disable**.

FINITE CRYSTAL: S WAVE

1 In the **Model Builder** window, under **Finite Crystal: S Wave** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Finite Crystal (comp2)** > **Solid Mechanics 2 (solid2)** > **Boundary Load: P Wave**.

4 Click **Disable**.

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

RESULTS

Array 2D 3

1. In the **Model Builder** window, under **Results** > **Datasets** right-click **Array 2D 2** and choose **Duplicate**.
2. In the **Settings** window for **Array 2D**, locate the **Data** section.
3. From the **Dataset** list, choose **Finite Crystal: S Wave/Solution 3 (sol3)**.

S Wave Incident: P Wave Scattered

1. In the **Model Builder** window, right-click **P Wave Incident: P Wave Scattered** and choose **Duplicate**.
2. In the **Settings** window for **2D Plot Group**, type `S Wave Incident: P Wave Scattered` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Array 2D 3**.
4. From the **Parameter value (freq (Hz))** list, choose **1200**.

Surface 1

1. In the **Model Builder** window, expand the **S Wave Incident: P Wave Scattered** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `d(u2+uS,x)+d(v2+vS,y)`.

Arrow Surface 1

1. In the **Model Builder** window, click **Arrow Surface 1**.
2. In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
3. In the **x-component** text field, type `u2+uS`.
4. In the **y-component** text field, type `v2+vS`.
5. In the **S Wave Incident: P Wave Scattered** toolbar, click **Plot**.

S Wave Incident: S Wave Scattered

1. In the **Model Builder** window, right-click **P Wave Incident: S Wave Scattered** and choose **Duplicate**.
2. In the **Settings** window for **2D Plot Group**, type `S Wave Incident: S Wave Scattered` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Array 2D 3**.
4. From the **Parameter value (freq (Hz))** list, choose **1200**.

Surface 1

1 In the **Model Builder** window, expand the **S Wave Incident: S Wave Scattered** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `-d(u2+uS,y)+d(v2+vS,x)`.

Arrow Surface 1

1 In the **Model Builder** window, click **Arrow Surface 1**.

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

3 In the **x-component** text field, type `u2+uS`.

4 In the **y-component** text field, type `v2+vS`.

5 In the **S Wave Incident: S Wave Scattered** toolbar, click **Plot**.

S Wave

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

2 In the **Settings** window for **Global**, type `S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Finite Crystal: S Wave/ Solution 3 (sol3)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`1-comp2.aveop1(solid2.IX*solid2.nx/I1S)`	`1`	`Transmission Coefficient`

5 In the **Transmission Coefficient** toolbar, click **Plot**.

P Wave Incident: P Wave Scattered

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