Created in COMSOL Multiphysics 6.3

Anisotropic Porous Absorber

Introduction

This model demonstrates how to model the absorption of an anisotropic porous material. The porous material is modeled with both poroelastic and poroacoustic waves. Comparing the two methods shows that it can be important to use the Poroelastic Waves interface when the poroelastic material has eigenfrequencies in the frequency range of interest.

Model Definition

Figure 1 depicts the geometry of the modeled system in which an incident sound field, with incident angle θ, hits an anisotropic porous absorber.

We only model an absorber section of width W and use periodic Floquet boundary conditions on the left and right boundaries. The anisotropic porous material is 4 cm thick with a fixed wall at the bottom boundary. At the top there is a perfectly matched layer (PML) domain to model an infinitely large air domain.

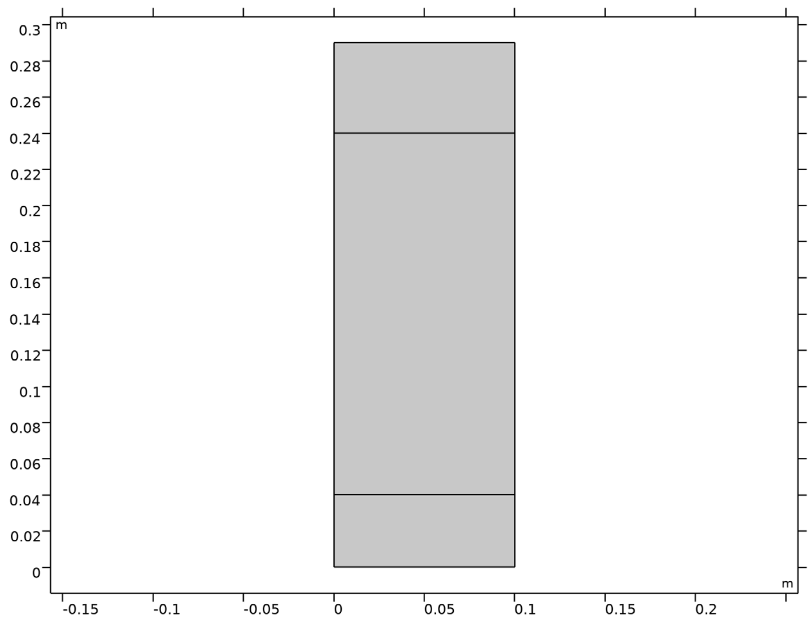


Figure 1: Model geometry with periodic condition at the right and left and a PML layer at the top.

Two components with the same geometry are created to compare two different methods of modeling the anisotropic porous material. The first method is using the **Anisotropic**

Poroacoustics feature in **Pressure Acoustics, Frequency Domain** interface using a Johnson–Champoux–Allard (JCA) model. The second method uses the **Poroelastic Waves** interface with an **Anisotropic Poroelastic Material** feature with a Biot–Allard poroelastic model.

The anisotropic porous material used in the model is a generic anisotropic foam.

Results and Discussion

Figure 2 shows the acoustic field at 5000 Hz and an incident angle of 80 degrees. An Array dataset has been used to extend the solution to a width of four times the modeling domain. The pressure is shown for the model using poroacoustics for the porous material.

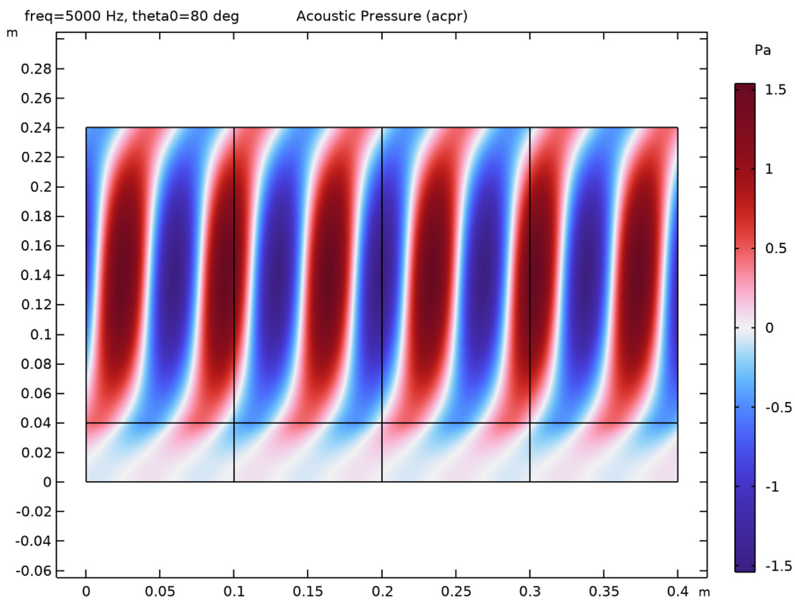


Figure 2: The acoustic pressure at 5000 Hz and an incident angle of 80 degrees.

Figure 3 shows the absorption coefficient for three different incident angles — 0, 40, and 80 degrees — and for both the poroacoustic model (dashed) and the poroelastic model (solid). The absorption spectra from the poroacoustic model are smooth and do not include the effects of resonances in the poroelastic material. These effects are included in the poroelastic models, which results in resonance effects around 500–1000 Hz. The main resonance around 580 Hz is the first compressional eigenmode of the anisotropic poroelastic structure. For the two nonzero incident angles there are also traces of an eigenmode around 950 Hz. This mode cannot be actuated by an acoustic field with a 0 degree incident angle and thus does not affect the absorption spectrum for this case.

