Created in COMSOL Multiphysics 5.5

Isotropic-Anisotropic Sample: Elastic Wave Propagation

Introduction

This tutorial shows how to use the Elastic Waves, Time Explicit physics interface to model the propagation of elastic waves in heterogeneous anisotropic linear elastic media. The case where the material properties differ from one subdomain to another requires an special handling. Namely, the Material Discontinuity interior boundary condition (or the Continuity pair condition for assemblies) is to be explicitly imposed upon the interface between such subdomains.

In addition, the model demonstrates how to retrieve the displacements at the points of interest from the solution that contains the structural velocity and the strain.

Model Definition

A two-dimensional space is occupied by two linear elastic materials: an anisotropic material to the left of the axis $x = 0$ and an isotropic material to the right. The material properties are given in Table 1 (kg/m^3 for the density and 10^{10} Pa for the stiffness). The other material parameters are $c_{33} = c_{22}$, $c_{13} = c_{23} = c_{12}$, and $c_{44} = c_{55} = c_{66}$.

TABLE 1: MATERIAL PROPERTIES.

	ρ	c_{11}	c_{12}	c_{22}	c_{66}
isotropic	7100	16.5	8.85	16.5	3.96
anisotropic	7100	16.5	5.00	6.2	3.96

The elastic waves are excited by a point force applied in the y direction at the point $x_0 = -2$ cm, $y_0 = 0$ cm. The source distribution in time is given by a Ricker (also known as Mexican hat) wavelet depicted in Figure 1. It has the dominant frequency $f_0 = 170$ kHz, the time delay $t_0 = 6$ µs, and the amplitude of 10^{13} N.

The elastic 2D space is modeled as a square with the side length of $2L$, $L = 20$ cm and the center located at (0, 0). The subdomains with the anisotropic and the isotropic properties lie to the left and to the right of the line $x = 0$, respectively. Four listening points are put to record the seismograms (displacement at points) at the coordinates (−10.5 cm, −8 cm), (−3.5 cm, −8 cm), (−1.0 cm, −8 cm), and (10.5 cm, −8 cm). The model geometry layout is shown in Figure 2.

The propagation of the elastic waves in infinite space is modeled by imposing the Absorbing Layers from all four sides of the square. Combined with the Low-Reflecting Boundary condition that is imposed upon the outer boundary of the square, the Absorbing Layers ensure the reduction of the spuriously reflected waves in the physical domain.

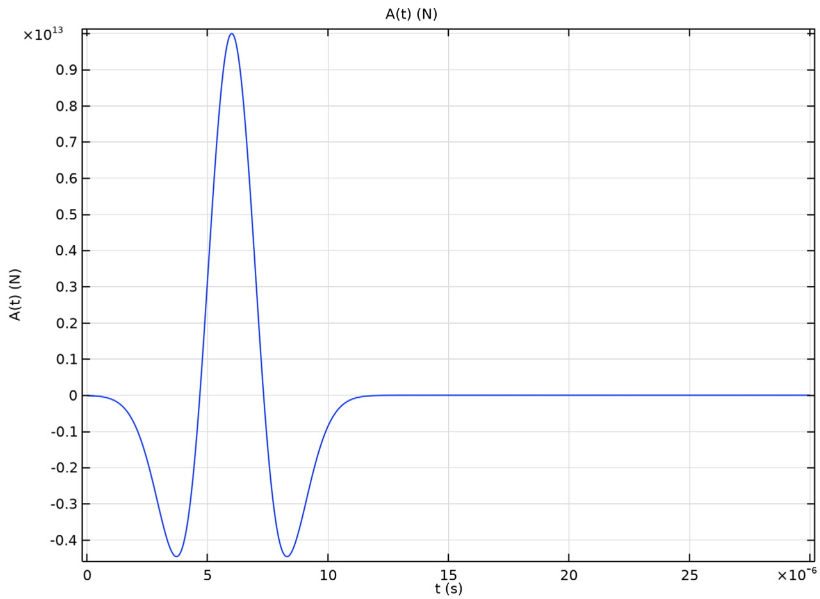


Figure 1: A Ricker wavelet that defines the source distribution in time.

The Elastic Waves, Time Explicit interface solves for the structural velocity and the strain. The displacements can be retrieved from the velocities by solving the corresponding ordinary differential equation

$$\frac{du}{dt} = v, \quad u(t_0) = u_0$$

In this model, this ODE is only solved at the probe points.

