



Model created in COMSOL Multiphysics 6.4

# 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 domain to another requires a 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 domains.

## 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](#) ( $\text{kg}/\text{m}^3$  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  $c_{44} = c_{55} = c_{66}$ .

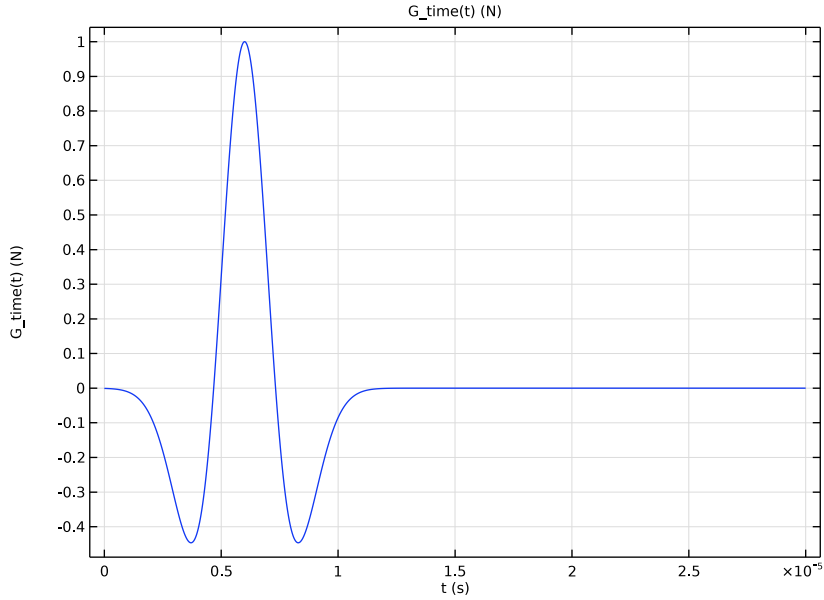
TABLE 1: MATERIAL PROPERTIES.

	$\rho$	$c_{11}$	$c_{12}$	$c_{22}$	$c_{66}$
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  $t_0 = 6$   $\mu\text{s}$ , 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 domains 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 by adding the Compute Displacement subfeature to the Elastic Waves, Time Explicit Model. In this model, the displacements are only retrieved at the probe points.

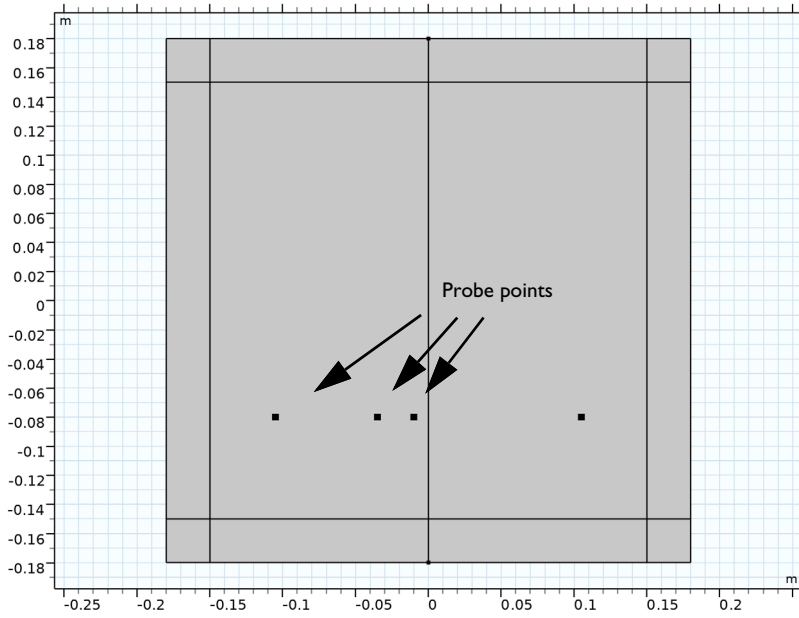


Figure 2: Model geometry layout.

