Acoustic Reflections off a Water-Sediment Interface

Introduction

This validation model computes the reflection coefficient of acoustic waves off a water-sediment interface. Homogeneous waves are incident from a fluid (water) domain and are reflected and transmitted at a water-sediment interface. The sediment domain is modeled using Biot's theory with the use of the Poroelastic Waves interface. It includes the detailed interaction between the pressure waves in the saturating fluid and the elastic waves in the porous matrix (the sediment). The model results are compared with those obtained by Stoll and Kan (Ref. 1). A visual comparison of the results with those presented in Ref. 1 shows good agreement.

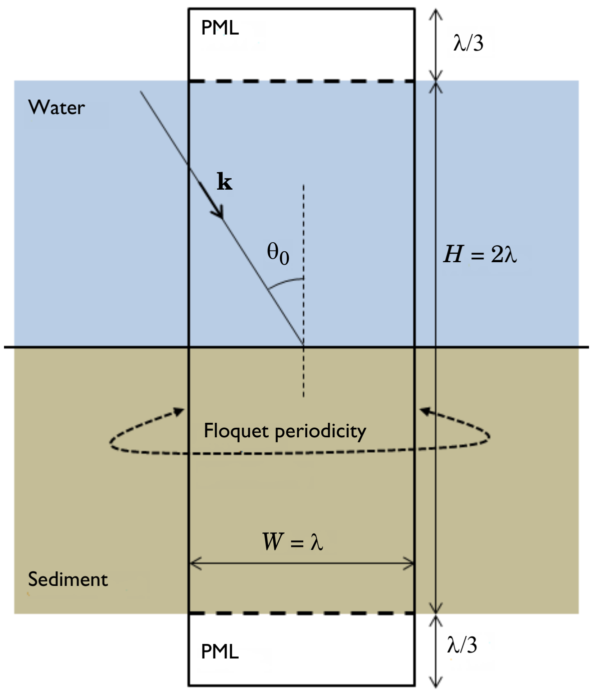


Figure 1: Sketch of the water-sediment system. The computational unit cell has the width W *and the height* H *with perfectly matched layers at the top and bottom.* λ *is the acoustic wavelength in water. An acoustic wave with wave vector* $\mathbf{k}$ *is incident at an angle* θ_0 *on the water-sediment interface.*

Knowledge about the plane wave reflection and refraction coefficients are important when interpreting acoustic data taken from marine seismology. The kind of model studied here can provide insight into the different phenomena at play. The sediment is modeled using Biot's theory with the addition of the porous matrix skeleton having viscoelastic properties. This is achieved by simply allowing the bulk modulus and the shear modulus to have a small imaginary component (a constant attenuation factor). Inside the porous material, three types of waves can propagate: a fast and a slow pressure wave, and a shear wave. The interplay between these waves and the incoming homogeneous pressure wave in the fluid results in nontrivial behavior of the reflection coefficient. See Ref. 1 for further discussion.

Model Definition

A sketch of the modeled system is shown in Figure 1. An acoustic wave is incident on the water-sediment interface at an angle θ_0, defining the wave vector

$$\mathbf{k} = k_0(\sin(\theta_0), -\cos(\theta_0)) \qquad k_0 = \frac{\omega}{c}$$

where ω is the angular frequency $2\pi f$. The wave vector is used to define an incident plane wave of the form

$$P_{\mathrm{in}}(\mathbf{x}) = 1\ \mathrm{Pa} \cdot \exp(-i(\mathbf{k} \cdot \mathbf{x}))$$

where $\mathbf{x} = (x, y)$. The wave vector is defined as a variable in the model and used to define the direction of the plane wave in the **Background Pressure Field** feature, as well as the wave vector for the Floquet periodicity in the model. Note that in the nomenclature of the Acoustics Module the incident field is the background acoustic field.

The fluid domain is modeled by classical pressure acoustics (lossless) solving the Helmholtz equation. The sediment domain is modeled by Biot's theory solving for both the pressure and the displacement field of the porous matrix. The coupled multiphysics problem is set up using the Acoustic-Poroelastic Waves Interaction multiphysics interface.

The problem is 2D and is modeled as a periodic structure using Floquet periodic conditions on both the acoustic and the porous domains. To ensure that the problem is well defined numerically, the domain size is set to depend on the wavelength in water (thus the frequency). The width of the computational unit cell W is equal to the wavelength and the height H is equal to two times the wavelength. The computational domain is truncated, in both the fluid and in the porous domain, using perfectly matched layers (PMLs). The thickness is set equal to one third of the wavelength.

The material parameters necessary for defining the properties of the porous domain are taken from Ref. 1 and some are calculated using these values. The porous material is here assumed to be sand. The most important parameters and their definitions are given in Table 1 below.

TABLE 1: POROUS MATERIAL PARAMETERS FOR SAND.