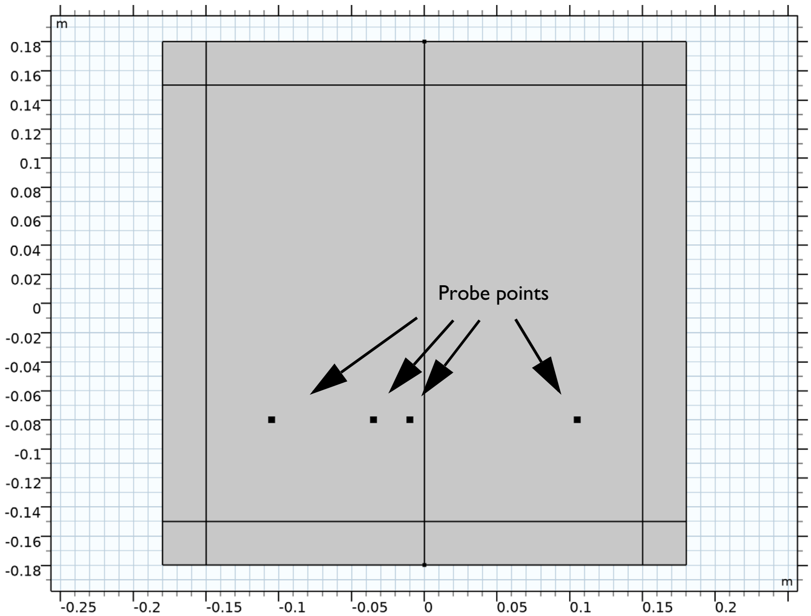


Figure 2: Model geometry layout.

Results and Discussion

Figure 3 shows the quantitative evolution of the waves propagating from the source at $t = 30$, 60, and 90 μs. One can see the round profiles of the longitudinal and the shear waves in the isotropic material on the right side. At the same time, the quasilongitudinal and the quasishear waves in the anisotropic material on the right side have the elliptic and the conical profiles, respectively. Note that the absorbing layer domains are not shown in Figure 3.

Figure 4 shows the profiles of the vertical displacement (seismograms) retrieved from the computed velocities at the probe points. The results have a good agreement in their shapes with those given in Ref. 1.