## Results and Discussion

Figure 3 shows the quantitative evolution of the waves propagating from the source at  $t = 30, 60,$  and  $90 \mu\text{s}$ . 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 elliptical 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.

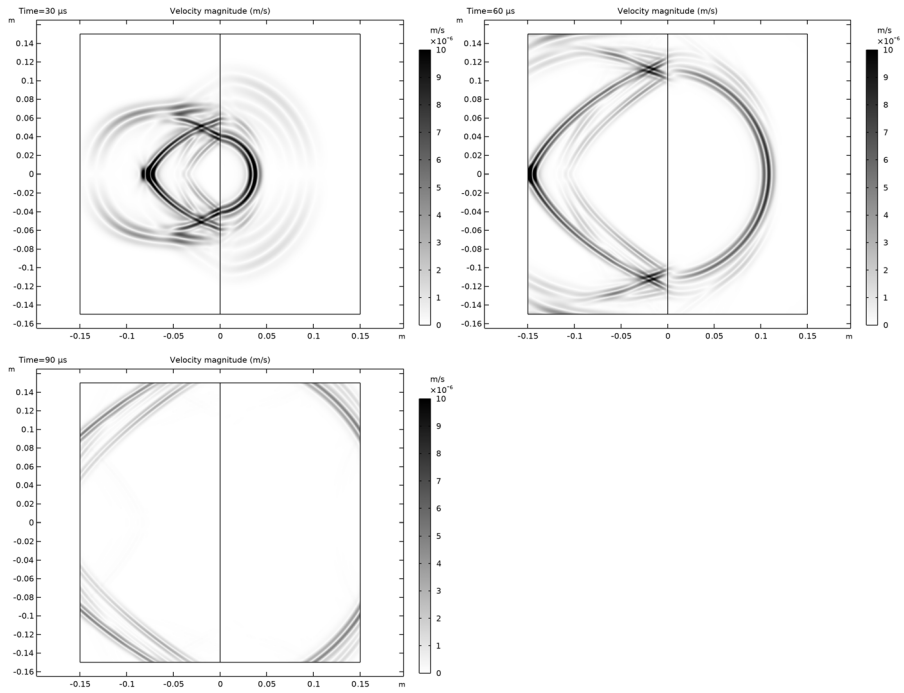


Figure 3: Velocity magnitude profiles at 30, 60, and 90  $\mu\text{s}$ .

