Created in COMSOL Multiphysics 6.2

Transverse Isotropic Porous Layer

Introduction

This tutorial investigates the acoustic properties of a porous layer made of glass wool. The porous material has transverse isotropic properties and is modeled with the full anisotropic poroelastic material model.

Model Definition

This model analyses the acoustic properties of a two-dimensional porous layer (see Figure 1); specifically, the surface impedance and absorption of the layer. The layer consists of glass wool which has transverse isotropic porous properties. The properties are thus anisotropic and are here modeled with the Anisotropic Poroelastic Material feature of the Poroelastic Waves interface. The results are compared with the isotropic case, as well as the experiential and transfer matrix based results of the same setup reported in P. Khurana and others (Ref. 1).

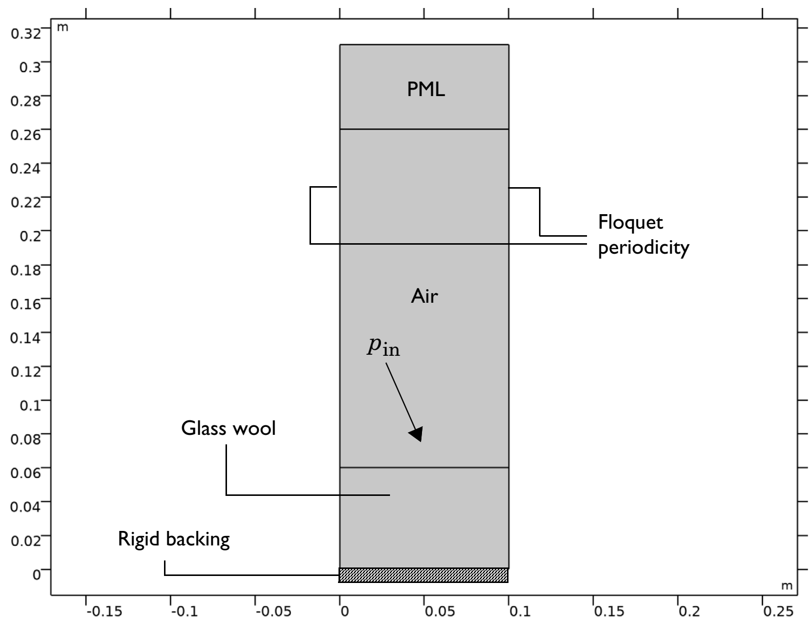


Figure 1: Model geometry and setup.

The model consists of an infinite rigidly backed porous sample of thickness 6 cm. The infinite characteristic of the model are included by using Floquet periodic conditions, following the same procedure as in the Porous Absorber tutorial. The material properties used in the poroelastic material model are reported in Table 1 of Ref. 1. A plane wave is incident on the porous layer at angles varying from 0° (normal incidence) to 85°.

Results and Discussion

The displacement in the porous layer, the total acoustic pressure, and the sound pressure level is depicted, for an angle of incidence of 45° at 3000 Hz, in Figure 2.

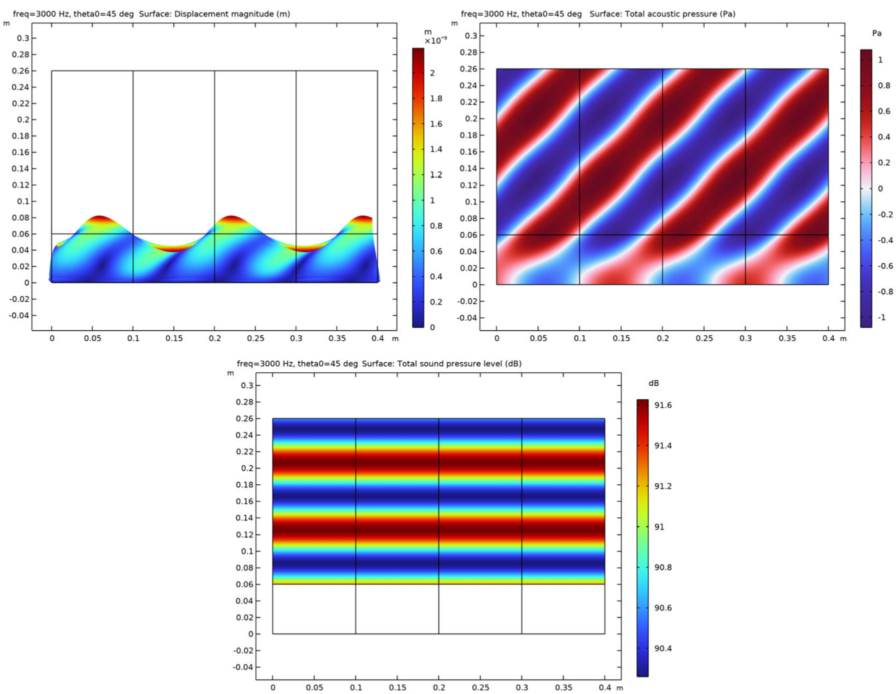


Figure 2: Displacement, pressure, and sound pressure level in the porous layer.