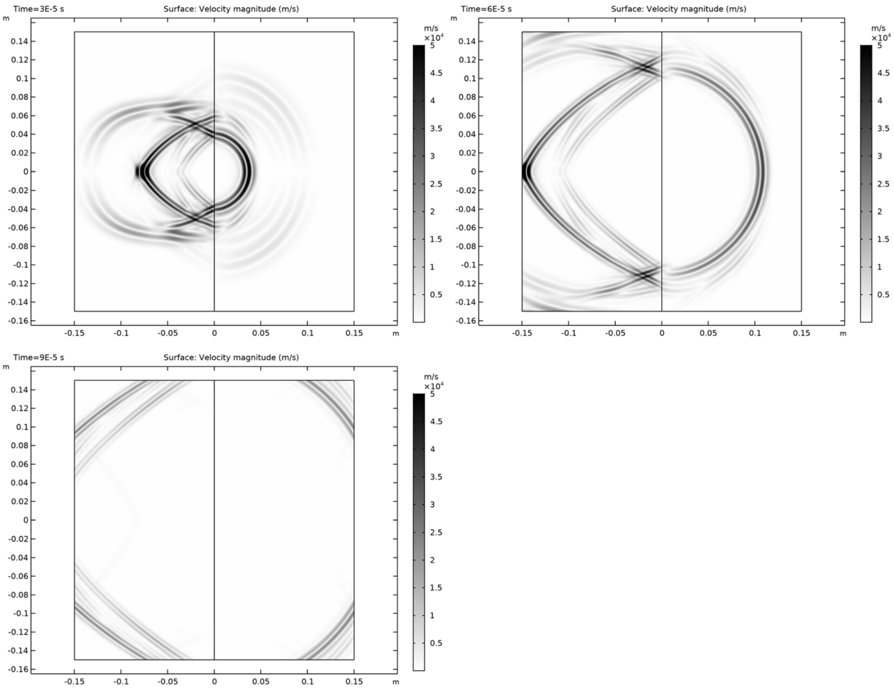


Figure 3: Velocity magnitude profiles at 30, 60, and 90 µs*.*

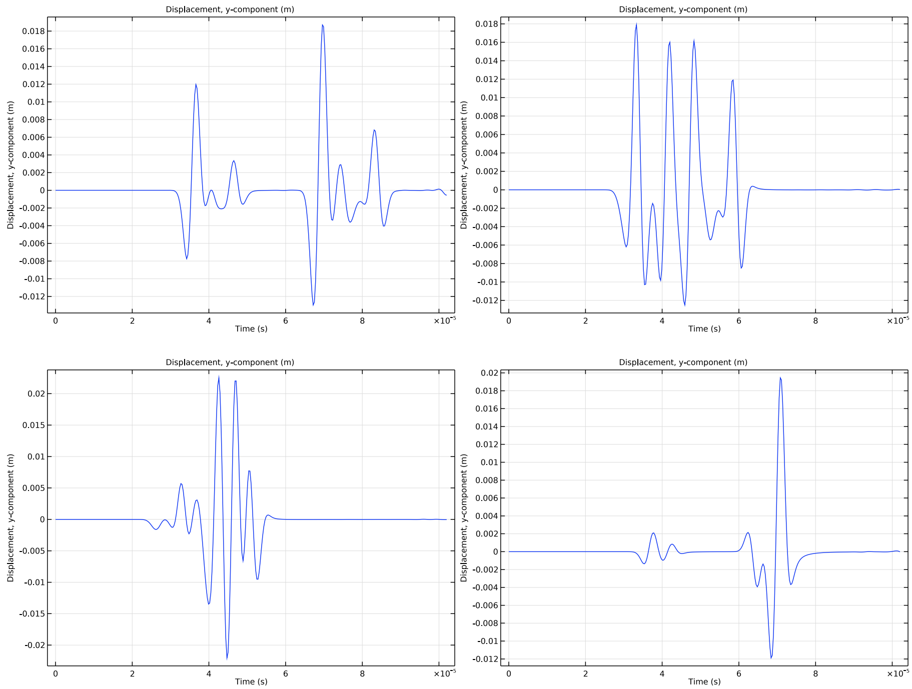


Figure 4: Seismograms at the probe points.

Notes About the COMSOL Implementation

The force load at the point (x_0, y_0) is modeled as a Body Load with a Total force applied to the physical domain. In theory, the load applied on the domain level must have proportional to the Dirac delta to be equivalent to a point load. That is,

$$F_{\mathrm{pnt}}(t) = F_{\mathrm{vol}}(t)\delta(x - x_0, y - y_0)$$

One of the Dirac delta representations reads

$$\delta(x - x_0) = \lim_{a \to 0} \frac{1}{\sqrt{\pi a^2}} e^{-\left(\frac{x - x_0}{a}\right)^2}$$

This fact is used to approximate the point source by a domain source that has the form of a product of the temporal part and the spatial part. Here, the former is the Ricker wavelet depicted in Figure 1 and the latter is a Gaussian pulse with a relatively small extent a.

The default approximation order used in the Elastic Wave, Time Explicit interface is quartic. In this case, the maximum mesh element size that resolves the minimum wavelength λ_{min} must not exceed $\lambda_{min}/1.5$. The minimum wavelength is defined by the frequency content of the source, which is the Ricker wavelet here. Its frequency domain representation reads

$$F(f) = \frac{2}{\sqrt{\pi}}\frac{f^2}{f_0^3}e^{-\frac{f^2}{f_0^2}}$$

where f_0 is the dominant frequency. Although the upper cutoff frequency of the wavelet is $3f_0$, λ_{min} is chosen to correspond to $2f_0$ in this model. This is a compromise that on one hand accounts for the greatest part of the source energy, and on the other hand does not result in a too fine mesh that would slow down the simulation and tremendously increase the model file size.

Reference

1. J. de la Puente, M. Käser, M. Dumbser and H. Igel, "An arbitrary high-order discontinuous Galerkin method for elastic waves on unstructured meshes – IV. Anisotropy", *Geophys. J. Int.*, vol. 169, issue 3, 2007.

Application Library path: `Acoustics_Module/Elastic_Waves/isotropic_anisotropic_sample`

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>Elastic Waves, Time Explicit (elte)**.
3 Click **Add**.

4 Click **Study**.

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

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 `isotropic_anisotropic_sample_parameters.txt`.

Create the source time function given by a Ricker wavelet.

DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `A` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `1e13*(1 - 2*pi^2*f0^2*(t - t0)^2)*exp(-pi^2*f0^2*(t - t0)^2)`.