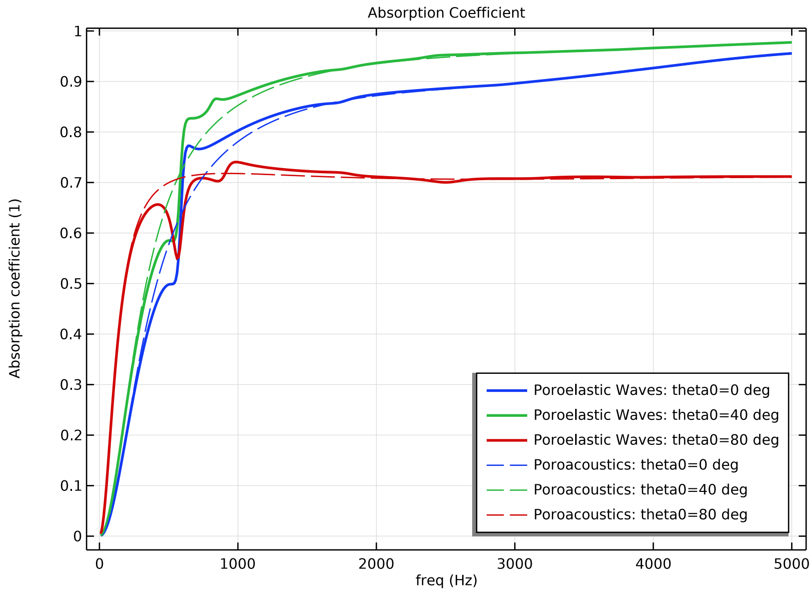


Figure 3: The absorption coefficient for three incident angles and for poroacoustics (dashed line) and poroelastic waves (solid lines).

When the acoustic frequency is close to a resonance it affects the viscous absorption because the relative velocity between the air and structure in the poroelastic material is affected. It can both reduce the absorption when the air and structure are moving in phase and increase the absorption when they are moving out of phase. This is demonstrated in Figure 4, where the absorption coefficient is plotted together with the phase difference between the fluid and frame motion in the porous material.

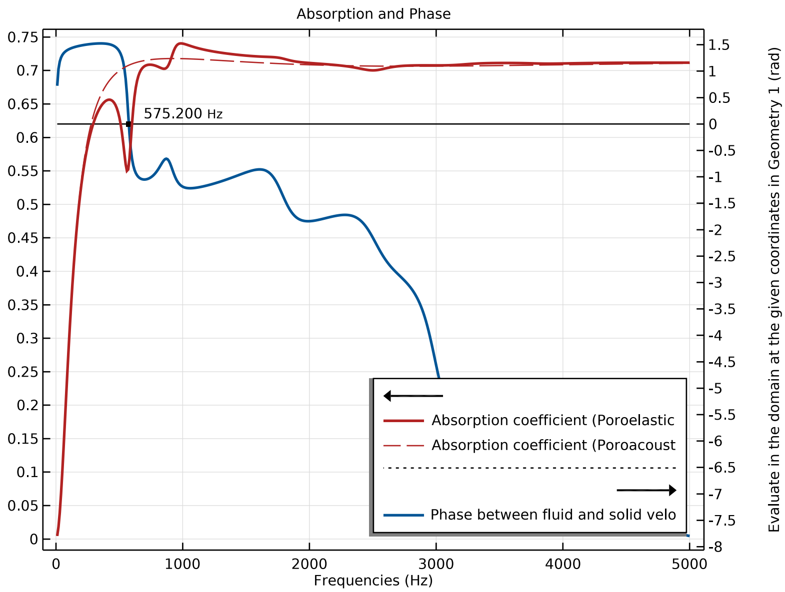


Figure 4: Left y-axis: The absorption coefficients for poroacoustic (dashed) and poroelastic (solid) waves. Right y-axis: the phase between the fluid and frame motion in the poroelastic material.

Application Library path: `Acoustics_Module/Building_and_Room_Acoustics/anisotropic_porous_absorber`

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**.

The first component will have the anisotropic poroacoustics defined in **Pressure Acoustics, Frequency Domain (acpr)**, which corresponds to the fluid moving while the frame of the porous material is fixed. A second component, to be added later, will compare this case with using a **Poroelastic Waves (pelw)** interface, which includes the movement of the frame.

2 In the **Select Physics** tree, select **Acoustics** > **Pressure Acoustics** > **Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 Click **Study**.

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

6 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 `anisotropic_porous_absorber_parameters.txt`.

GEOMETRY 1

The simple geometry to be created for this model corresponds to an air domain followed by the porous material. A perfectly matched layer is added on top of the air domain.

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:

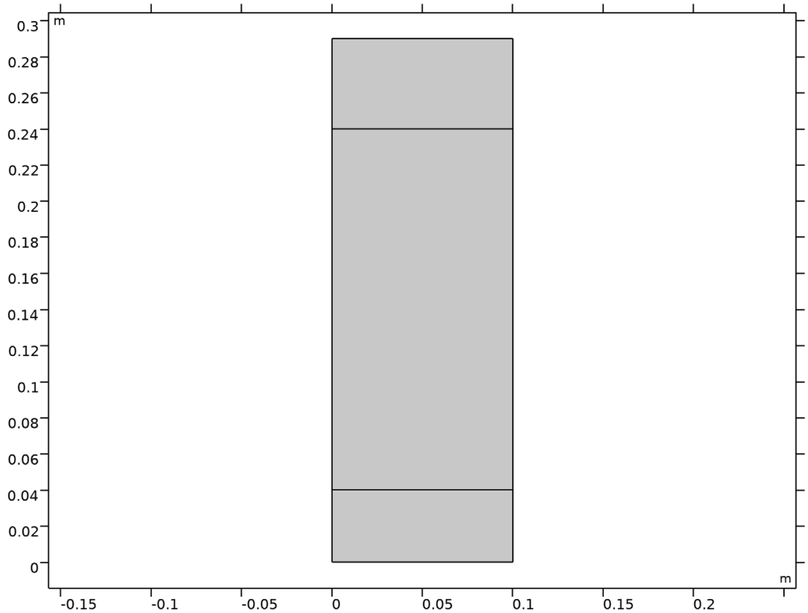
Layer name	Thickness (m)
Layer 1	H
Layer 2	Hair

Form Union (fin)

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

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

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



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 **Load from File**.

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

Integration 1 (intop1)

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

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

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

4 Select Point 3 only.

5 In the **Operator name** text field, type `intop_pnt`.

Average 1 (aveop1)

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 Boundary 4 only.
5 In the **Operator name** text field, type `aveop_bnd`.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 4 only.
5 In the **Operator name** text field, type `intop_bnd`.

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 **Materials** 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 the **Add to Component** button in the window toolbar.
5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Porous Material

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Porous Material` in the **Label** text field.
3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Basic Properties** > **Porosity**.

4 Click + **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Porosity	epsilon	0.98	1	Basic

6 Locate the **Material Properties** section. In the **Material properties** tree, select **Acoustics** > **Poroacoustics Model**.

7 Click + **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Flow resistivity	{Rf11, Rf22, Rf33} ; Rfij = 0	{26000[N*s/m^4], 48000[N*s/m^4], 26000[N*s/m^4]}	Pa·s/m²	Poroacoustics model
Thermal characteristic length	Lth	150[um]	m	Poroacoustics model
Viscous characteristic length	{Lv11, Lv22, Lv33} ; Lvij = 0	{68[um], 57[um], 68[um]}	m	Poroacoustics model
Tortuosity factor	{tau11, tau22, tau33} ; tauij = 0	{1.03, 1.08, 1.03}	1	Poroacoustics model

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Background Pressure Field 1

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

2 Select Domain 2 only.

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

4 In the p_0 text field, type 1.

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

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

kx_e	x
ky_e	y

7 Select the **Calculate background and scattered field intensity** checkbox.

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

Periodic Condition 1

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 $\mathbf{k}_\mathrm{F}$ vector as

kx	x
ky	y

Periodic Condition 2

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 $\mathbf{k}_\mathrm{F}$ vector as

kx	x
ky	y

Periodic Condition 3

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

Anisotropic Poroacoustics 1

1 In the **Physics** toolbar, click **Domains** and choose **Anisotropic Poroacoustics**.
2 Select Domain 1 only.
3 In the **Settings** window for **Anisotropic Poroacoustics**, locate the **Fluid Properties** section.
4 From the **Fluid material** list, choose **Air (mat1)**.
5 Locate the **Porous Matrix Properties** section. From the **Porous elastic material** list, choose **Porous Material (mat2)**.

MESH 1

Next, set up the mesh manually, starting by adding the mesh components.

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/24`.
5 In the **Minimum element size** text field, type `H/24`.

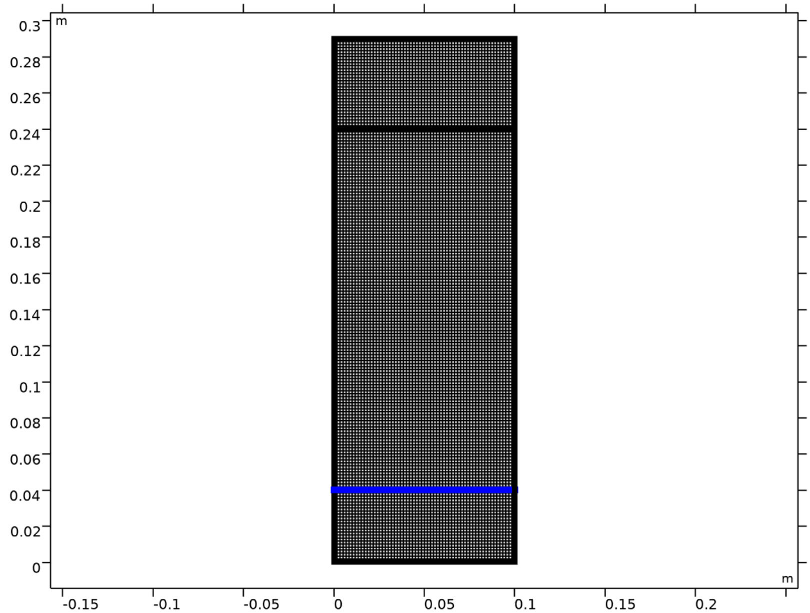
Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 2 only.
5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** checkbox.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.
2 Select Boundary 4 only.
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.
4 In the **Number of layers** text field, type `1`.

5 Click **Build All**.



STUDY 1 - POROACOUSTICS WAVES

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

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

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

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

Step 1: Frequency Domain

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

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

3 Click **Range**.

4 In the **Range** dialog, type `10` in the **Start** text field.

5 In the **Step** text field, type `10`.

6 In the **Stop** text field, type `5000`.

7 Click **Add**.

8 In the **Settings** window for **Frequency Domain**, click to expand the **Study Extensions** section.

9 Select the **Auxiliary sweep** checkbox.

10 Click **Add**.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Angle of incidence)	`0,40,80`	`deg`

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

COMPONENT 1 (COMP1)

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

COMPONENT 2 (COMP2)

In the **Model Builder** window, right-click the root node and choose **Paste Multiple Items**.

ARTIFICIAL DOMAINS, COMPONENT 2 (COMP2), DEFINITIONS (COMP2), GEOMETRY 1, MATERIALS, MESH 1, PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR2)

In the **Messages from Paste** dialog, click **OK**.

DEFINITIONS (COMP2)

Variables 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Definitions** click **Variables 1**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
k0	comp2.intop_pnt(acpr2.k)	rad/m	Free-field wave number
Zn	comp2.aveop_bnd(acpr2.p_t/(nx*up(acpr2.vx)+ny*up(acpr2.vy)))	Pa·s/m	Specific surface impedance
Pin	comp2.intop_bnd(-acpr2.I_by)	W/m	Incident power
Pout	comp2.intop_bnd(acpr2.I_sy)	W/m	Outgoing power

MATERIALS

Porous Material (mat4)

1. In the **Model Builder** window, under **Component 2 (comp2)** > **Materials** click **Porous Material (mat4)**.
2. Select Domain 1 only.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR2)

Anisotropic Poroacoustics 1

1. In the **Model Builder** window, expand the **Component 2 (comp2)** > **Pressure Acoustics, Frequency Domain (acpr2)** node.
2. Right-click **Component 2 (comp2)** > **Pressure Acoustics, Frequency Domain (acpr2)** > **Anisotropic Poroacoustics 1** and choose **Delete**.

Periodic Condition 3

1. Right-click **Periodic Condition 3** and choose **Delete**.
2. In the **Model Builder** window, click **Pressure Acoustics, Frequency Domain (acpr2)**.
3. Select Domains 2 and 3 only.

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 **Acoustics** > **Elastic Waves** > **Poroelastic Waves (pelw)**.
4. Click the **Add to Component 2** button in the window toolbar.
5. In the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

POROELASTIC WAVES (PELW)

Select Domain 1 only.

Poroelastic Material 1

1. In the **Model Builder** window, under **Component 2 (comp2)** > **Poroelastic Waves (pelw)** click **Poroelastic Material 1**.
2. In the **Settings** window for **Poroelastic Material**, locate the **Porous Matrix Properties** section.
3. From the **Specify** list, choose **Young's modulus and shear modulus**.
4. Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Air (mat3)**.

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 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Air (mat3)**.

In this case, you need both **Fixed Constraint** and **Impervious Layer** because both the fluid and the frame are fixed to the bottom.

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 $\mathbf{k}_F$ vector as

kx	X
ky	Y

MATERIALS

Porous Material (mat4)

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Materials** click **Porous Material (mat4)**.
2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evector1, Evector2, Evector3}	`{42e5, 12e5, 42e5}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{0, 0, 0}`	1	Orthotropic
Shear modulus, Voigt notation	{GvectorVo1, GvectorVo2, GvectorVo3}	`{21e5, 60e4, 21e5}`	N/m²	Orthotropic, Voigt notation
Density	rho	`140`	kg/m³	Basic
Isotropic structural loss factor	eta_s	`0.1`	1	Basic

MULTIPHYSICS

Acoustic–Porous Boundary 1 (apb1)

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

2 Select Boundary 4 only.

MESH 1

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 1** and choose **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.

STUDY 2

Step 1: Frequency Domain

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

2 Click **Range**.

3 In the **Range** dialog, type `10` in the **Start** text field.

4 In the **Step** text field, type `10`.

5 In the **Stop** text field, type `5000`.

6 Click **Add**.

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

8 In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

9 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

10 Click **Add**.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Angle of incidence)	`0,40,80`	`deg`

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

13 In the **Settings** window for **Study**, locate the **Study Settings** section.

14 Clear the **Generate default plots** checkbox.

15 In the **Label** text field, type `Study 2 - Poroelastic Waves`.

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

RESULTS

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

Study 2 - Poroelastic Waves/Solution 2 (2) (sol2)

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

2 Right-click **Results > Datasets > Study 2 - Poroelastic Waves/Solution 2 (2) (sol2)** and choose **Delete**.

Array 2D 1

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 **X size** text field, type `4`.

4 Click to expand the **Advanced** section. Select the **Floquet-Bloch periodicity** checkbox.

5 Find the **Wave vector** subsection. In the **X** text field, type `kx`.

6 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.

Study 1 - Poroacoustics Waves/Solution 1 (4) (sol1)

In the **Results** toolbar, click **More Datasets** and choose **Solution**.

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 Select Domain 1 only.

Acoustic Pressure (acpr)

1 In the **Results** toolbar, click **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 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Color Legend** section. Select the **Show units** checkbox.

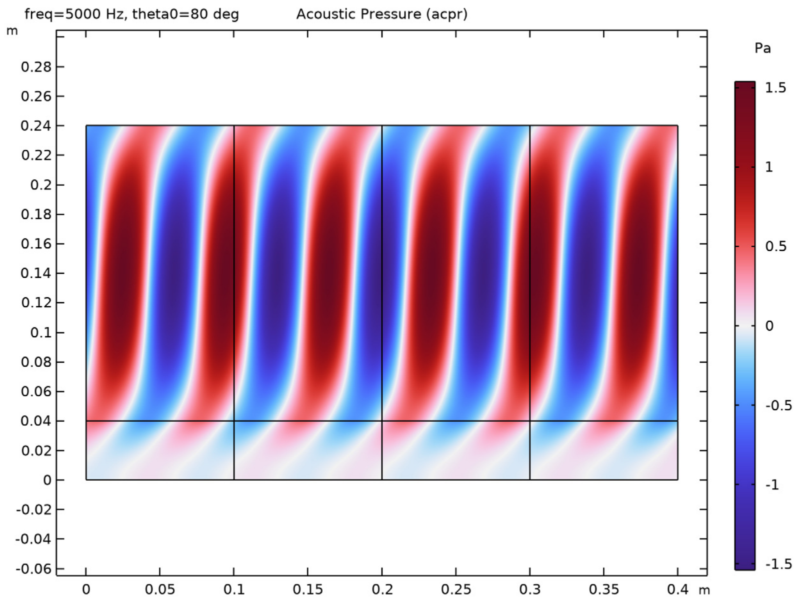
Surface 1

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

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

3 From the **Scale** list, choose **Linear**.

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



Sound Pressure Level (acpr)

1 In the **Model Builder** window, right-click **Acoustic Pressure (acpr)** and choose **Duplicate**.

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

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

Surface 1

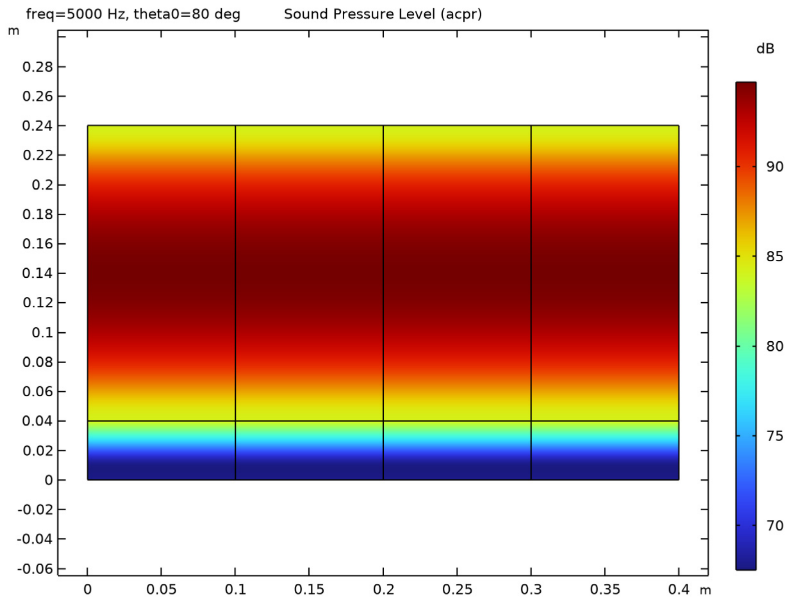
1 In the **Model Builder** window, click **Surface 1**.

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

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

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

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



Absorption Coefficient

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

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

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

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Poroelastic Waves/ Solution 2 (sol2)**.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Absorption Coefficient** 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 **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

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

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

7 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Poroelastic Waves:` .

Absorption Coefficient

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

Global 2

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

2 From the **Dataset** list, choose **Study 1 - Poroacoustics Waves/Solution 1 (1) (sol1)**.

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

Expression	Unit	Description
alpha	1	Absorption coefficient

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

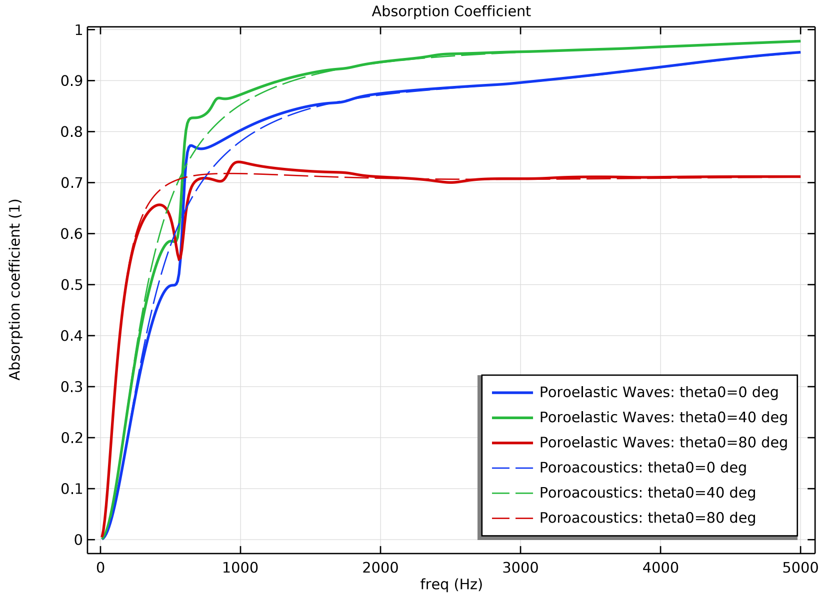
5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

7 Locate the **Legends** section. Find the **Include** subsection. Clear the **Description** checkbox.

8 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Poroacoustics:` .

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



Vertical velocity of Fluid and Frame at 80 deg and 500 Hz

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

2 In the **Settings** window for **2D Plot Group**, type `Vertical velocity of Fluid and Frame at 80 deg and 500 Hz` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Poroelastic Waves/ Solution 2 (sol2)**.

4 From the **Parameter value (freq (Hz))** list, choose **500**.

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

6 Locate the **Color Legend** section. Select the **Show units** checkbox.

7 Click to expand the **Plot Array** section. Select the **Enable** checkbox.

Surface 1

1 In the **Vertical velocity of Fluid and Frame at 80 deg and 500 Hz** toolbar, click **Surface**.

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

3 In the **Expression** text field, type `pelw.iomega*v`.

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

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

Vertical velocity of Fluid and Frame at 80 deg and 500 Hz

In the **Vertical velocity of Fluid and Frame at 80 deg and 500 Hz** toolbar, click **Surface**.

Surface 2

1. In the **Settings** window for **Surface**, locate the **Expression** section.
2. In the **Expression** text field, type `pelw.v_tY`.
3. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Vertical velocity of Fluid and Frame at 80 deg and 500 Hz

In the **Vertical velocity of Fluid and Frame at 80 deg and 500 Hz** toolbar, click **Surface**.

Surface 3

1. In the **Settings** window for **Surface**, locate the **Data** section.
2. From the **Dataset** list, choose **Study 1 - Poroacoustics Waves/Solution 1 (4) (sol1)**.
3. From the **Parameter value (freq (Hz))** list, choose **500**.
4. Locate the **Expression** section. In the **Expression** text field, type `acpr.vy`.
5. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Annotation 1

1. In the **Model Builder** window, right-click **Vertical velocity of Fluid and Frame at 80 deg and 500 Hz** and choose **Annotation**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `Study 2: Solid velocity`.
4. Locate the **Coloring and Style** section. Clear the **Show point** checkbox.
5. Click to expand the **Plot Array** section. Select the **Belongs to array** checkbox.
6. Select the **Manual indexing** checkbox.

Annotation 2

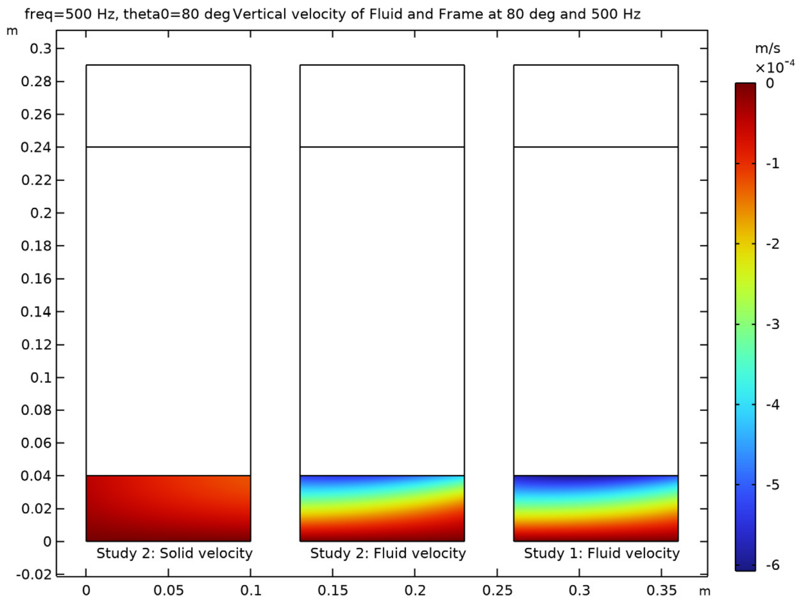
1. Right-click **Annotation 1** and choose **Duplicate**.
2. In the **Settings** window for **Annotation**, locate the **Plot Array** section.
3. In the **Index** text field, type `1`.
4. Locate the **Annotation** section. In the **Text** text field, type `Study 2: Fluid velocity`.

Annotation 3

1. Right-click **Annotation 2** and choose **Duplicate**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `Study 1: Fluid velocity`.

4 Locate the **Plot Array** section. In the **Index** text field, type `2`.

5 In the **Vertical velocity of Fluid and Frame at 80 deg and 500 Hz** toolbar, click **Plot**.



Vertical velocity of Fluid and Frame at 80 deg and 5000 Hz

1 In the **Model Builder** window, right-click **Vertical velocity of Fluid and Frame at 80 deg and 500 Hz** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Vertical velocity of Fluid and Frame at 80 deg and 5000 Hz` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **5000**.

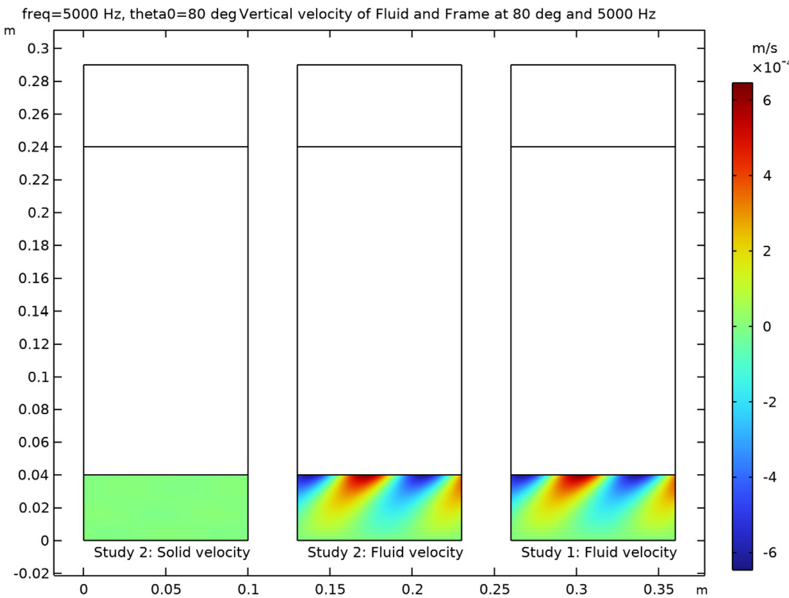
Surface 3

1 In the **Model Builder** window, expand the **Vertical velocity of Fluid and Frame at 80 deg and 5000 Hz** node, then click **Surface 3**.

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

3 From the **Parameter value (freq (Hz))** list, choose **5000**.

4 In the **Vertical velocity of Fluid and Frame at 80 deg and 5000 Hz** toolbar, click **Plot**.



Absorption and Phase

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Poroelastic Waves/ Solution 2 (sol2)**.

4 From the **Parameter selection (theta0)** list, choose **From list**.

5 In the **Parameter values (theta0 (deg))** list, select **80**.

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

7 Locate the **Plot Settings** section.

8 Select the **x-axis label** checkbox. In the associated text field, type `Frequencies (Hz)`.

9 Select the **Two y-axes** checkbox.

10 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Absorption and Phase** toolbar, click **Global**.

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

3 Select the **Plot on secondary y-axis** checkbox.

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

Expression	Unit	Description
`at2(0.06,0.02, arg(pelw.iomega*v)-arg(pelw.vY))`	`rad`	`Evaluate in the domain at the given coordinates in Geometry 1`

5 Select the **Unwrap phase** checkbox.

6 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

7 Locate the **Legends** section. Find the **Include** subsection. Clear the **Solution** checkbox.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Custom**.

9 Click **Define custom colors**.

10 Set the RGB values to 0, 85, and 150, respectively.

11 Click **Add to custom colors**.

12 Click **Show color palette only** or **OK** on the cross-platform desktop.

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

14 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

15 In the table, enter the following settings:

Legends
`Phase between fluid and solid velocity`

Graph Marker 1

1 Right-click **Global 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Display mode** list, choose **Line intersection**.

4 From the **Line type** list, choose **Horizontal**.

5 In the **y-coordinates** text field, type `0`.

6 Select the **Show lines** checkbox.

7 Locate the **Text Format** section. Select the **Include unit** checkbox.

Absorption and Phase

In the **Absorption and Phase** toolbar, click **Global**.

Global 2

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

2 In the table, enter the following settings:

Expression	**Unit**	**Description**
alpha	1	Absorption coefficient (Poroelastic Waves)

3 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

4 Locate the **Legends** section. Find the **Include** subsection. Clear the **Solution** checkbox.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Custom**.

6 Click **Define custom colors**.

7 Set the RGB values to 178, 34, and 34, respectively.

8 Click **Add to custom colors**.

9 Click **Show color palette only** or **OK** on the cross-platform desktop.

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

Global 3

1 Right-click **Results** > **Absorption and Phase** > **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1 - Poroacoustics Waves/Solution 1 (1) (sol1)**.

4 From the **Parameter selection (theta0)** list, choose **From list**.

5 In the **Parameter values (theta0 (deg))** list, select **80**.

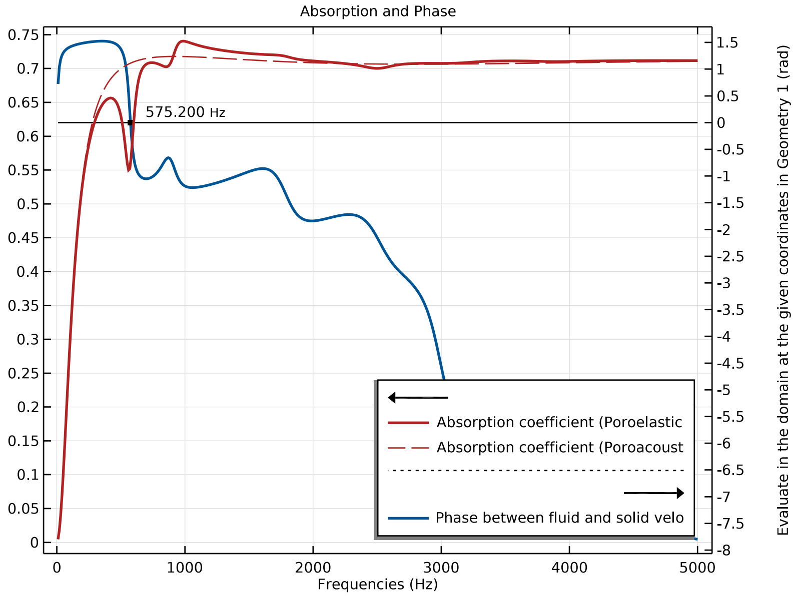
6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
alpha	1	Absorption coefficient (Poroacoustics)

7 Locate the **Coloring and Style** section. From the **Width** list, choose **1**.

8 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

9 In the **Absorption and Phase** toolbar, click **Plot**.



Shear-Wave Speed (Real Part)

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

2 In the **Settings** window for **1D Plot Group**, type `Shear-Wave Speed (Real Part)` in the **Label** text field.

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

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Poroelastic Waves/ Solution 2 (sol2)**.

5 From the **Parameter selection (theta0)** list, choose **From list**.

6 In the **Parameter values (theta0 (deg))** list, select **80**.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Shear-Wave Speed (Real Part)** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `pelw.apm1.cs_poroXX`.

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`x direction`

Shear-Wave Speed (Real Part)

In the **Shear-Wave Speed (Real Part)** toolbar, click **Point Graph**.

Point Graph 2

1 Select Point 1 only.

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

3 In the **Expression** text field, type `pelw.apm1.cs_poroYY`.

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

5 Locate the **Legends** section. Select the **Show legends** checkbox.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`y direction`

Shear-Wave Speed (Real Part)

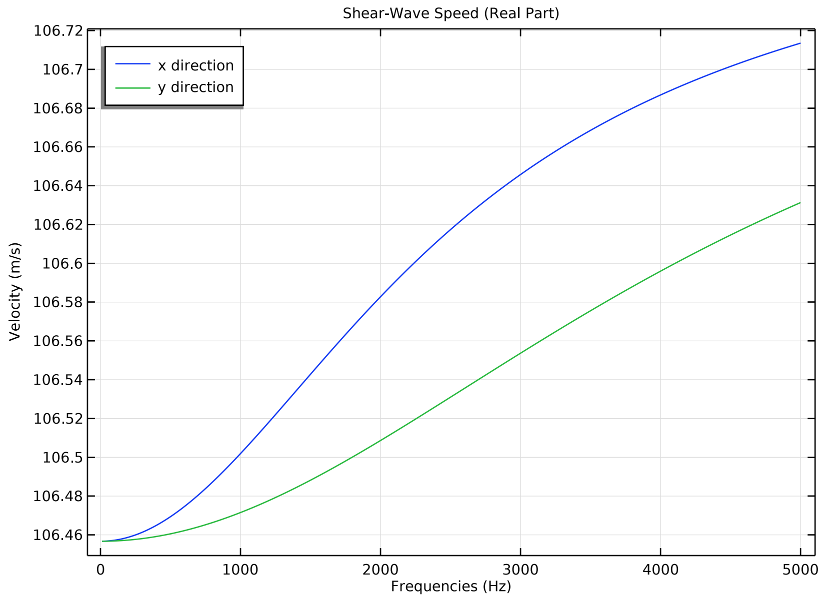
1 In the **Model Builder** window, click **Shear-Wave Speed (Real Part)**.

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

3 Select the **x-axis label** checkbox. In the associated text field, type `Frequencies (Hz)`.

4 Select the **y-axis label** checkbox. In the associated text field, type `Velocity (m/s)`.

5 In the **Shear-Wave Speed (Real Part)** toolbar, click **Plot**.



Shear-Wave Speed (Real Part) 1

Right-click **Shear-Wave Speed (Real Part)** and choose **Duplicate**.

Point Graph 1

1 In the **Model Builder** window, expand the **Shear-Wave Speed (Real Part) 1** node, then click **Point Graph 1**.

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

3 In the **Expression** text field, type `pelw.apm1.cp_fastXX`.

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

Legends
`fast x direction`

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 In the **Expression** text field, type `pelw.apm1.cp_fastYY`.

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

Legends
`fast y direction`

Point Graph 1, Point Graph 2

1 In the **Model Builder** window, under **Results** > **Shear-Wave Speed (Real Part) 1**, Ctrl-click to select **Point Graph 1** and **Point Graph 2**.

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

Pressure Wave Speeds (Real Part)

In the **Settings** window for **1D Plot Group**, type `Pressure Wave Speeds (Real Part)` in the **Label** text field.

Point Graph 3, Point Graph 4

1 In the **Model Builder** window, under **Results** > **Pressure Wave Speeds (Real Part)**, Ctrl-click to select **Point Graph 3** and **Point Graph 4**.

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

3 In the **Expression** text field, type `pelw.apm1.cp_slowXX`.

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

Legends
`slow x direction`

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

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

3 In the **Expression** text field, type `pelw.apm1.cp_slowYY`.

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

Legends
`slow y direction`

5 In the **Pressure Wave Speeds (Real Part)** toolbar, click **Plot**.