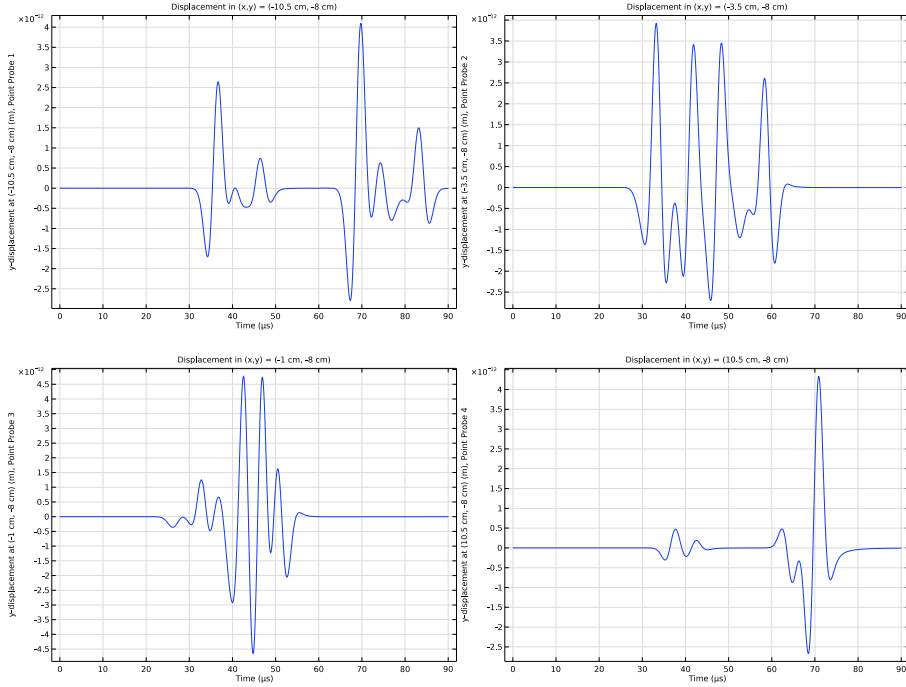


Figure 4: Seismograms at the probe points.

### 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,

$$\mathbf{F}_{\text{pnt}}(t) = \mathbf{F}_{\text{vol}}(t)\delta(x - x_0, y - y_0) = \mathbf{F}_{\text{vol}}(t)\delta(x - x_0)\delta(y - y_0)$$

One of the Dirac delta representations reads

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

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  $\lambda_{\min}$  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

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

where  $f_0$  is the dominant frequency. Although the upper cutoff frequency of the wavelet is  $3f_0$ ,  $\lambda_{\min}$  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 and others, “An arbitrary high-order discontinuous Galerkin method for elastic waves on unstructured meshes – IV. Anisotropy,” *Geophysical J. Int'l*, vol. 169, no. 3, pp. 1210–1228, 2007.

---

**Application Library path:** Structural\_Mechanics\_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 space and time functions given by a Gaussian and a Ricker wavelet, respectively.

### Analytic 1 (an1)

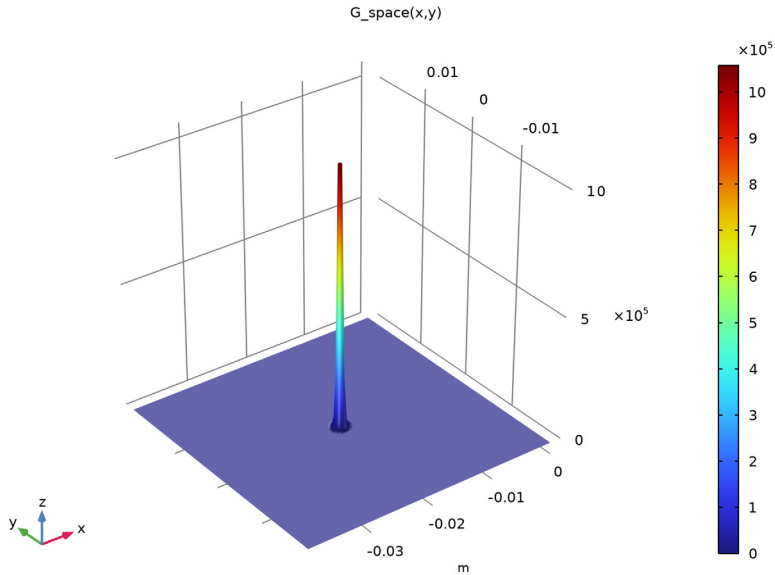
- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Analytic**.
- 2 In the **Settings** window for **Analytic**, type `G_space` in the **Function name** text field.
- 3 Locate the **Definition** section. In the **Expression** text field, type  $1 / (\pi * dS) * \exp(-((x - x0)^2 + (y - y0)^2) / dS)$ .
- 4 In the **Arguments** text field, type `x`, `y`.
- 5 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
<code>x</code>	<code>m</code>


- 6 In the **Function** text field, type `1`.
- 7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	<code>x</code>	-0.04	0	0	<code>m</code>
√	<code>y</code>	-0.02	0.02	0	

8 Click  **Plot**.



*Analytic 2 (an2)*

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Analytic**.
- 2 In the **Settings** window for **Analytic**, type `G_time` in the **Function name** text field.
- 3 Locate the **Definition** section. In the **Expression** text field, type  $(1 - 2\pi^2 f_0^2 (t - t_0)^2) \exp(-\pi^2 f_0^2 (t - t_0)^2)$ .
- 4 In the **Arguments** text field, type `t`.
- 5 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
t	s

- 6 In the **Function** text field, type `N`.
- 7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	t	0	5*t0	0	s

8 Click  **Create Plot**.


The signal should look like the one in [Figure 1](#).