4 In the **Arguments** text field, type `t`.

5 Locate the **Units** section. In the **Arguments** text field, type `s`.

6 In the **Function** text field, type `N`.

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

Argument	Lower limit	Upper limit
t	0	5*t0

8 Click **Plot**.

The signal should look like the one in Figure 1

Now, create a two-dimensional Gaussian function that will be used to approximate the Dirac delta at (x_0, y_0).

Analytic 2 (an2)

1 In the **Home** toolbar, click **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `G` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `1/sqrt(pi*dS)*exp(-((x - x0)^2 + (y - y0)^2)/dS)`.

4 In the **Arguments** text field, type `x, y`.

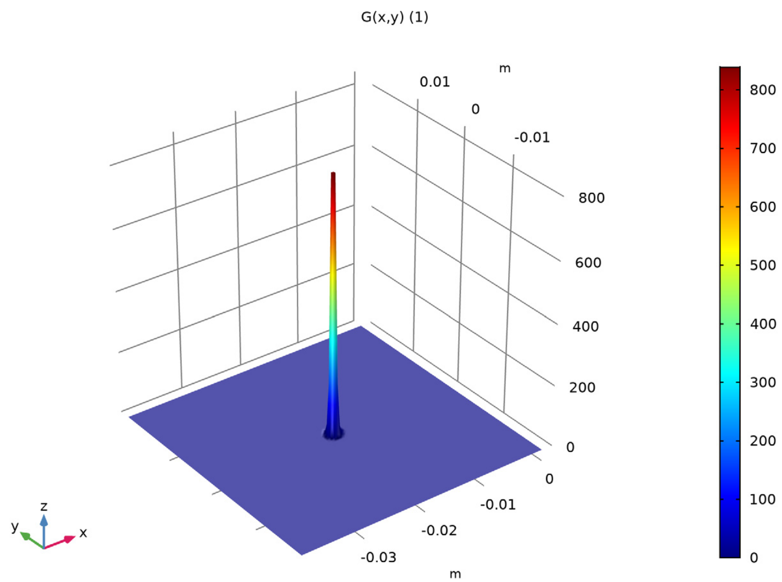
5 Locate the **Units** section. In the **Arguments** text field, type `m`.

6 In the **Function** text field, type `1`.

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

Argument	Lower limit	Upper limit
x	-0.04	0
y	-0.02	0.02

8 Click **Plot**.



GEOMETRY 1

Square 1 (sq1)

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

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

3 In the **Side length** text field, type `2*L`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 Click to expand the **Layers** section. Select the **Layers to the left** check box.

6 In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`L/6`

7 Select the **Layers to the right** check box.

8 Select the **Layers on top** check box.

Create a line segment that separates the subdomains with isotropic and anisotropic materials.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

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

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **y** text field, type `L`.

Create four probe points for calculating the seismograms.

Point 1 (pt1)

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

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

3 In the **x** text field, type `-10.5[cm] -3.5[cm] -1[cm] 10.5[cm]`.

4 In the **y** text field, type `-8[cm] -8[cm] -8[cm] -8[cm]`.

5 Click **Build All Objects**.

   The geometry should look like the one in Figure 2

DEFINITIONS

Explicit 1

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Points 9–11 and 16 only.

5 In the **Label** text field, type `Probe Points`.

Now, set up the absorbing layers (sponge layers) used to truncate the computational domain.

Absorbing Layer 1 (ab1)

1 In the **Definitions** toolbar, click **Absorbing Layer**.

2 Select Domains 1–4, 6, 7, and 9–12 only.

Setting up the materials is simpler if the correct material model is selected in the physics. Here we want to use an anisotropic material.

ELASTIC WAVES, TIME EXPLICIT (ELTE)

Elastic Waves, Time Explicit Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Elastic Waves, Time Explicit (elte)** click **Elastic Waves, Time Explicit Model 1**.

2 In the **Settings** window for **Elastic Waves, Time Explicit Model**, locate the **Linear Elastic Material** section.

3 From the **Solid model** list, choose **Anisotropic**.

MATERIALS

Material 1 (mat1)

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 `Material Isotropic` in the **Label** text field.