The real and imaginary part of the surface impedance of the layer is depicted in Figure 3, Figure 4, Figure 5, and Figure 6; for the frequencies of 500 Hz, 700 Hz, 1000 Hz, and 3000 Hz, respectively. Both the anisotropic and the isotropic results are depicted. This corresponds the results reported in Figure 2 in Ref. 1. Finally, the surface absorption of the porous layer is depicted as function of the angle of incidence in Figure 7.

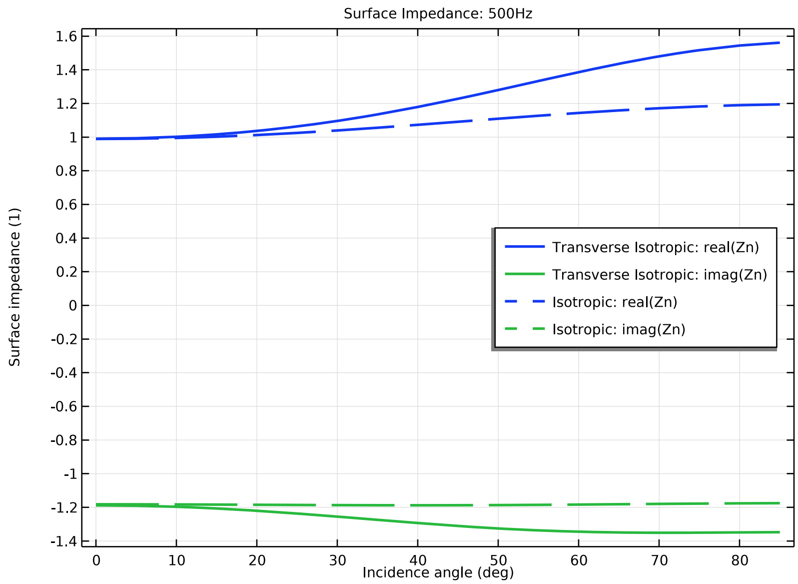


Figure 3: Real and imaginary part of the surface impedance at 500 Hz.

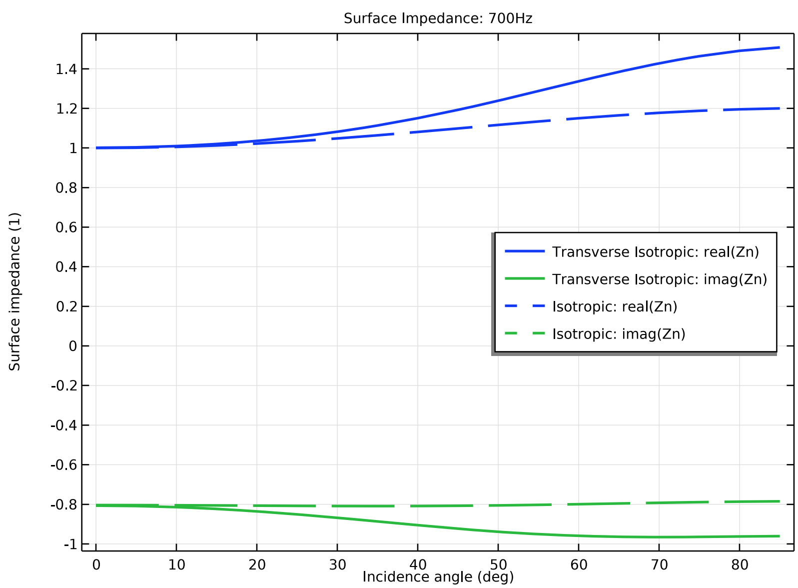


Figure 4: Real and imaginary part of the surface impedance at 700 Hz.

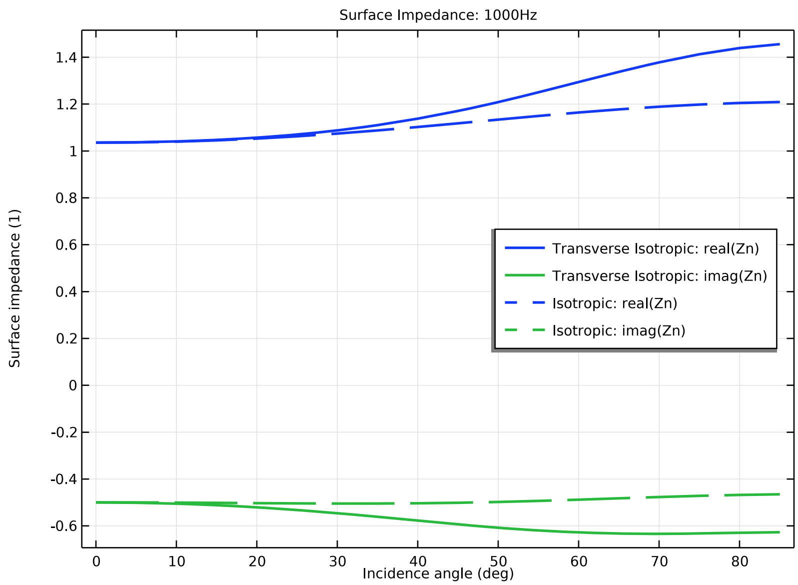


Figure 5: Real and imaginary part of the surface impedance at 1000 Hz.