## RESULTS

### *Impulse Frequency Content*

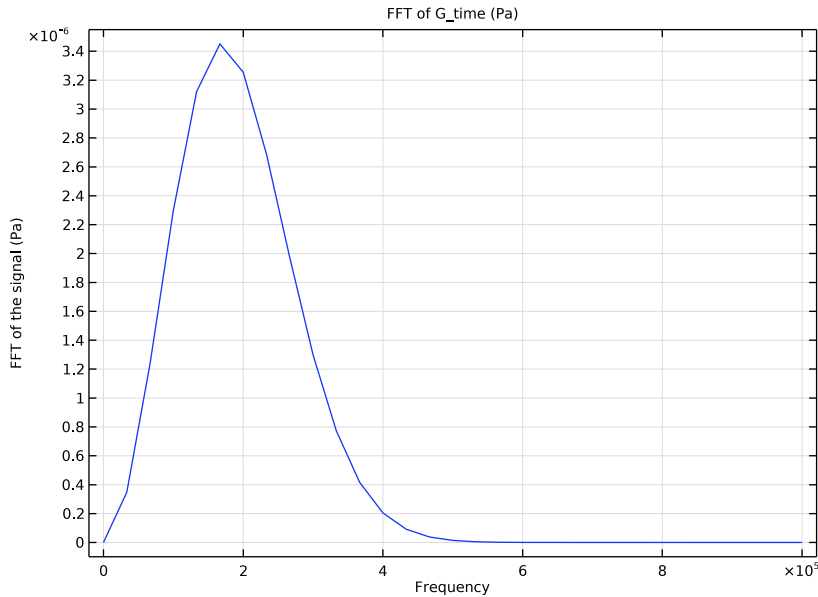
- 1 In the **Settings** window for **ID Plot Group**, type Impulse Frequency Content in the **Label** text field.
- 2 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 3 In the **Title** text area, type FFT of G\_time (Pa).
- 4 Locate the **Plot Settings** section.
- 5 Select the **y-axis label** checkbox. In the associated text field, type FFT of the signal (Pa).

### *Function 1*

- 1 In the **Model Builder** window, expand the **Impulse Frequency Content** node, then click **Function 1**.
- 2 In the **Settings** window for **Function**, locate the **Output** section.
- 3 From the **Display** list, choose **Discrete Fourier transform**.
- 4 From the **Show** list, choose **Frequency spectrum**.
- 5 From the **Scale** list, choose **Multiply by sampling period**.
- 6 In the **Impulse Frequency Content** toolbar, click  **Plot**.
- 7 Select the **Frequency range** checkbox.
- 8 In the **Maximum** text field, type 1000000.

9 In the **Impulse Frequency Content** toolbar, click  **Plot**.


The Fourier transformation of the signal should look like this. The plot indicates that the mesh should resolve frequency content up to 400 to 500 kHz.



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

## GEOMETRY I

*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** checkbox.
- 6 In the table, enter the following settings:


Layer name	Thickness (m)
Layer 1	L/6

- 7 Select the **Layers to the right** checkbox.

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



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

#### *Probe Points*

- 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 (abl)*

- 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, use an anisotropic material.