3 Select Domains 7–12 only.

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

Property	Variable	Value	Unit	Property group
Elasticity matrix, Voigt notation	{DVo11, DVo12, DVo22, DVo13, DVo23, DVo33, DVo14, DVo24, DVo34, DVo44, DVo15, DVo25, DVo35, DVo45, DVo55, DVo16, DVo26, DVo36, DVo46, DVo56, DVo66} ; DVoij = DVoji	`{16.5e10, 8.58e10, 16.5e10, 8.58e10, 8.58e10, 16.5e10, 0, 0, 0, 3.96e10, 0, 0, 0, 0, 3.96e10, 0, 0, 0, 0, 0, 3.96e10}`	Pa	Anisotropic, Voigt notation
Density	rho	`7100`	kg/m³	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Select Domains 1–6 only.

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

Property	Variable	Value	Unit	Property group
Elasticity matrix, Voigt notation	{DVo11, DVo12, DVo22, DVo13, DVo23, DVo33, DVo14, DVo24, DVo34, DVo44, DVo15, DVo25, DVo35, DVo45, DVo55, DVo16, DVo26, DVo36, DVo46, DVo56, DVo66} ; DVoij = DVoji	`{16.5e10, 5.0e10, 6.2e10, 5.0e10, 5.0e10, 6.2e10, 0,0,0, 3.96e10 ,0,0,0, 0, 3.96e10 ,0,0,0, 0,0, 3.96e10 }`	Pa	Anisotropic, Voigt notation
Density	rho	`7100`	kg/m³	Basic

ELASTIC WAVES, TIME EXPLICIT (ELTE)

Define the force load acting at point (x_0, y_0) in the y-direction.

Body Load 1

1 In the **Physics** toolbar, click **Domains** and choose **Body Load**.

2 Select Domains 5 and 8 only.

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

`0`	x
`G(x,y)*A(t)`	y

Low-Reflecting Boundary 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Low-Reflecting Boundary**.
2. Select Boundaries 1–3, 5, 7, 9, 14, 16, 21, 23, and 28–31 only.

Impose the Material Discontinuity boundary condition upon the interface between the isotropic and anisotropic parts of the computational domain.

Material Discontinuity 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Material Discontinuity**.
2. Select Boundaries 15, 17, and 19 only.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.
3. From the **Sequence type** list, choose **User-controlled mesh**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `cs_an/(2*f0)/1.5`.
5. In the **Minimum element size** text field, type `cs_an/(2*f0)/1.5`.
6. Click **Build All**.

STUDY 1

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1 - ELTE (store full solution)` in the **Label** text field.

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1 - ELTE (store full solution)** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Times** text field, type `range(0, 30[us], 90[us])`.

   This setting saves the solution at 30, 60, and 90 microseconds in the whole computational domain. It only influences the stored solution (and thus the file size).

The internal time steps taken by the solver are automatically controlled by COMSOL to fulfill the appropriate CFL condition.

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

Before plotting the results, process the dataset to display the solution in the physical domains only.

RESULTS

Selection

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

2 Right-click **Study 1 - ELTE (store full solution)/Solution 1 (sol1)** and choose **Selection**.

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

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

5 Select Domains 5 and 8 only.

Velocity Magnitude (elte)

1 In the **Model Builder** window, click **Velocity Magnitude (elte)**.

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

3 From the **Time (s)** list, choose **3E-5**.

Surface 1

1 In the **Model Builder** window, expand the **Velocity Magnitude (elte)** node, then click **Surface 1**.

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

3 From the **Color table** list, choose **GrayScale**.

4 Select the **Reverse color table** check box.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `5e4`.

These settings will sharpen the contrast of the velocity profile.

7 In the **Velocity Magnitude (elte)** toolbar, click **Plot**.

The structural velocity profiles at the time steps chosen in Study 1 are shown in Figure 3.