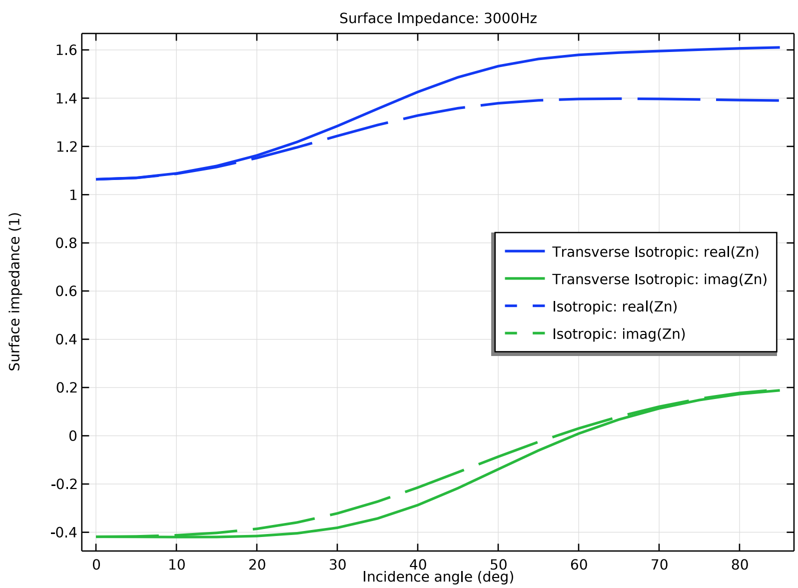


Figure 6: Real and imaginary part of the surface impedance at 3000 Hz.

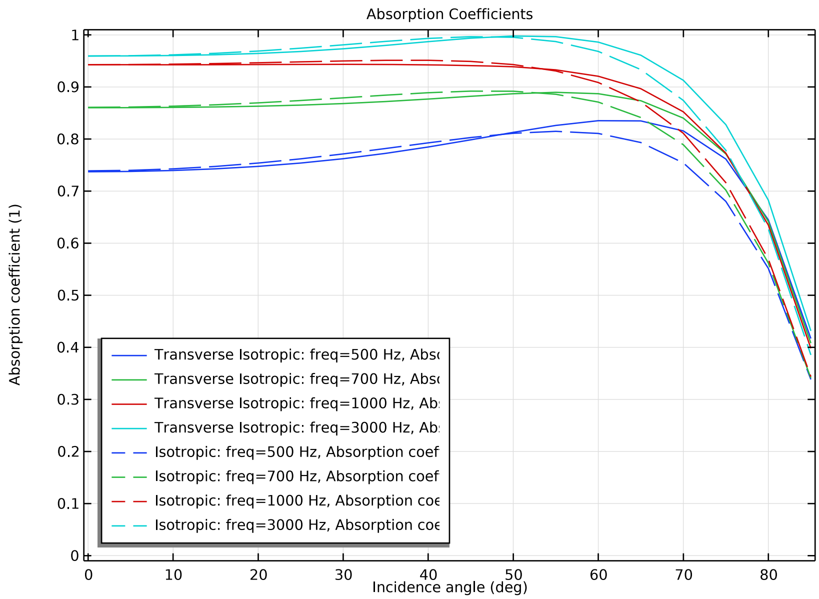


Figure 7: Surface absorption coefficient as function of angle of incidence.

Reference

1. P. Khurana, L. Boeckx, W. Lauriks, P. Leclaire, O. Dazel, and J. F. Allard, "A description of transversely isotropic sound absorbing porous materials by transfer matrices," *J. Acoust. Soc. Am.*, vol. 125, no. 2, pp. 915–921, 2009.

Application Library path: `Acoustics_Module/Building_and_Room_Acoustics/transverse_isotropic_porous_layer`

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 **Acoustics>Elastic Waves>Poroelastic Waves (pelw)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Frequency Domain**.
8 Click **Done**.

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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `transverse_isotropic_porous_layer_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

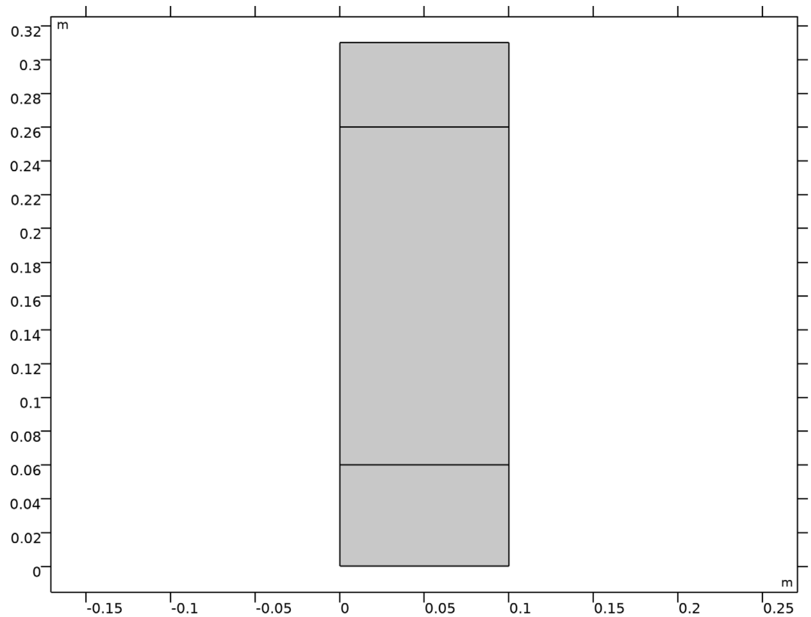
1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `W`.
4 In the **Height** text field, type `H+Hair+Hpml`.
5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	H
Layer 2	Hair