SYMBOL	VALUE	DESCRIPTION
μ_F	$1 \cdot 10^{-3}$ Pa·s	Fluid viscosity (water)
ρ_F	1000 kg/m^3	Fluid density (water)
ε_P	0.47	Porosity
κ_P	$1 \cdot 10^{-6}$ cm^2	Permeability
K_b	$4.36 \cdot 10^7$ Pa	Bulk modulus of frame (drained porous matrix)
K_s	$3.6 \cdot 10^{10}$ Pa	Bulk modulus of solid grains (constituting the frame)
α_B	$1 - K_b/K_s = 0.999$	Biot-Willis coefficient: Because it is very close to 1 the frame is said to be limp.
$\log_{dec}$	0.15	Logarithmic decrement factor
K_{bc}	$(1+i \cdot \log_{dec}/\pi) \cdot K_b$	Complex bulk modulus of frame
G	$2.61 \cdot 10^7$ Pa	Shear modulus of frame (drained porous matrix)
G_c	$(1+i \cdot \log_{dec}/\pi) \cdot G$	Complex shear modulus of frame
ρ_s	2650 kg/m^3	Solid density (of material constituting the frame)
ρ_d	$(1-\varepsilon_P) \cdot \rho_s$ = 1404.5 kg/m^3	Drained density (of the porous matrix)
τ	1.25	Tortuosity (chosen, not specified in Ref. 1)
a	$4 \cdot 10^{-3}$ cm	Pore size parameter (chosen to have f_c = 100 Hz)
f_r	$\mu_F/(2\pi \cdot \rho_F \cdot a^2)$ = 99.5 Hz	Reference frequency: frequency at which the viscous boundary layer thickness (viscous penetration depth) is equal to the pore size parameter a. This value is automatically calculated by COMSOL

Note: Further information about the physics solved by the Poroelastic Waves interface, about Biot's theory, and about the meaning of the material parameters is given in the documentation. Press Ctrl+F1 and browse to the *Acoustics Module* and the *User's Guide*. The relevant theory is found in *Elastic Waves Interfaces>Theory for the Poroelastic Waves Interfaces*.

Results and Discussion

The reflection coefficient R and the absorption coefficient α are plotted for different combinations of the angle of incidence and the frequency (the model parameters). The reflection coefficient is the complex ratio of the reflected (scattered) pressure to the incident (background) pressure, and is given by

$$R = \frac{p_{\mathrm{re}}}{p_{\mathrm{in}}} = \frac{p_{\mathrm{tot}} - p_{\mathrm{in}}}{p_{\mathrm{in}}} \tag{1}$$

where the subscript re stands for reflected and tot for total. The absorption coefficient α gives a measure of the relative absorbed energy. It is in general defined through the incident power P_{in} and the reflected power P_{re}, through

$$\alpha = 1 - \frac{P_{\mathrm{in}}}{P_{\mathrm{re}}} \tag{2}$$

Note that for plane waves (when no higher oder diffraction modes exist) the absorption coefficient is given by

$$\alpha = 1 - |R|^2 \tag{3}$$

The absolute value of the reflection coefficient R is shown in Figure 2 and Figure 3. Note that average of the reflection coefficient across the interface is used. This is because the model extracts the effective homogenized properties of the water-sediment interface. In the first figure, R is depicted as function of the angle of incidence θ_0 and in the second figure as function of the frequency f. The absorption coefficient α, corresponding to the reflection coefficient in Figure 2, is shown in Figure 4. An interesting feature that can be deduced from Figure 3 is that the reflection at the interface acts as a filter with respect to a broadband signal; this is especially true at high angles of incidence.

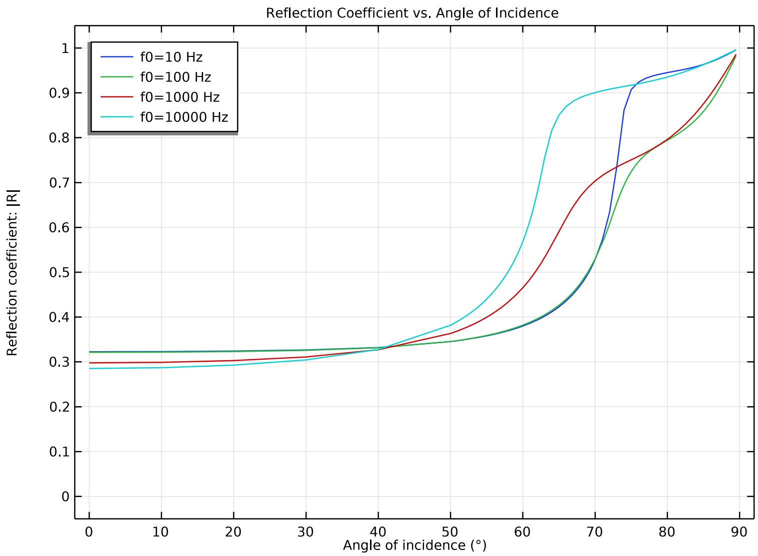


Figure 2: Absolute value of the reflection coefficient R as function of the angle of incidence for the driving frequencies 10 Hz, 100 Hz, 1 kHz, and 10 kHz.

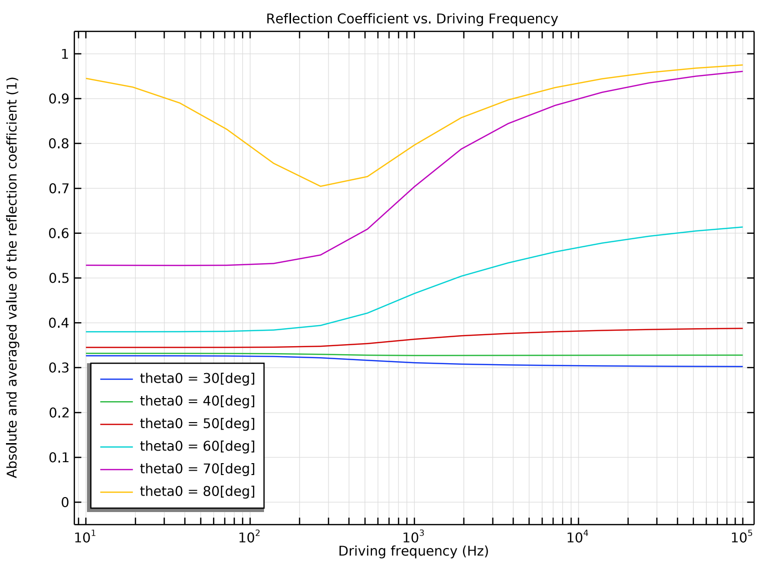


Figure 3: Absolute value of the reflection coefficient R as function of frequency for the angles of incidence 30°, 40°, 50°, 60°, 70°, and 80°.