### **ELASTIC WAVES, TIME EXPLICIT (ELTE)**

*Elastic Waves, Time Explicit Model I*

- 1** In the **Model Builder** window, under **Component 1 (comp1) > Elastic Waves, Time Explicit (elte)** click **Elastic Waves, Time Explicit Model I**.
- 2** In the **Settings** window for **Elastic Waves, Time Explicit Model**, locate the **Linear Elastic Material** section.
- 3** From the **Material symmetry** list, choose **Anisotropic**.

### **MATERIALS**

*Material Isotropic*

- 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 Material1 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, 0, 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, 3.96e10, 0, 0, 0, 0, 3.96e10}	Pa	Anisotropic, Voigt notation
Density	rho	7100	kg/m <sup>3</sup>	Basic

*Material Anisotropic*


- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Material1 Anisotropic in the **Label** text field.
- 3 Select Domains 1–6 only.



- 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  $\mathbf{F}_{\text{tot}}$  vector as


0	x
$G_{\text{space}}(x,y)*G_{\text{time}}(t)$	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.

Define a **Point Probe** for each of the probe points to measure the vertical displacement.

### **DEFINITIONS**

#### *Point Probe 1 (point1)*

- 1 In the **Definitions** toolbar, click  **Probes** and choose **Point Probe**.
- 2 Select Point 9 only.
- 3 In the **Settings** window for **Point Probe**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Elastic Waves, Time Explicit > Displacement > Displacement - m > elte.uy - Displacement, y-component**.
- 4 Locate the **Expression** section.
- 5 Select the **Description** checkbox. In the associated text field, type `y-displacement at (-10.5 cm, -8 cm)`.

#### *Point Probe 2 (point2)*

- 1 Right-click **Point Probe 1 (point1)** and choose **Duplicate**.
- 2 Select Point 10 only.
- 3 In the **Settings** window for **Point Probe**, locate the **Expression** section.

4 In the **Description** text field, type y-displacement at (-3.5 cm, -8 cm).

*Point Probe 3 (point3)*

1 Right-click **Point Probe 2 (point2)** and choose **Duplicate**.

2 Select Point 11 only.

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

4 In the **Description** text field, type y-displacement at (-1 cm, -8 cm).

*Point Probe 4 (point4)*

1 Right-click **Point Probe 3 (point3)** and choose **Duplicate**.

2 Select Point 16 only.

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

4 In the **Description** text field, type y-displacement at (10.5 cm, -8 cm).

Create groups to facilitate the navigation through the model.

*Point Probe 1 (point1), Point Probe 2 (point2), Point Probe 3 (point3), Point Probe 4 (point4)*

1 In the **Model Builder** window, under **Component 1 (comp1) > Definitions**, Ctrl-click to select **Point Probe 1 (point1)**, **Point Probe 2 (point2)**, **Point Probe 3 (point3)**, and **Point Probe 4 (point4)**.


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

## MESH 1

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

Use a mapped mesh. This will generate a uniform mesh adequate for time-explicit simulations.

*Mapped 1*

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

*Size*

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

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type  $cs_{an}/(2*f0)/1.5$ .

4 Click  **Build All**.

## STUDY 1 - ELTE (STORE FULL SOLUTION)

- 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 From the **Time unit** list, choose **µs**.
- 4 In the **Output 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.

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


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

## RESULTS

### Study 1 - ELTE (store full solution)/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results > Datasets** node, then click **Study 1 - ELTE (store full solution)/Solution 1 (sol1)**.

### 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 5 and 8 only.


### Velocity Magnitude (elte)

- 1 In the **Model Builder** window, under **Results** click **Velocity Magnitude (elte)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3 From the **Time (µs)** list, choose **30**.

### *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** From the **Color table transformation** list, choose **Reverse**.
- 5** Click to expand the **Range** section. Select the **Manual color range** checkbox.
- 6** In the **Minimum** text field, type 0.
- 7** In the **Maximum** text field, type  $1e-5$ .


These settings will sharpen the contrast of the velocity profile.