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.



DEFINITIONS

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

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

4 Click the **Load** button. From the menu, choose **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `transverse_isotropic_porous_layer_variables.txt`.

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, type `intop_pnt` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Point 3 only.

Average 1 (aveop1)

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

2 In the **Settings** window for **Average**, type aveop_bnd in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

Integration 2 (intop2)

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

2 In the **Settings** window for **Integration**, type intop_bnd in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

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 **Scaling** section.

4 In the **PML scaling factor** text field, type 1/cos(theta0).

5 In the **PML scaling curvature parameter** text field, type 3.

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

POROELASTIC WAVES (PELW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Poroelastic Waves (pelw)**.

2 Select Domain 1 only.

Poroelastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Poroelastic Waves (pelw)** click **Poroelastic Material 1**.

2 In the **Settings** window for **Poroelastic Material**, locate the **Porous Matrix Properties** section.

3 From the G_d list, choose **User defined**. In the associated text field, type (50+7i)[kPa].

4 From the ν_d list, choose **User defined**. In the associated text field, type `0.1`.

5 From the ρ_d list, choose **User defined**. In the associated text field, type `60[kg/m^3]`.

6 From the η_s list, choose **User defined**. From the ε_p list, choose **User defined**. In the associated text field, type `0.99`.

7 From the R_f list, choose **User defined**. In the associated text field, type `17000[N*s/m^4]`.

8 From the τ_∞ list, choose **User defined**. In the associated text field, type `1.01`.

9 From the L_v list, choose **User defined**. In the associated text field, type `140[um]`.

10 From the L_{th} list, choose **User defined**. In the associated text field, type `150[um]`.

Anisotropic Poroelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Anisotropic Poroelastic Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Anisotropic Poroelastic Material**, locate the **Porous Matrix Properties** section.

4 From the **Porous model** list, choose **Drained matrix, orthotropic**.

5 From the **E** list, choose **User defined**. Specify the **associated** vector as

10	X
10	Y

6 From the **G** list, choose **User defined**. Specify the **associated** vector as

(50+7i)[kPa]	YZ
(120+22i)[kPa]	XZ

7 From the ν list, choose **User defined**. Specify the **associated** vector as

0.1	XY
0.1	YZ

8 From the ρ_d list, choose **User defined**. In the associated text field, type `60[kg/m^3]`.

9 From the η_s list, choose **User defined**. From the ε_p list, choose **User defined**. In the associated text field, type `0.99`.

10 From the $[R_f]_{ij}$ list, choose **User defined**. From the list, choose **Diagonal**.

11 In the $[R_{\mathrm{f}}]_{ij}$ table, enter the following settings:

5000[N*s/m^4]	0
0	17000[N*s/m^4]

12 From the $[\tau_{\infty}]_{ij}$ list, choose **User defined**. In the associated text field, type 1.01.

13 From the $[L_{\mathrm{v}}]_{ij}$ list, choose **User defined**. From the list, choose **Diagonal**.

14 In the $[L_{\mathrm{v}}]_{ij}$ table, enter the following settings:

126[um]	0
0	140[um]

15 From the L_{th} list, choose **User defined**. In the associated text field, type 150[um].

Fixed Constraint 1

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

2 Select Boundary 2 only.

Impervious Layer 2

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

2 Select Boundary 2 only.

Periodic Condition 1

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

2 Select Boundaries 1 and 8 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 $\mathbf{k}_{\mathrm{F}}$ vector as

kx	X
ky	Y

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.

Background Pressure Field 1

1 In the **Physics** toolbar, click **Domains** and choose **Background Pressure Field**.

2 In the **Model Builder** window, expand the **Pressure Acoustics, Frequency Domain (acpr)** node, then click **Background Pressure Field 1**.

3 Select Domain 2 only.

4 In the **Settings** window for **Background Pressure Field**, locate the **Background Pressure Field** section.

5 In the p_0 text field, type 1.

6 From the c list, choose **From material**.

7 Specify the $\mathbf{e}_k$ vector as

kx_e	x
ky_e	y

8 Select the **Calculate background and scattered field intensity** check box.

9 From the ρ list, choose **From material**.

Periodic Condition 1

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

2 Select Boundaries 3 and 9 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

kx	x
ky	y

Periodic Condition 2

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

2 Select Boundaries 5 and 10 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

kx	x
ky	y

MULTIPHYSICS

Acoustic–Porous Boundary 1 (apb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic–Porous Boundary**.