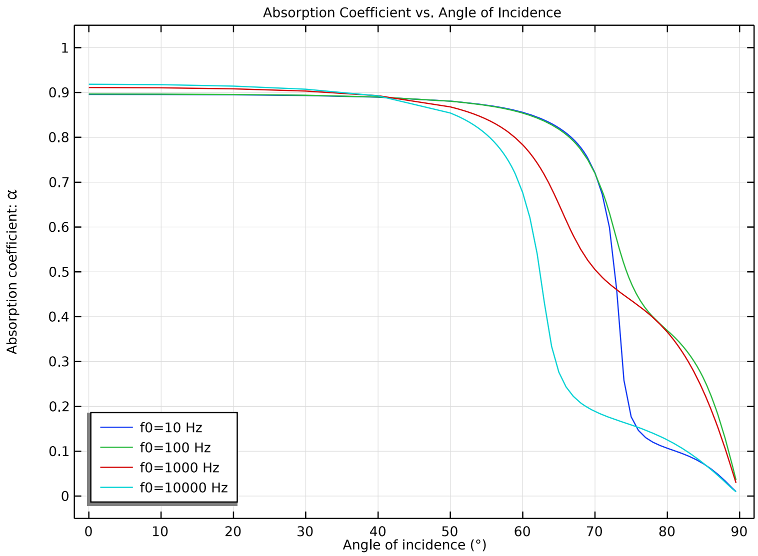


Figure 4: Absorption coefficient α *as function of the angle of incidence for the driving frequencies 10 Hz, 100 Hz, 1 kHz, and 10 kHz.*

As can be seen from Figure 2, the reflection coefficient has significantly different behavior for 3 of the driving frequencies at an incidence angle of, for example, 70°. The total pressure is shown in Figure 5 and the displacement field is shown in Figure 6. Note the different length scales as the computational domain (geometry) scales with the wavelength.

From the graphs in Figure 2 and Figure 4, it is evident that the behavior shifts for the different frequencies for increasing angles of incidence. At 90°, the reflection coefficient is of course 1 and the absorption 0.

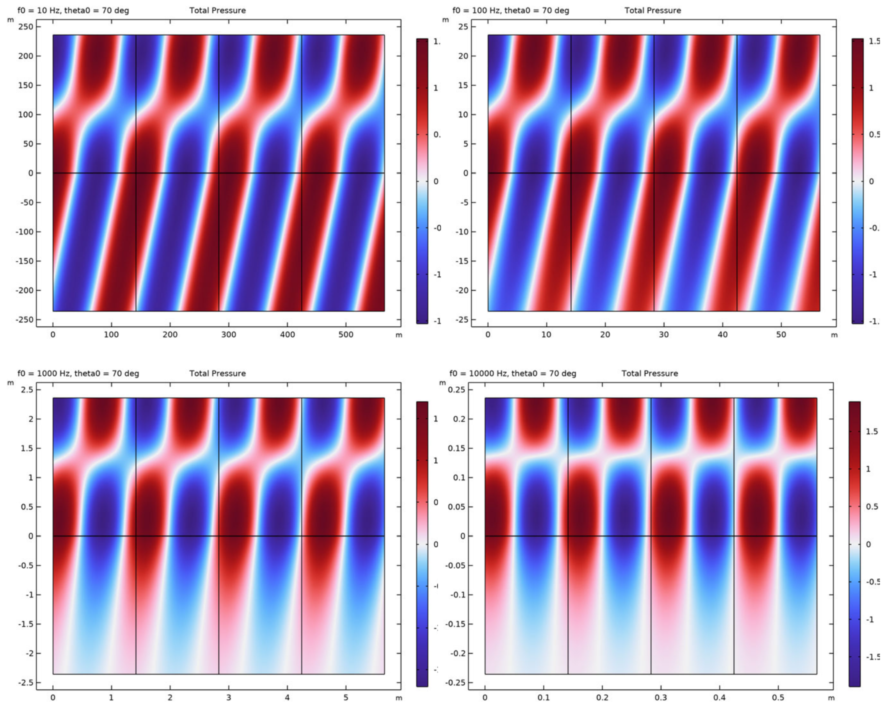


Figure 5: Total pressure distribution for an incidence angle of 70° for 10 Hz (top left), 100 Hz (top right), 1 kHz (bottom left), and 10 kHz (bottom right). Note that the computational domain size (geometry) scales with the wavelength.