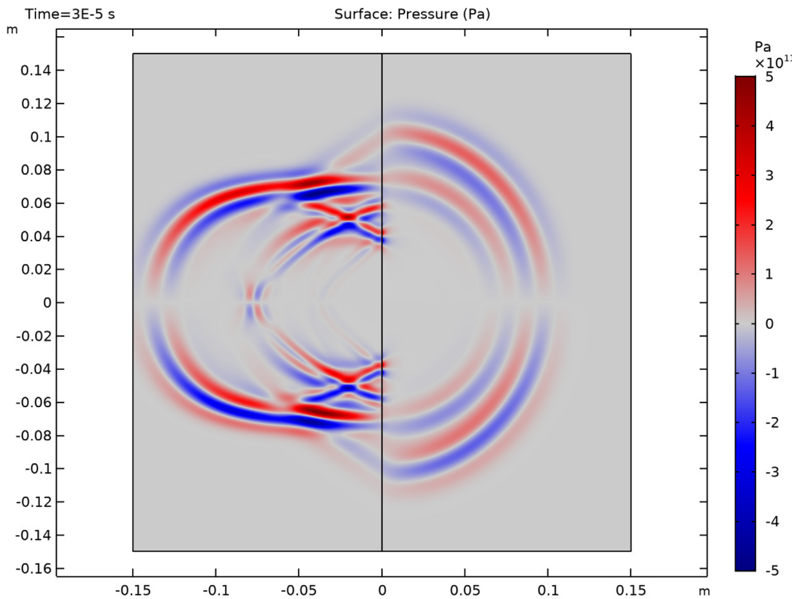
Pressure (elte)

1 In the **Model Builder** window, click **Pressure (elte)**.

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

3 From the **Time (s)** list, choose **3E-5**.

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



Next, inspect the shear and pressure wave speeds in the isotropic material and their equivalents in the anisotropic material.

2D Plot Group 3

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

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

3 Locate the **Data** section. From the **Time (s)** list, choose **3E-5**.

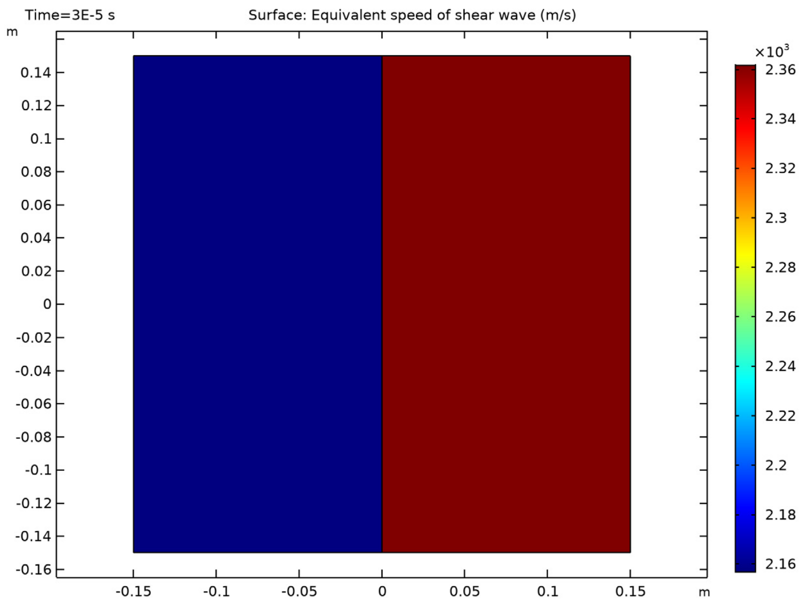
Surface 1

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

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

3 In the **Expression** text field, type `elte.cs`.

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



Apparent Shear Wave Speed 1

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

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

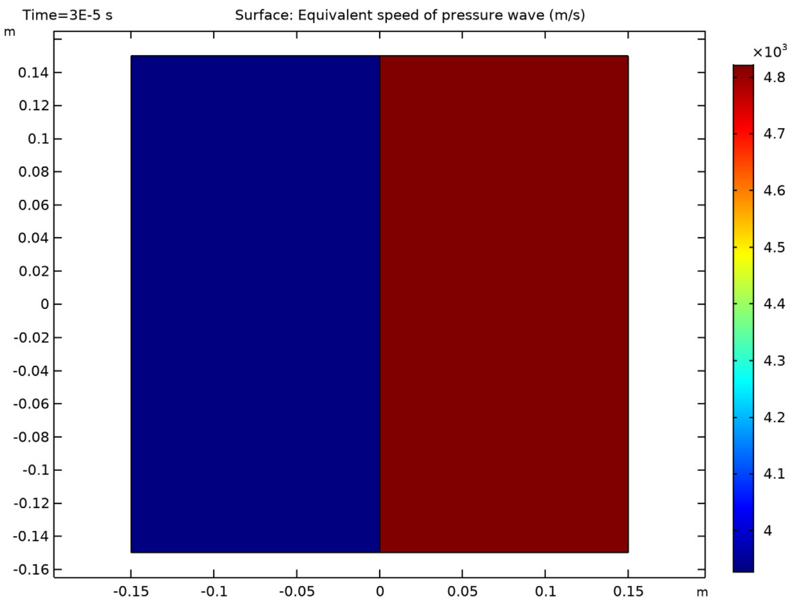
Surface 1

1 In the **Model Builder** window, expand the **Results>Apparent Pressure Wave Speed** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `elte.cp`.