   Proceed to set up the Multiphysics Coupling that couples the **Pressure Acoustics, Frequency Domain (acpr)** and the **Poroelastic Waves (pelw)**.

2 Select Boundary 4 only.

MESH 1

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh component.

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 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `H/12`.

5 In the **Minimum element size** text field, type `H/12`.

6 Click **Build All**.

STUDY 1 - TRANSVERSE ISOTROPIC

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

2 In the **Settings** window for **Study**, type `Study 1 - Transverse Isotropic` in the **Label** text field.

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

   Turn off the generation of default plots for each study. If turned on, all the default plots for each physics interface will be generated.

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1 - Transverse Isotropic** node, then 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 `500 700 1000 3000`.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Angle of incidence)	`range(0,5,85)`	`deg`

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

ADD STUDY

1 In the **Home** 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 **Add Study** in the window toolbar.

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

STUDY 2 - ISOTROPIC

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

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

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

1 In the **Model Builder** window, under **Study 2 - Isotropic** 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 `500 700 1000 3000`.

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

5 In the tree, select **Component 1 (comp1)>Poroelastic Waves (pelw)> Anisotropic Poroelastic Material 1**.

6 Right-click and choose **Disable**.

7 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Angle of incidence)	`range(0,5,85)`	`deg`

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

RESULTS

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

First, some extra datasets need to be defined to have a better overview of the 2D results: Displacement, Acoustic Pressure and Sound Pressure Level.

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 `4`.
5 Click to expand the **Advanced** section. Select the **Floquet-Bloch periodicity** check box.
6 Find the **Wave vector** subsection. In the **X** text field, type `kx`.
7 In the **Y** text field, type `ky`.

Selection

1 Right-click **Array 2D 1** 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 Select Domains 1 and 2 only.

Array 2D 1

In the **Model Builder** window, right-click **Array 2D 1** and choose **Duplicate**.

Array 2D 2

1 In the **Model Builder** window, click **Array 2D 2**.
2 In the **Settings** window for **Array 2D**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2 - Isotropic/Solution 2 (sol2)**.

Displacement (pelw)

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Displacement (pelw)` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Array 2D 1**.
4 From the **Parameter value (freq (Hz))** list, choose **3000**.
5 From the **Parameter value (theta0 (deg))** list, choose **45**.

6 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

In the **Displacement (pelw)** toolbar, click **Surface**.

Deformation 1

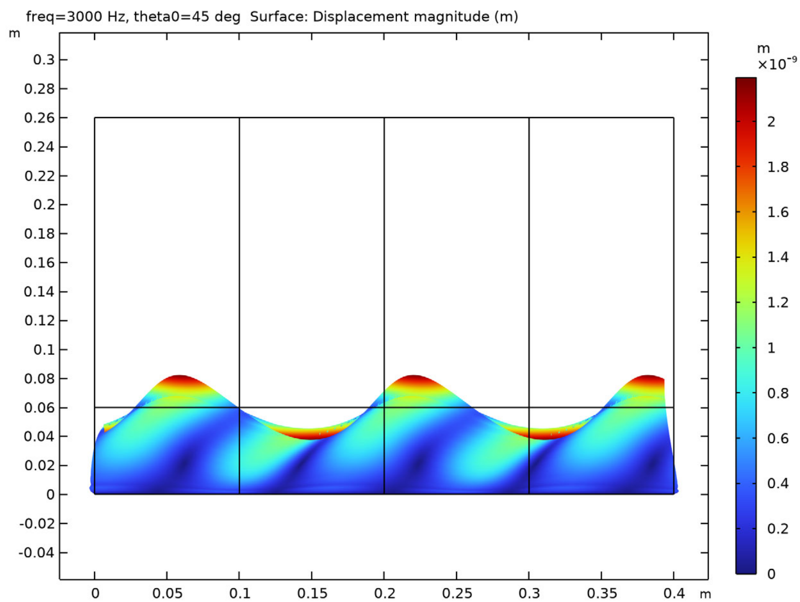
1 In the **Displacement (pelw)** toolbar, click **Deformation**.

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

3 In the **x-component** text field, type `u`.

4 In the **y-component** text field, type `v`.

5 In the **Displacement (pelw)** toolbar, click **Plot**.



Acoustic Pressure (acpr)

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Acoustic Pressure (acpr)` in the **Label** text field.

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

4 From the **Parameter value (freq (Hz))** list, choose **3000**.

5 From the **Parameter value (theta0 (deg))** list, choose **45**.

6 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 In the **Acoustic Pressure (acpr)** toolbar, click **Surface**.