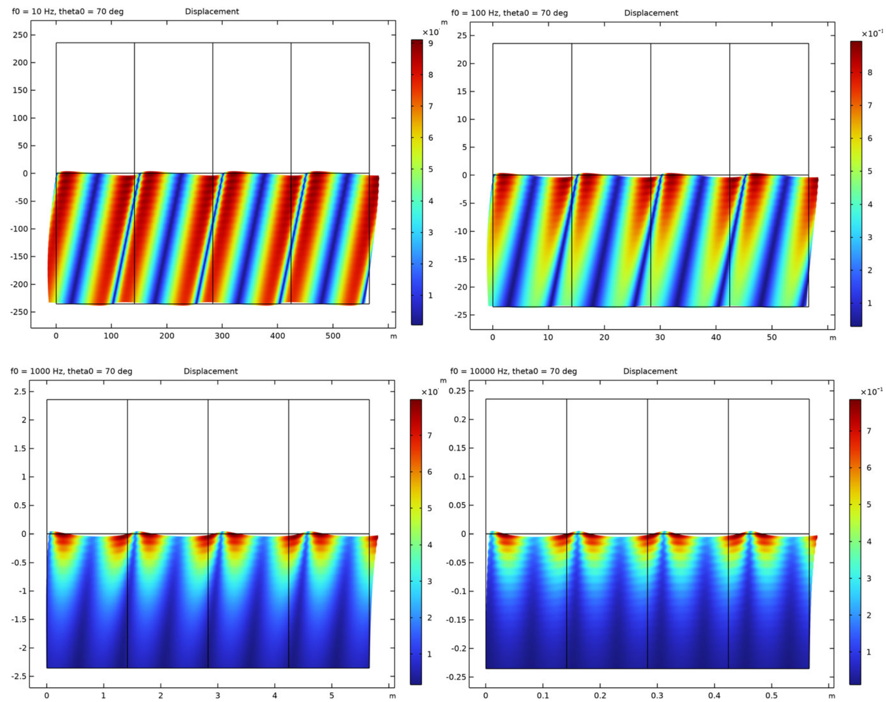


Figure 6: Displacement amplitude with deformation of the porous matrix for an incidence angle of 70° for 10 Hz (top left), 100 Hz (top right), 1 kHz (bottom left), and 10 kHz (bottom right). Note that the computational domain size (geometry) scales with the wavelength.

Notes About the COMSOL Implementation

- In order for the model to be numerically well behaved for all frequencies studied, the geometry of the periodic unit cell is set to depend on the wavelength `lambda0`. This means that the geometry depends on the frequency. To model this, a parametric sweep which updates the geometry is added. The sweep parameter is the frequency `f0`, which is then entered as the study frequency in the Frequency Domain study step.
- The PMLs do not automatically compensate for the oblique angle of incidence of the waves. It is recommended to use a PML stretching factor of `1/cos(theta)` in situations with plane waves at an oblique incidence.

- Rational PML stretching is unsuitable for evanescent wave components such as the fast wave above the critical angle of total internal reflection, so it is recommended to use polynomial stretching in the sediment domains.
- The periodic plots shown in Figure 5 and Figure 6 are set up using a Periodic 2D dataset. Using the **Floquet-Bloch periodicity** option and entering the wave vector components `k0x` and `k0y` with this option, the correct phase is automatically added to the underlying solution variables.
- From the material parameters in Table 1, it is seen that the Biot-Willis coefficient α_B is close to 1. This means that the porous matrix is in a so-called limp configuration. At the other extreme, when α_B is close to the value of porosity ε_P, the configuration is named rigid (the skeleton is not moving). When a porous material is in one of these states it is possible to use an equivalent fluid model from the Poroacoustics domain feature (found in the Pressure Acoustics, Frequency Domain interface) to model the porous material. Use, for example, the Johnson-Champoux-Allard (limp frame option) for the case studied here. The advantage of using this model is that it is much less computationally expensive than the full Biot model. An equivalent fluid model generates or mimics the losses in the porous domain and therefore predicts the reflection and absorption coefficient. However, it does not account for the details of the movement of the skeleton and the propagation of the three different types of waves. Moreover, as soon as a porous material is neither limp nor rigid, the full Biot model of the Poroelastic Waves interface is necessary.

Reference

1. R.D. Stoll and T.-K. Kan, “Reflections of Acoustic Waves at a Water-Sediment Interface”, *J. Acoust. Soc. Am.*, vol. 70, no. 1, pp. 149–156, 1981.

Application Library path: `Acoustics_Module/Underwater_Acoustics/reflections_water_sediment`

Modeling Instructions

Start by adding the **Acoustic-Poroelastic Waves Interaction** multiphysics interface. The interface consists of the **Pressure Acoustics, Frequency Domain** interface, the **Poroelastic Waves** interface, and the necessary **Multiphysics** coupling.

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>Acoustic-Structure Interaction>Acoustic-Poroelastic Waves Interaction**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Frequency Domain**.
6 Click **Done**.

GEOMETRY 1

First, load the material parameters from a file. The list contains definitions of material parameters (see Table 1).

GLOBAL DEFINITIONS

Parameters - Material

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Parameters - Material` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `reflections_water_sediment_material_parameters.txt`.

Parameters - Model

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

   This second parameter group loaded from a file contains geometry lengths, and the two model parameters (frequency `f0` and angle of incidence `theta0`).
2 In the **Settings** window for **Parameters**, type `Parameters - Model` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `reflections_water_sediment_model_parameters.txt`.

Build the geometry, a sketch is given in Figure 1.

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`.
5 Click to expand the **Layers** section. Clear the **Layers on bottom** check box.
6 Select the **Layers on top** check box.
7 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Hpml

8 Click **Build Selected**.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `W`.
4 In the **Height** text field, type `H`.
5 Locate the **Position** section. In the **y** text field, type `-H`.
6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Hpml

7 Click **Build All Objects**.
8 Click the **Zoom Extents** button in the **Graphics** toolbar.

Under the **Definitions** node add variables as well as three nonlocal couplings. The couplings are used to pick up variable values. One samples a point in the water domain and the others evaluates the average/integral over the water-sediment interface.

DEFINITIONS

Variables 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `reflections_water_sediment_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 4 only.

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 6 only.

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 6 only.

Now, create selections to be used when setting up the physics.

Porous Domain

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

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

3 Select Domains 1 and 2 only.

Water Domain

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

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

3 Select Domains 3 and 4 only.

Proceed to set up the physics and boundary conditions of the problem. The condition at the water-sediment interface is defined under the **Multiphysics** node.

The properties of the porous material are edited after adding the porous material.

POROELASTIC WAVES (PELW)

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

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

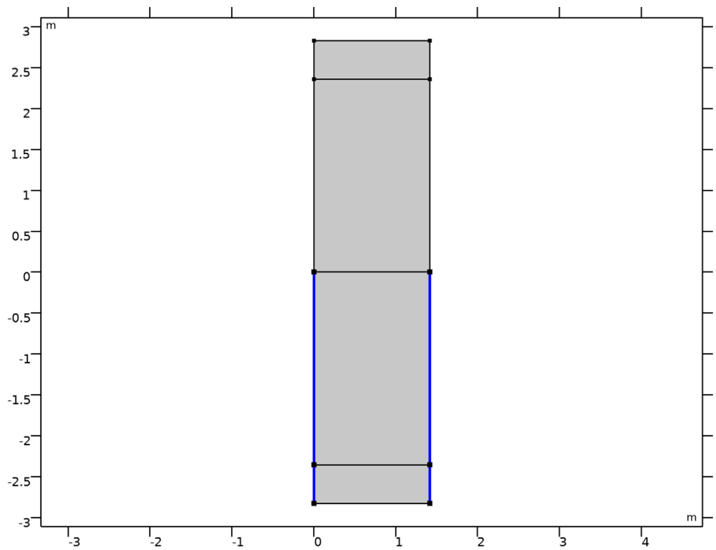
3 From the **Selection** list, choose **Porous Domain**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

Periodic Condition 1

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

2 Select Boundaries 1, 3, 10, and 11 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

k0x	X
k0y	Y

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

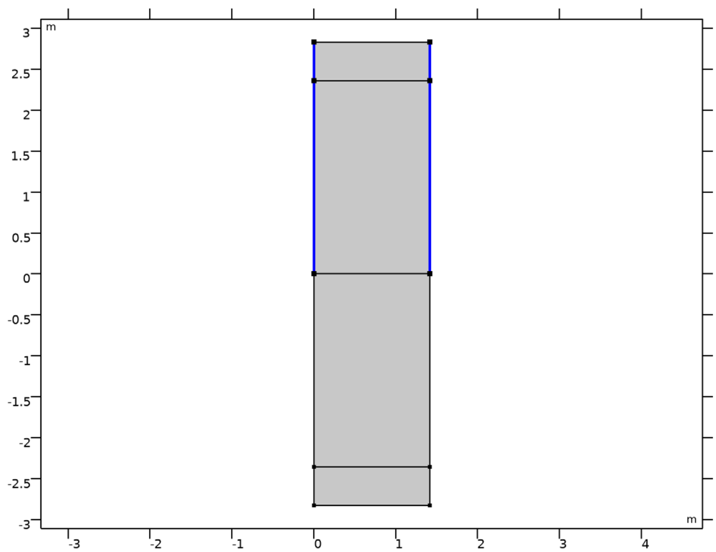
3 From the **Selection** list, choose **Water Domain**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

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

Periodic Condition 1

1 Select Boundaries 5, 7, 12, and 13 only.



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

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

4 Specify the $\mathbf{k}_\mathrm{F}$ vector as

k0x	x
k0y	y

Background Pressure Field 1

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

2 In the **Settings** window for **Background Pressure Field**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Water Domain**.

4 Locate the **Background Pressure Field** section. 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** check box.

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

MATERIALS

Water

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**, twice, to create two new blank materials.

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

Sediment

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 2 (mat2)**.

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

Now, go to the **Poroelastic Material** node and specify the porous matrix material parameters to be taken from the **Sediment** material that you just created. When done, return to the material, where it is now evident which parameters to define.

POROELASTIC WAVES (PELW)

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 **Porous elastic material** list, choose **Sediment (mat2)**.

4 Locate the **Fluid Properties** section. From the **Viscosity model** list, choose **Biot's high frequency range**.

5 From the **Specify** list, choose **Characteristic pore size**.

6 In the a text field, type `a`.

MATERIALS

Water (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water (mat1)**.

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
Density	rho	`rhoF`	kg/m³	Basic
Speed of sound	c	`c0`	m/s	Basic
Dynamic viscosity	mu	`muF`	Pa·s	Basic
Compressibility of fluid	chif	`1/Kf`	1/Pa	Basic

Sediment (mat2)

1 In the **Model Builder** window, click **Sediment (mat2)**.

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
Bulk modulus	K	`Kbc`	N/m²	Bulk modulus and shear modulus
Shear modulus	G	`Gc`	N/m²	Bulk modulus and shear modulus
Density	rho	`rhod`	kg/m³	Basic
Porosity	epsilon	`epsilonP`	1	Basic
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	`kappaP`	m²	Basic
Biot-Willis coefficient	alphaB	`alphaB0`	1	Poroelastic material
Tortuosity factor	tau	`tau0`	1	Poroacoustics model

MULTIPHYSICS

Acoustic-Porous Boundary 1 (apb1)

Now is a good time to inspect the **Multiphysics** node and look at the multiphysics coupling. Click on **Acoustic-Porous Boundary 1**. You can see that it is active on the interface between the water domain and the porous sediment.

The last step before building the mesh is to set up and define the perfectly matched layers (PMLs). Define one for the poroelastic domain and another for the water domain.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 Select Domain 1 only.

Switch the **Physics** to **Poroelastic Waves**. The polynomial stretching is generally most appropriate when there is a mix of different wave types in the model, as it is the case for the **Poroelastic Waves** domain.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Physics** list, choose **Poroelastic Waves (pelw)**.

The PMLs do not compensate for the oblique angle of incidence of the waves. Typically, the PML stretching factor should be `1/cos(theta)` in situations with plane waves at oblique incidence.

5 In the **PML scaling factor** text field, type `if(theta0==90[deg],1e6,1/cos(theta0))`.

Perfectly Matched Layer 2 (pml2)

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

2 Select Domain 4 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

The PMLs do not compensate for the oblique angle of incidence of the waves. Typically, the PML stretching factor should be `1/cos(theta)` in situations with plane waves at oblique incidence.

5 In the **PML scaling factor** text field, type `if(theta0==90[deg],1e6,1/cos(theta0))`.

Keep the default setting in the **Physics** list, that is **Pressure Acoustics, Frequency Domain (acpr)**.

Build the mesh by following the instructions below. When modeling systems with different materials it is important to have a mesh that solves for the slowest wave traveling through each of the materials. In this particular example, where the background field could travel in any direction, it is important to have enough elements in the PML domain in both directions. It is also important for models with periodic conditions to have the same mesh at source and destination. Ideally, a mesh sensitivity analysis should be made for models of this kind. In this model, the mesh is set up manually. Proceed by directly adding a mapped mesh in order to reduce the computational cost of the analysis.

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 Click the **Custom** button.

In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. In this model we use 5 elements per wavelength. Use the wavelength in water in the water domain.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lam0/5`.

5 In the **Minimum element size** text field, type `lam0/10`.

Mapped 1

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

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

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

4 Click in the **Graphics** window and then press Ctrl+A to select all domains.

Size 1

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

Use the wavelength of the slowest wave in the poroelastic waves domains.

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

3 From the **Selection** list, choose **Porous Domain**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Maximum element size** check box. In the associated text field, type `cs_poro/f0/5`.

7 Select the **Minimum element size** check box. In the associated text field, type `cs_poro/f0/10`.

Add a distribution to make sure that the PML domains have enough number of elements to resolve all the waves.

Distribution 1

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

2 Select Boundaries 1 and 7 only.

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

4 In the **Number of elements** text field, type `30`.

Proceed and add a single boundary layer mesh at the interface between the water and the sediment. This layer will ensure a good evaluation of the gradient involved in the intensity computation.

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 From the **Selection** list, choose **Water Domain**.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

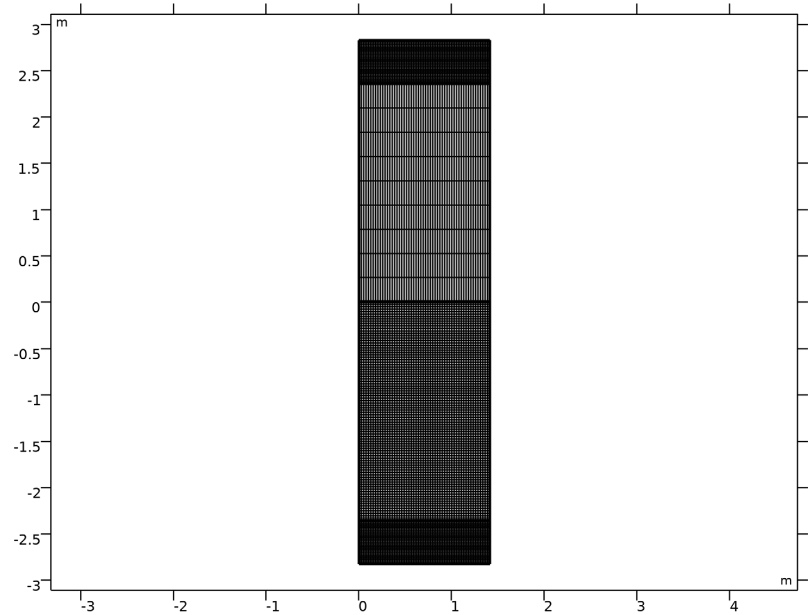
2 Select Boundary 6 only.

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**.

The finalized mesh should look like that in the figure below.



STUDY 1 - DETAILED ANGULAR SWEEP

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

2 In the **Settings** window for **Study**, type `Study 1 - Detailed Angular Sweep` in the **Label** text field.

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

Step 1: Frequency Domain

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

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

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

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

5 From the **Sweep type** list, choose **All combinations**.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Angle of incidence)	`range(0,10[deg],50[deg]) range(51[deg],1[deg], 89[deg]) 89.5[deg]`	`deg`

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Driving frequency)	10 100 1000 10000	Hz

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

RESULTS

Add a selection to the dataset to only plot results in the physical domains (not in the PMLs). You can study the behavior of the PMLs by removing this selection again or creating a separate dataset without the selection.

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

Study 1 - Detailed Angular Sweep/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1 - Detailed Angular Sweep/Parametric Solutions 1 (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 Select Domains 2 and 3 only.

Now, also add an array dataset in order to easily plot the result in the periodic structure.

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 **Data** section.

3 From the **Dataset** list, choose **Study 1 - Detailed Angular Sweep/ Parametric Solutions 1 (sol2)**.
4 Locate the **Array Size** section. 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 `k0x`.
7 In the **Y** text field, type `k0y`.

Now, create four different 2D plots representing the incident pressure (background pressure), the reflected pressure (scattered pressure), the total pressure, and the displacement in the porous matrix.

Incident Pressure

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Incident Pressure` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Incident Pressure`.
5 In the **Parameter indicator** text field, type `f0 = eval(freq) Hz, theta0 = eval(theta0, deg) deg`.

Surface 1

1 Right-click **Incident Pressure** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `acpr.p_b`.
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**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Incident Pressure** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
3 In the **X-component** text field, type `k0x`.
4 In the **Y-component** text field, type `k0y`.

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

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Incident Pressure

1 In the **Model Builder** window, click **Incident Pressure**.

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

3 From the **Dataset** list, choose **Study 1 - Detailed Angular Sweep/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (freq (Hz),theta0 (deg))** list, choose **4: freq=10000 Hz, theta0=30 deg**.

Select any of the desired combinations of the frequency `f0` and the angle of incidence `theta0` and plot to have a look at the results. You may need to use the **Zoom Extent** tool as the geometry depends on the frequency.

5 In the **Incident Pressure** toolbar, click **Plot**.

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

Reflected Pressure

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Detailed Angular Sweep/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (freq (Hz),theta0 (deg))** list, choose **4: freq=10000 Hz, theta0=30 deg**.

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

6 In the **Title** text area, type `Reflected Pressure`.

7 In the **Parameter indicator** text field, type `f0 = eval(freq) Hz, theta0 = eval(theta0, deg) deg`.

Surface 1

1 Right-click **Reflected Pressure** and choose **Surface**.

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

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

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 **Reflected Pressure** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, right-click **Reflected Pressure** and choose **Surface**.

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

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

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

5 In the **Reflected Pressure** toolbar, click **Plot**.

Note that this gives the scattered field in the water domain while it is equal to the total acoustic field in the porous domain.

In the next figure, plot the total pressure (water and porous domains). Use the periodic dataset to display the periodic nature of the modeled system.

The Floquet-periodic nature of the solution is recovered with the selected **Floquet-Bloch periodicity** option in the **Array 2D** dataset. With this option the correct phase is added to the underlying solution variables.

Total Pressure

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

2 In the **Settings** window for **2D Plot Group**, type `Total Pressure` 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),theta0 (deg))** list, choose **4: freq=10000 Hz, theta0=30 deg**.

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

6 In the **Title** text area, type `Total Pressure`.

7 In the **Parameter indicator** text field, type `f0 = eval(freq) Hz, theta0 = eval(theta0, deg) deg`.

Surface 1

1 Right-click **Total Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Multiphysics>apb1.p_t - Total acoustic pressure - Pa**.

The multiphysics pressure variable is defined as `acpr.p_t` in the water domain and `pelw.p_t` in the sediment domain.

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

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

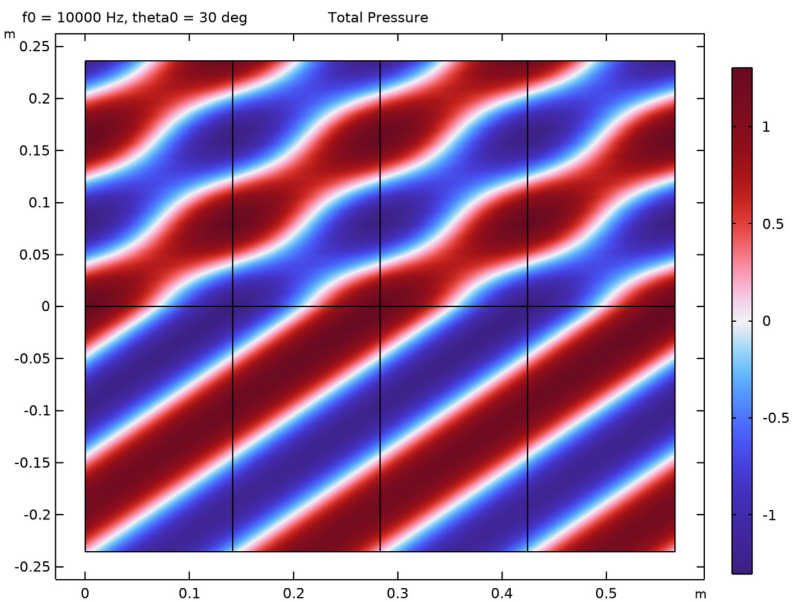
5 Click **OK**.

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

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

8 In the **Total Pressure** toolbar, click **Plot**.

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



Now, plot the displacement of the porous matrix. Again use the periodic dataset to represent the periodic nature of the modeled system, just as for the total pressure.

Displacement

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

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

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

4 In the **Title** text area, type `Displacement`.

5 In the **Parameter indicator** text field, type `f0 = eval(freq) Hz, theta0 = eval(theta0, deg) deg`.

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

7 From the **Parameter value (freq (Hz),theta0 (deg))** list, choose **4: freq=10000 Hz, theta0=30 deg**.

Surface 1

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

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

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

Deformation 1

1 Right-click **Surface 1** and choose **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 Locate the **Scale** section.

6 Select the **Scale factor** check box. In the associated text field, type `2e9`.

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

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

The total pressure and the displacement are depicted in Figure 5 and Figure 6 for several combinations of the driving frequency and the angle of incidence.

Wave Speed

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

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

Surface 1

1 Right-click **Wave Speed** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Pressure Acoustics, Frequency Domain>Material properties>acpr.c - Speed of sound - m/s**.

3 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

4 In the **Wave Speed** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, right-click **Wave Speed** and choose **Surface**.

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

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

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Filter 1

1 Right-click **Surface 2** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(y>-H/3)`.

4 In the **Wave Speed** toolbar, click **Plot**.

Surface 3

1 In the **Model Builder** window, under **Results>Wave Speed** right-click **Surface 2** and choose **Duplicate**.

2 In the **Model Builder** window, click **Surface 3**.

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

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

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `(y<-H/3)*(y>-2*H/3)`.

Surface 4

1 In the **Model Builder** window, under **Results>Wave Speed** right-click **Surface 2** and choose **Duplicate**.

2 In the **Model Builder** window, click **Surface 4**.

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

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

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `(y<-2*H/3)`.

Annotation 1

1 In the **Model Builder** window, right-click **Wave Speed** and choose **Annotation**.

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

3 In the **Text** text field, type `Speed of sound = eval(acpr.c) m/s`.

4 Locate the **Position** section. In the **X** text field, type `W`.

5 In the **Y** text field, type `H/2`.

Annotation 2

1 Right-click **Wave Speed** and choose **Annotation**.

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

3 In the **Text** text field, type `Speed of fast pressure wave = eval(real(pelw.cp_fast)) m/s`.

4 Locate the **Position** section. In the **X** text field, type `W`.

5 In the **Y** text field, type `-H/6`.

Annotation 3

1 Right-click **Wave Speed** and choose **Annotation**.

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

3 In the **Text** text field, type `Speed of slow pressure wave = eval(real(pelw.cp_slow)) m/s`.

4 Locate the **Position** section. In the **X** text field, type `W`.

5 In the **Y** text field, type `-3*H/6`.

Annotation 4

1 Right-click **Wave Speed** and choose **Annotation**.

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

3 In the **Text** text field, type `Speed of shear-wave = eval(real(pelw.cs_poro)) m/s`.

4 Locate the **Position** section. In the **X** text field, type `W`.

5 In the **Y** text field, type `-5*H/6`.

6 In the **Wave Speed** toolbar, click **Plot**.

Create Figure 2 from Ref. 1 by plotting the absolute value of the reflection coefficient as function of the angle of incidence for four values of the frequency `f0`.

Reflection Coefficient vs. Angle of Incidence

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

2 In the **Settings** window for **1D Plot Group**, type `Reflection Coefficient vs. Angle of Incidence` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Detailed Angular Sweep/ Parametric Solutions 1 (sol2)**.

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

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Reflection coefficient: |R|`.

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

8 In the **x minimum** text field, type `-2`.

9 In the **x maximum** text field, type `92`.

10 In the **y minimum** text field, type `-0.05`.

11 In the **y maximum** text field, type `1.05`.

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

Global 1

1 Right-click **Reflection Coefficient vs. Angle of Incidence** 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
absR	1	Absolute and averaged value of the reflection coefficient

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

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `theta0`.

7 From the **Unit** list, choose **°**.

8 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

9 In the **Reflection Coefficient vs. Angle of Incidence** toolbar, click **Plot**.

The plot should look like the one in Figure 2.

Next, create a second study to sweep over the frequency for a given number of angles of incidence.

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 - DETAILED FREQUENCY SWEEP

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

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

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Detailed Frequency Sweep** click **Step 1: Frequency Domain**.

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

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

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

5 From the **Sweep type** list, choose **All combinations**.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Angle of incidence)	`30[deg] 40[deg] 50[deg] 60[deg] 70[deg] 80[deg]`	`deg`

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Driving frequency)	`10^{range(1,4/14,5)}`	Hz

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

RESULTS

Create Figure 3 from Ref. 1 by plotting the absolute value of the reflection coefficient as function of the frequency `f0` for six values of the angle of incidence.

Reflection Coefficient vs. Driving Frequency

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

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

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

6 In the **x minimum** text field, type `8.5`.

7 In the **x maximum** text field, type `117171`.

8 In the **y minimum** text field, type `-0.05`.

9 In the **y maximum** text field, type `1.05`.

10 Select the **x-axis log scale** check box.

11 In the **Reflection Coefficient vs. Driving Frequency** toolbar, click **Plot**.

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

Global 1

1 Right-click **Reflection Coefficient vs. Driving Frequency** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 2 - Detailed Frequency Sweep/ Parametric Solutions 2 (sol8)**.

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

5 In the **Parameter indices (1-6)** text field, type `1`.

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

Expression	Unit	Description
`absR`	`1`	`Absolute and averaged value of the reflection coefficient`

7 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

8 From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `f0`.

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

11 In the table, enter the following settings:

Legends
`theta0 = 30[deg]`

Global 2

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

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

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

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

Legends
`theta0 = 40[deg]`

Global 3

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

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

3 In the **Parameter indices (1-6)** text field, type `3`.

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

Legends
`theta0 = 50[deg]`

Global 4

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

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

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

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

Legends
`theta0 = 60[deg]`

Global 5

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

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

3 In the **Parameter indices (1-6)** text field, type `5`.

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

Legends
`theta0 = 70[deg]`

Global 6

1 Right-click **Global 5** and choose **Duplicate**.

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

3 In the **Parameter indices (1-6)** text field, type `6`.

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

Legends
`theta0 = 80[deg]`

5 In the **Reflection Coefficient vs. Driving Frequency** toolbar, click **Plot**.

The plot should look like the one in Figure 3.

Finally, plot the absorption coefficient as function of the frequency `f0` for six values of the angle of incidence. Simply duplicate the plot with the absorption coefficient, change `absR` to `alpha` in the plots, and rename titles where necessary.

Absorption Coefficient vs. Angle of Incidence

1 In the **Model Builder** window, right-click **Reflection Coefficient vs. Angle of Incidence** and choose **Duplicate**.

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

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Absorption coefficient: \alpha`.

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

Global 1

1 In the **Model Builder** window, expand the **Absorption Coefficient vs. Angle of Incidence** node, then click **Global 1**.

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 In the **Absorption Coefficient vs. Angle of Incidence** toolbar, click **Plot**.

The plot should look like the one in Figure 4.