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



Set up an extra study to compute the seismograms at the probe points.

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> Time Dependent**.

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

STUDY 2

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

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 2 - ELTE (store at points)`.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2 - ELTE (store at points)** click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(0, t0/20, 17*t0)`.

4 Click to expand the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

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

6 In the **Add** dialog box, select **Probe Points** in the **Selections** list.

7 Click **OK**.

With these settings, a finer time resolution is used, but the solution is only stored at the probe points. This will reduce the size of the model file.

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

Next, add an auxiliary ODE to retrieve the displacements from the velocities computed in the previous study. Here, we will be interested in the *y*-components of the corresponding vector fields only.

ADD PHYSICS

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

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

3 In the tree, select **Mathematics>ODE and DAE Interfaces>Point ODEs and DAEs (pode)**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1 - ELTE (store full solution)	
Study 2 - ELTE (store at points)	

5 Click **Add to Selection** in the window toolbar.

POINT ODES AND DAES (PODE)

1 In the **Settings** window for **Point ODEs and DAEs**, locate the **Point Selection** section.

2 From the **Selection** list, choose **Probe Points**.

3 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

4 In the **Physical Quantity** dialog box, type `id:displacement` in the text field.

5 Click **Filter**.

6 In the tree, select **General>Displacement (m)**.

7 Click **OK**.

8 In the **Settings** window for **Point ODEs and DAEs**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog box, type `id:velocity` in the text field.

11 Click **Filter**.

12 In the tree, select **General>Velocity (m/s)**.

13 Click **OK**.

14 In the **Settings** window for **Point ODEs and DAEs**, click to expand the **Dependent Variables** section.

15 In the **Field name** text field, type `uy`.

16 In the **Dependent variables** table, enter the following settings:

uy

Distributed ODE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Point ODEs and DAEs (pode)** click **Distributed ODE 1**.

2 In the **Settings** window for **Distributed ODE**, locate the **Source Term** section.

3 In the *f* text field, type `vy`.

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

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 **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Elastic Waves, Time Explicit (elte)	

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

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

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

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - Displacement (ODE)` in the **Label** text field.

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

Use the same time interval as in Study 2.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3 - Displacement (ODE)** click **Step 1: Time Dependent**.

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

3 In the **Times** text field, type `range(0, t0/20, 17*t0)`.

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 2 - ELTE (store at points), Time Dependent**.

7 From the **Time (s)** list, choose **All**.

Modify the solver to the one appropriate for solving wave problems. This is the same default when e.g. solving a transient pressure acoustics model.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Method** list, choose **Generalized alpha**.

5 From the **Steps taken by solver** list, choose **Manual**.

6 In the **Time step** text field, type `1/(60*f0)`.

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

Now, plot the computed seismograms at the probe points. The results should look like the ones in Figure 4.

RESULTS

1D Plot Group 5

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

2 In the **Settings** window for **1D Plot Group**, type `Displacement in (x,y) = (-10.5 cm, -8 cm)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Displacement (ODE)/ Solution 3 (sol3)**.

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

5 In the **Title** text area, type `Displacement, y-component (m)`.

6 Locate the **Plot Settings** section. Select the **y-axis label** check box.

7 In the associated text field, type `Displacement, y-component (m)`.

Point Graph 1

1 In the **Displacement in (x,y) = (-10.5 cm, -8 cm)** toolbar, click **Point Graph**.

2 Select Point 9 only.

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

4 In the **Expression** text field, type `uy`.

5 In the **Displacement in (x,y) = (-10.5 cm, -8 cm)** toolbar, click **Plot**.

Displacement in (x,y) = (-10.5 cm, -8 cm) 1

1 In the **Model Builder** window, right-click **Displacement in (x,y) = (-10.5 cm, -8 cm)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Displacement in (x,y) = (-3.5 cm, -8 cm)` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>Displacement in (x,y) = (-3.5 cm, -8 cm)** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Point 10 only.

5 In the **Displacement in (x,y) = (-3.5 cm, -8 cm)** toolbar, click **Plot**.

Generate the seismograms at the remaining probe points by repeating the previous instructions.