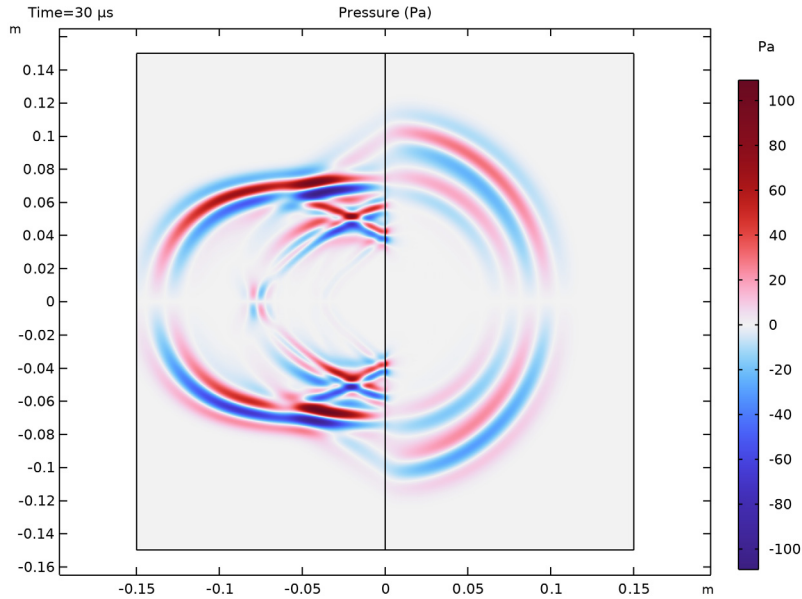
- 8** In the **Velocity Magnitude (elte)** toolbar, click  **Plot**.

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, under **Results** click **Pressure (elte)**.
- 2** In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3** From the **Time ( $\mu$ s)** list, choose **30**.


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



*Displacement in  $(x,y) = (-10.5 \text{ cm}, -8 \text{ cm})$*

- 1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.
- 2 In the **Settings** window for **ID Plot Group**, type Displacement in  $(x,y) = (-10.5 \text{ cm}, -8 \text{ cm})$  in the **Label** text field.
- 3 Locate the **Title** section. From the **Title type** list, choose **Label**.
- 4 Locate the **Legend** section. Clear the **Show legends** checkbox.

*Probe Table Graph 1*


- 1 In the **Model Builder** window, expand the **Displacement in  $(x,y) = (-10.5 \text{ cm}, -8 \text{ cm})$**  node, then click **Probe Table Graph 1**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 In the **Columns** list box, select **y-displacement at  $(-10.5 \text{ cm}, -8 \text{ cm})$  (m), Point Probe 1**.
- 4 In the **Displacement in  $(x,y) = (-10.5 \text{ cm}, -8 \text{ cm})$**  toolbar, click  **Plot**.

*Displacement in  $(x,y) = (-3.5 \text{ cm}, -8 \text{ cm})$*

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

- 3 In the **Settings** window for **ID Plot Group**, type Displacement in  $(x,y) = (-3.5 \text{ cm}, -8 \text{ cm})$  in the **Label** text field.


*Probe Table Graph 1*

- 1 In the **Model Builder** window, click **Probe Table Graph 1**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 In the **Columns** list box, select **y-displacement at  $(-3.5 \text{ cm}, -8 \text{ cm})$  (m), Point Probe 2**.
- 4 In the **Displacement in  $(x,y) = (-3.5 \text{ cm}, -8 \text{ cm})$**  toolbar, click  **Plot**.

*Displacement in  $(x,y) = (-1 \text{ cm}, -8 \text{ cm})$*

- 1 In the **Model Builder** window, right-click **Displacement in  $(x,y) = (-3.5 \text{ cm}, -8 \text{ cm})$**  and choose **Duplicate**.
- 2 In the **Model Builder** window, click **Displacement in  $(x,y) = (-3.5 \text{ cm}, -8 \text{ cm})$  1**.
- 3 In the **Settings** window for **ID Plot Group**, type Displacement in  $(x,y) = (-1 \text{ cm}, -8 \text{ cm})$  in the **Label** text field.