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

3 In the **Expression** text field, type `apb1.p_t`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

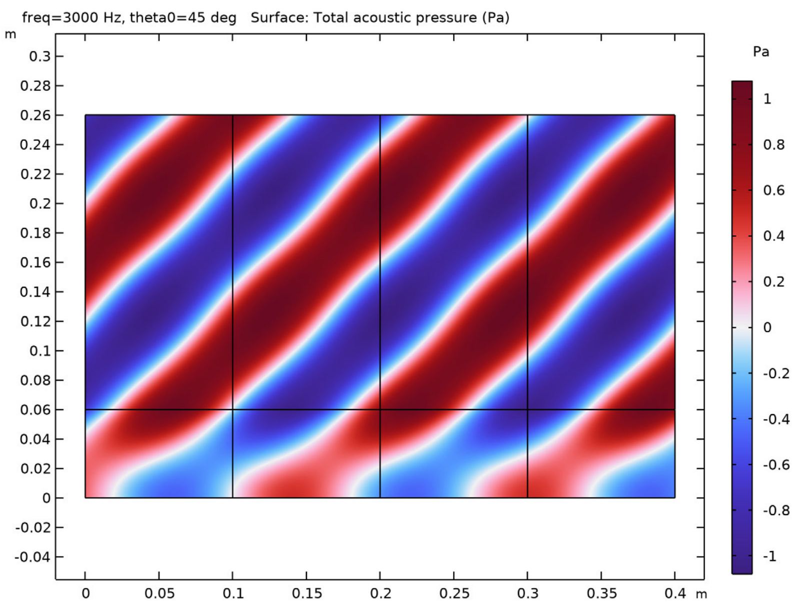
5 In the **Color Table** dialog box, select **Wave>Wave** in the tree.

6 Click **OK**.

7 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

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

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



Sound Pressure Level (acpr)

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Sound Pressure Level (acpr)` in the **Label** text field.

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

4 From the **Parameter value (freq (Hz))** list, choose **3000**.

5 From the **Parameter value (theta0 (deg))** list, choose **45**.

6 Locate the **Color Legend** section. Select the **Show units** check box.

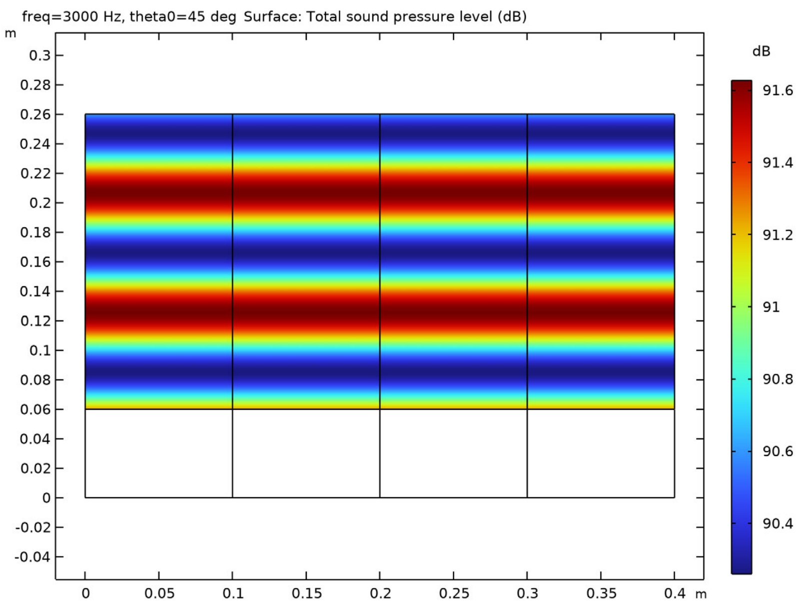
Surface 1

1 In the **Sound Pressure Level (acpr)** toolbar, click **Surface**.

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

3 In the **Expression** text field, type `acpr.Lp_t`.

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



Surface Impedance: 500Hz

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

2 In the **Settings** window for **1D Plot Group**, type `Surface Impedance: 500Hz` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.

4 In the **Parameter values (freq (Hz))** list, select **500**.

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

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `Incidence angle (deg)`.

8 Select the **y-axis label** check box. In the associated text field, type `Surface impedance (1)`.

9 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Global 1

1 In the **Surface Impedance: 500Hz** toolbar, click **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
`real(-Zn)`	1	`real(Zn)`
`imag(-Zn)`	1	`imag(Zn)`

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

5 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

6 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Transverse Isotropic:` .

Surface Impedance: 500Hz

In the **Surface Impedance: 500Hz** toolbar, click **Global**.

Global 2

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

2 From the **Dataset** list, choose **Study 2 - Isotropic/Solution 2 (sol2)**.

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

4 In the **Parameter values (freq (Hz))** list, select **500**.

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

Expression	Unit	Description
`real(-Zn)`	1	`real(Zn)`
`imag(-Zn)`	1	`imag(Zn)`

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 From the **Color** list, choose **Cycle (reset)**.

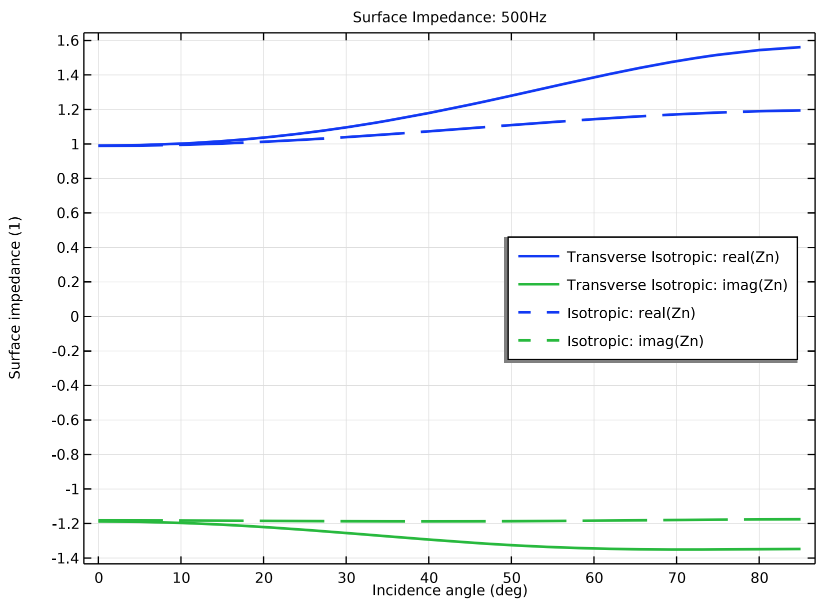
8 From the **Width** list, choose **2**.

9 Locate the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

10 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Isotropic: `.

11 In the **Surface Impedance: 500Hz** toolbar, click **Plot**.

The surface impedance for both studies, isotropic and transverse isotropic, for 500 Hz should look like the following figure:



Surface Impedance: 500Hz

In the **Model Builder** window, right-click **Surface Impedance: 500Hz** and choose **Duplicate**.

Surface Impedance: 700Hz

1 In the **Model Builder** window, under **Results** click **Surface Impedance: 500Hz 1**.

2 In the **Settings** window for **1D Plot Group**, type `Surface Impedance: 700Hz` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (freq (Hz))** list, select **700**.

Global 2

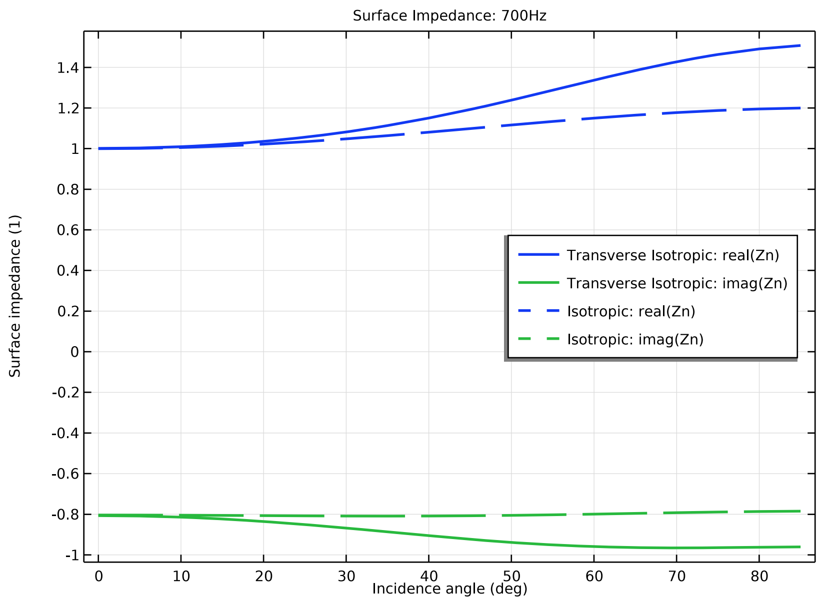
1 In the **Model Builder** window, expand the **Surface Impedance: 700Hz** node, then click **Global 2**.

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

3 In the **Parameter values (freq (Hz))** list, select **700**.

4 In the **Surface Impedance: 700Hz** toolbar, click **Plot**.

The surface impedance for both studies, isotropic and transverse isotropic, for 700 Hz should look like the following figure:



Surface Impedance: 700Hz

In the **Model Builder** window, right-click **Surface Impedance: 700Hz** and choose **Duplicate**.

Surface Impedance: 1000Hz

1 In the **Model Builder** window, under **Results** click **Surface Impedance: 700Hz 1**.

2 In the **Settings** window for **1D Plot Group**, type `Surface Impedance: 1000Hz` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (freq (Hz))** list, select **1000**.