*Probe Table Graph 1*

- 1 In the **Model Builder** window, click **Probe Table Graph 1**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 In the **Columns** list box, select **y-displacement at  $(-1 \text{ cm}, -8 \text{ cm})$  (m), Point Probe 3**.
- 4 In the **Displacement in  $(x,y) = (-1 \text{ cm}, -8 \text{ cm})$**  toolbar, click  **Plot**.

*Displacement in  $(x,y) = (10.5 \text{ cm}, -8 \text{ cm})$*


- 1 In the **Model Builder** window, right-click **Displacement in  $(x,y) = (-1 \text{ cm}, -8 \text{ cm})$**  and choose **Duplicate**.
- 2 In the **Model Builder** window, click **Displacement in  $(x,y) = (-1 \text{ cm}, -8 \text{ cm})$  1**.
- 3 In the **Settings** window for **ID Plot Group**, type Displacement in  $(x,y) = (10.5 \text{ cm}, -8 \text{ cm})$  in the **Label** text field.

*Probe Table Graph 1*


- 1 In the **Model Builder** window, click **Probe Table Graph 1**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 In the **Columns** list box, select **y-displacement at  $(10.5 \text{ cm}, -8 \text{ cm})$  (m), Point Probe 4**.
- 4 In the **Displacement in  $(x,y) = (10.5 \text{ cm}, -8 \text{ cm})$**  toolbar, click  **Plot**.

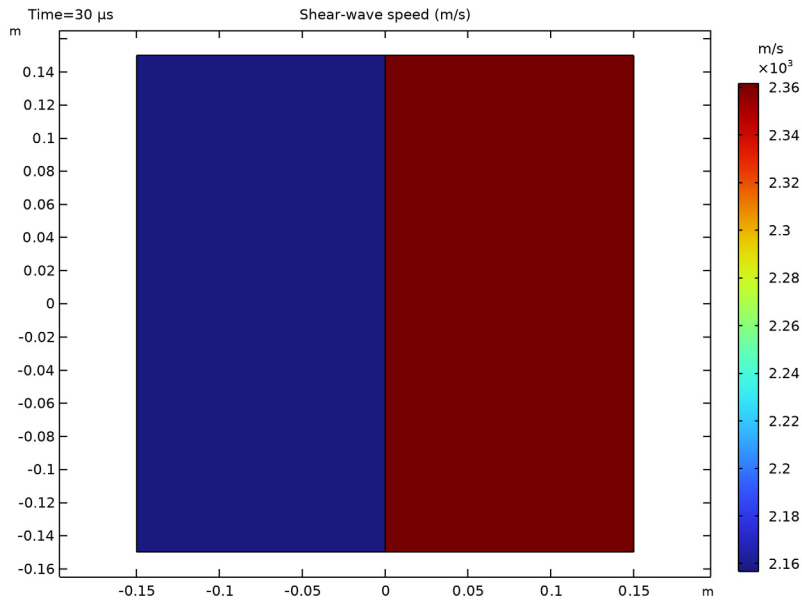
Next, inspect the shear and pressure wave speeds in the isotropic material and their equivalents in the anisotropic material.

### Apparent Shear Wave Speed

- 1 In the **Results** toolbar, click  **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 ( $\mu\text{s}$ )** list, choose **30**.
- 4 Locate the **Color Legend** section. Select the **Show units** checkbox.

### 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 `e1te.cs`.
- 4 In the **Apparent Shear Wave Speed** toolbar, click  **Plot**.



### Apparent Pressure Wave Speed

- 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 **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 `e1te.cp`.
- 4 In the **Apparent Pressure Wave Speed** toolbar, click  **Plot**.