Global 2

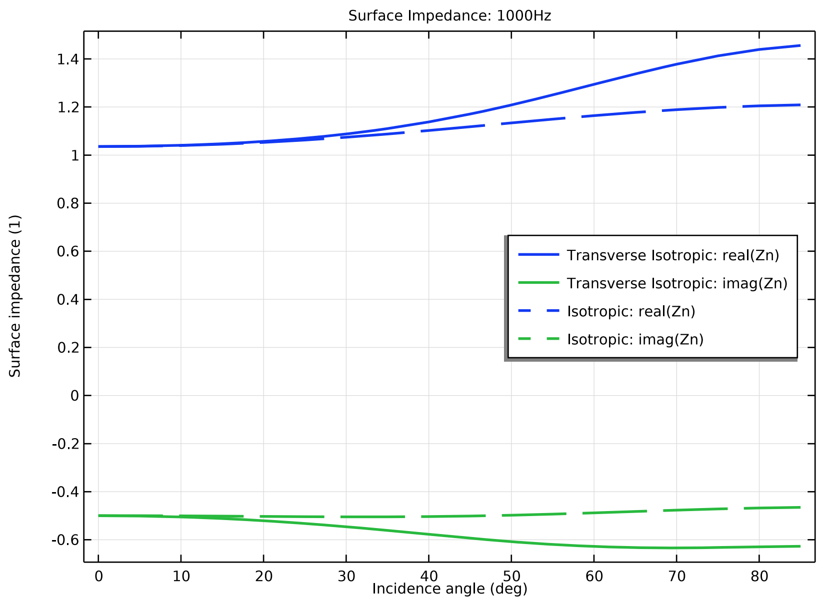
1 In the **Model Builder** window, expand the **Surface Impedance: 1000Hz** node, then click **Global 2**.

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

3 In the **Parameter values (freq (Hz))** list, select **1000**.

4 In the **Surface Impedance: 1000Hz** toolbar, click **Plot**.

The surface impedance for both studies, isotropic and transverse isotropic, for 1000 Hz should look like the following figure:



Surface Impedance: 1000Hz

In the **Model Builder** window, right-click **Surface Impedance: 1000Hz** and choose **Duplicate**.

Surface Impedance: 3000Hz

1 In the **Model Builder** window, under **Results** click **Surface Impedance: 1000Hz 1**.

2 In the **Settings** window for **1D Plot Group**, type `Surface Impedance: 3000Hz` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (freq (Hz))** list, select **3000**.

Global 2

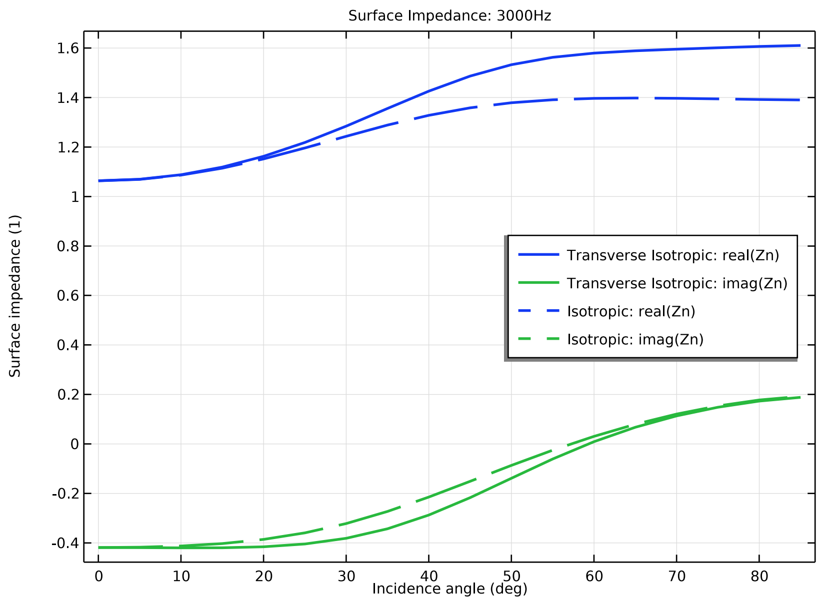
1 In the **Model Builder** window, expand the **Surface Impedance: 3000Hz** node, then click **Global 2**.

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

3 In the **Parameter values (freq (Hz))** list, select **3000**.

4 In the **Surface Impedance: 3000Hz** toolbar, click **Plot**.

The surface impedance for both studies, isotropic and transverse isotropic, for 3000 Hz should look like the following figure:



Absorption Coefficients

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

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

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Incidence angle (deg)`.

6 Locate the **Axis** section. Select the **Manual axis limits** check box.

7 In the **x minimum** text field, type `-0.5`.

8 In the **x maximum** text field, type `85.5`.

9 In the **y minimum** text field, type `-0.01`.

10 In the **y maximum** text field, type `1.01`.

11 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 In the **Absorption Coefficients** toolbar, click **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
alpha	1	Absorption coefficient

4 Locate the **Legends** section. Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Transverse Isotropic: `.

Absorption Coefficients

In the **Absorption Coefficients** toolbar, click **Global**.

Global 2

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

2 From the **Dataset** list, choose **Study 2 - Isotropic/Solution 2 (sol2)**.

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

Expression	Unit	Description
alpha	1	Absorption coefficient

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Locate the **Legends** section. Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Isotropic: `.

7 In the **Absorption Coefficients** toolbar, click **Plot**.

The absorption coefficients for both studies, isotropic and transverse isotropic, and for all four frequencies, 500 Hz, 700 Hz, 1000 Hz, 3000Hz, should look like the following figure: