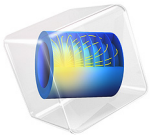


Model created in COMSOL Multiphysics 6.4

Seismic Event, Explicit Dynamics

Introduction

This tutorial studies the evolution of seismic waves in a two-dimensional isotropic linear elastic half space with a small mountain on its top. The seismic event is due to a vertical load applied at a point on the top of the half space to the right of the mountain. As a result of the applied load, various types of elastic waves propagate into the bulk and along the top surface. In particular, these include longitudinal, shear, and head (von Schmidt) waves, as well as Rayleigh waves.

The mountain acts as a scatterer, which results in a new envelope of backscattering waves of the same nature.

This is a version of the model Ground Motion After Seismic Event: Scattering off a Small Mountain, where a Solid Mechanics, Explicit Dynamics interface is used instead of the Elastic Waves, Time Explicit interface.

Model Definition

The half space $y < 0$ with a small mountain on its top is made of a linear elastic material with the properties given in Table 1.

TABLE 1: MATERIAL PROPERTIES.

ρ	c_p	c_s
2200 kg/m^3	3.2 km/s	1.8475 km/s

The mountain has the height of 200 m. Its shape is given by a Gaussian bell

$$y(x) = 0.2e^{-\left(\frac{x}{0.3\ \mathrm{km}}\right)^2}\ \mathrm{km}$$

located 5 km to the left of the vertical line $y = 0$. The geometry repeats the one used in Ref. 1.

The half space is modeled as a 40 km wide and 20 km high rectangle with the center of its top side placed at the origin $(0, 0)$.

The reflections of the waves at the outer boundaries are suppressed by adding a Low-Reflecting Boundary condition to the sides and bottom of the rectangle.

The top of the half space is free, except for the load at the point $(0, 0)$. Its distribution in time is given by a Ricker wavelet shown in Figure 1.

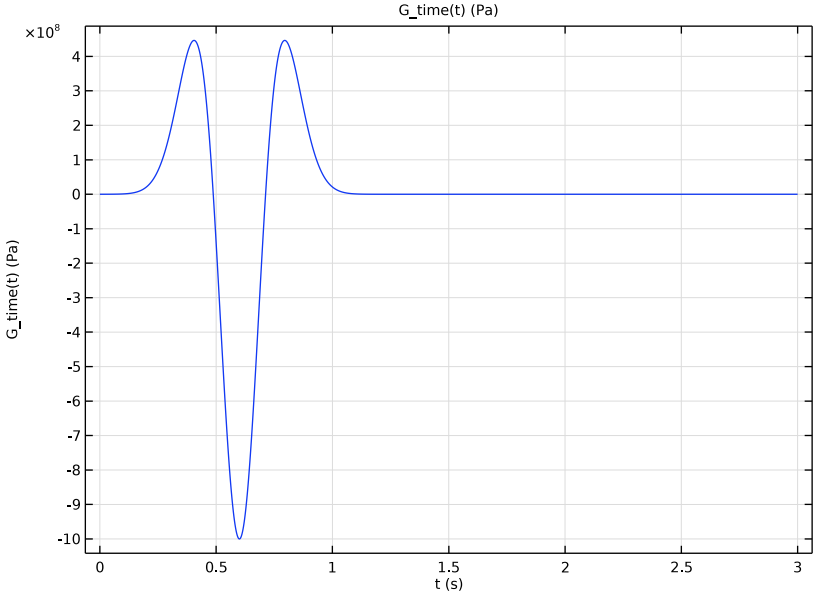


Figure 1: Source distribution in time.

The free top boundary is of particular interest, because it gives rise to Rayleigh waves that propagate near the surface with a speed lower than that of the shear wave in the bulk material. One of the classical estimates of the Rayleigh wave speed, v_R, with respect to the shear wave speed is given by

$$\frac{v_R}{c_s} \approx \frac{0{,}87 + 1{,}12\nu}{1 + \nu}$$

where ν is the Poisson's ratio. It is therefore important to use the Rayleigh wave speed while building the mesh in order to resolve the Rayleigh waves properly. That is, the minimum wavelength used to define the maximum mesh element size will be

$$\lambda_{min} = \frac{v_R}{f_{max}}$$

Figure 2 shows the velocity magnitude profiles in the physical domain computed at 3, 6, 9, and 12 s. In the upper-left figure, the propagation of the pressure (faster) and the shear (slower) waves are shown at t = 3 s.

When the wave front reaches the mountain, it results in the backscattering of the waves. This is seen in the upper-right picture (t = 6 s). The Rayleigh wave also becomes distinguishable in the region below the surface, as it travels slower than the shear wave. The head (von Schmidt) wave that has a conical shape and propagates at the critical angle β to the ground top, such that $\sin\beta = c_s/c_p$ is also visible.

In the lower-left and lower-right pictures (t = 9 s and t = 12 s), the initial wave front and the scattered waves leave the computational domain.

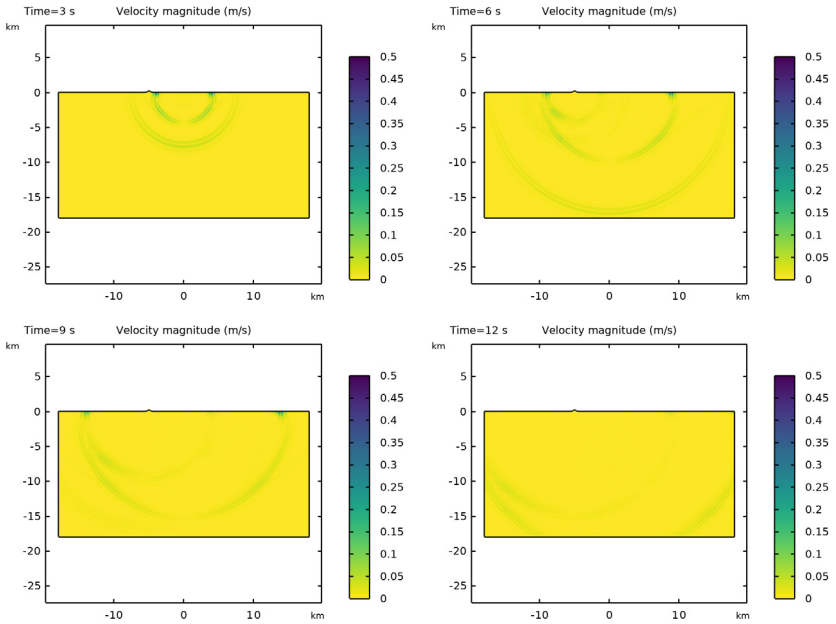


Figure 2: Velocity magnitude profiles at four different time steps

The second principal invariant of the stress tensor contains information on the pressure and shear waves traveling through the solid. Pressure waves have a positive value of this stress invariant, while shear waves show a negative value. As this invariant has units of

squared stress, we take the square root of the absolute value of the invariant and multiply it with its sign to create Figure 3.

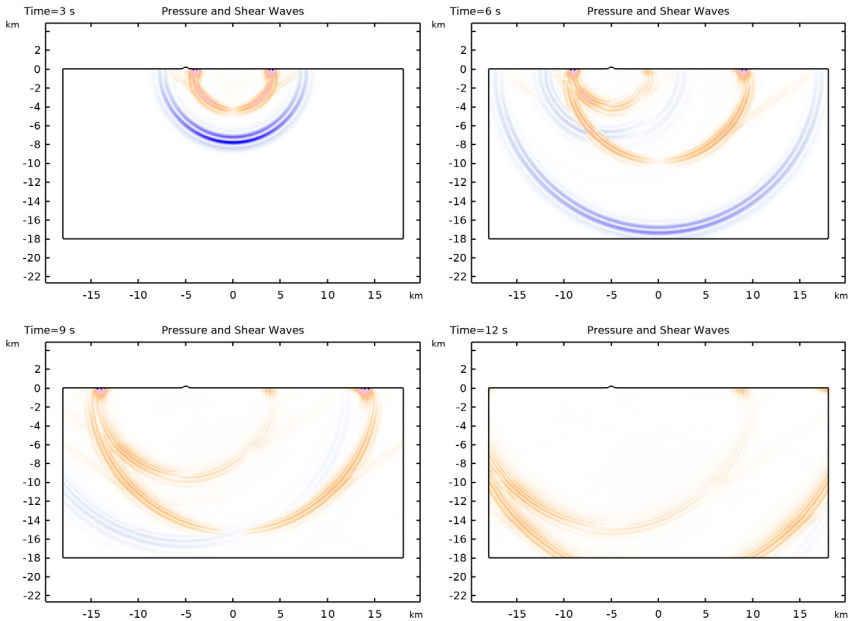


Figure 3: Signed squared root of the second principal invariant at four different time steps.

Pressure waves are colored in blue, while shear, Rayleigh, and head waves are colored in orange.

Notes About the COMSOL Implementation

To resolve the Rayleigh wave properly the maximum element size is computed as the minimum wavelength divided by 10 since linear elements are used in the **Solid Mechanics, Explicit Dynamics** interface.

Reference

1. D. Appelö and N.A. Petersson, "A Stable Finite Difference Method for the Elastic Wave Equation on Complex Geometries with Free Surfaces," *Commun. Comput. Phys.*, vol. 5, pp. 84–107, 2009.

Application Library path: `Structural_Mechanics_Module/Elastic_Waves/seismic_event`

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 **Structural Mechanics** > **Explicit Dynamics** > **Solid Mechanics, Explicit Dynamics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Explicit Dynamics**.

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 `seismic_event_parameters.txt`.

Create the source space and time functions given by a Gaussian and a Ricker wavelet, respectively.

G_space

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

2 In the **Settings** window for **Analytic**, type `G_space` in the **Label** text field.

3 In the **Function name** text field, type `G_space`.

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

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

Argument	Unit
x	m

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

G_time

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Analytic**.

2 In the **Settings** window for **Analytic**, type `G_time` in the **Label** text field.

3 In the **Function name** text field, type `G_time`.

4 Locate the **Definition** section. In the **Expression** text field, type `1e9*(2*(pi*f0*(t - t0))^2 - 1)*exp(-(pi*f0*(t - t0))^2)`.

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

6 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
t	s

7 In the **Function** text field, type `Pa`.

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

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	t	0	3	0	s

9 Click **Create Plot**.

RESULTS

Impulse Frequency Content

1 In the **Settings** window for **1D 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`.

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**.

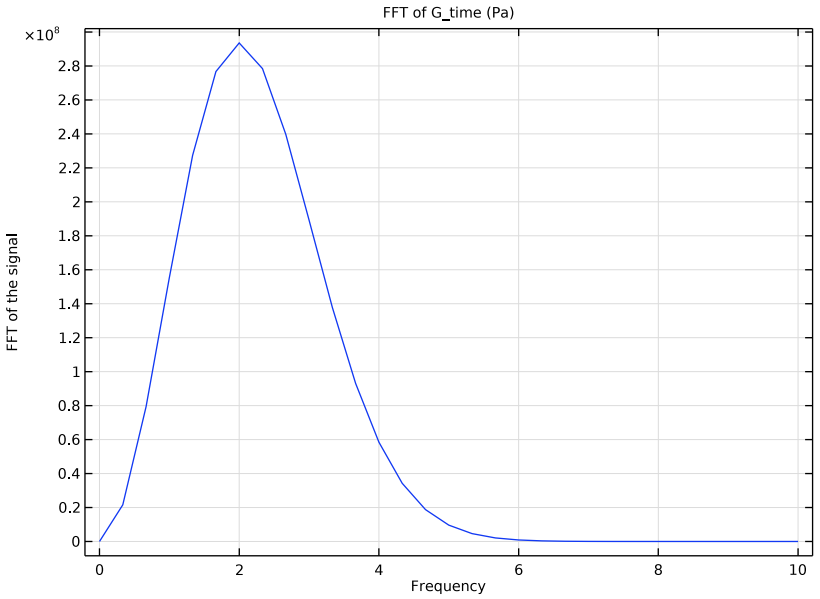
6 In the **Impulse Frequency Content** toolbar, click **Plot**.

7 Select the **Frequency range** checkbox.

8 In the **Maximum** text field, type 10.

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 4 to 5 Hz.



GEOMETRY 1

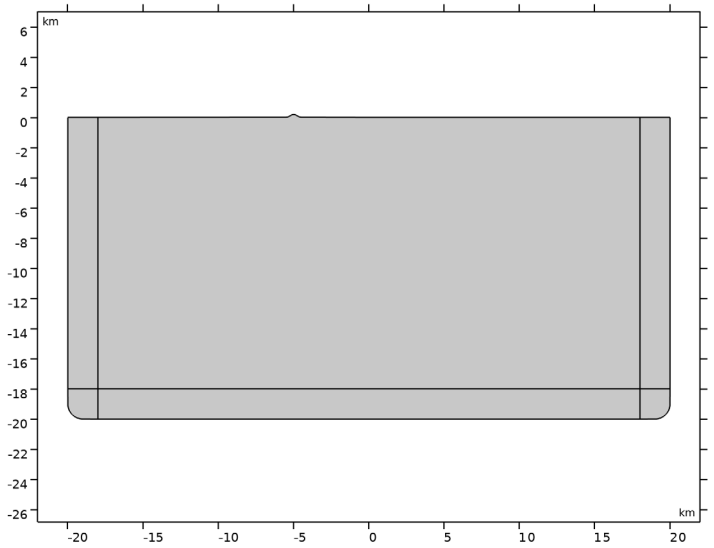
The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `seismic_event_geom_sequence.mph`.

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

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.



MATERIALS

Ground Material

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics** > **Linear Elastic Material** > **Pressure-Wave and Shear-Wave Speeds**.

4 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	3.2[km/s]	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	1.8475[km/s]	m/s	Pressure-wave and shear-wave speeds
Density	rho	2200[kg/m^3]	kg/m³	Basic

SOLID MECHANICS, EXPLICIT DYNAMICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Ground Material (mat1)** node, then click **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics (solid)** > **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Quadrature Settings** section.

3 Find the **Hexahedron** subsection. From the **Hourglass stabilization** list, choose **Hessian**.

Low-Reflecting Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Low-Reflecting Boundary**.

2 Select Boundaries 1, 2, 6, 12, and 16 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 9 and 10 only.

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

4 Specify the $\mathbf{f}_A$ vector as

G_space(x)*G_time(t)	y

MESH 1

Build the mesh specifying the maximum element size.

Free Quad 1

1 In the **Mesh** toolbar, click **Free Quad**.

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

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

4 Select Domain 7 only.

5 Click to expand the **Control Entities** section. From the **Smooth across removed control entities** list, choose **Off**.

6 Click to expand the **Mesh Generation** section. From the **Method** list, choose **Quad**.

Size 1

Right-click **Free Quad 1** and choose **Size**.

Size

1 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

2 In the **Maximum element size** text field, type `cr/(2*f0)/10`.

3 In the **Minimum element size** text field, type `cr/(2*f0)/10`.

4 In the **Maximum element growth rate** text field, type `1.5`.

5 In the **Curvature factor** text field, type `0.5`.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** > **Free Quad 1** click **Size 1**.

2 Select Domains 10 and 12 only.

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

4 Click the **Custom** button.

Copy Domain 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Domain**.

2 Select Domain 7 only.

3 In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Domain 12 only.

6 Click to expand the **Control Entities** section. From the **Smooth across removed control entities** list, choose **On**.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.

3 From the **Smooth across removed control entities** list, choose **Off**.

STUDY 1

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

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

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

Step 1: Explicit Dynamics

1 In the **Model Builder** window, under **Study 1** click **Step 1: Explicit Dynamics**.

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

3 In the **Output times** text field, type `range(0,1,12)`.

4 Click to expand the **Results While Solving** section. From the **Update at** list, choose **Time steps taken by solver**.

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

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, type `pre` in the text field.

5 In the tree, select **General** > **Pressure (Pa)**.

6 Click **OK**.

7 In the **Settings** window for **Preferred Units**, locate the **Units** section.

8 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Pressure	Pa	MPa

9 Select the **Apply conversions to expressions with the same dimensions** checkbox.

10 Click **Apply**.

Velocity

1 In the **Results** toolbar, click **2D Plot Group**.

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

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

4 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 4 only.

6 Select the **Apply to dataset edges** checkbox.

7 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Surface 1

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

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

3 In the **Expression** text field, type `solid.vel`.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type `0.5`.

These settings will sharpen the contrast of the velocity profile.

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

7 From the **Color table transformation** list, choose **Reverse**.

8 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

9 From the **Smoothing** list, choose **Everywhere**.

Then generate the plots at 6, 9, and 12 s. The structural velocity profiles at the time steps chosen in Study 1 are shown in Figure 2.

Pressure

1 In the **Results** toolbar, click **2D Plot Group**.

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

3 Locate the **Data** section. From the **Time (s)** list, choose **6**.

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

5 Select Domain 4 only.

6 Select the **Apply to dataset edges** checkbox.

Surface 1

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

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

3 In the **Expression** text field, type `solid.pmGp`.

4 Locate the **Range** section. Select the **Manual color range** checkbox.

5 In the **Minimum** text field, type `-0.25`.

6 In the **Maximum** text field, type `0.25`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Dipole**.

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

9 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

10 In the **Pressure** toolbar, click **Plot**.

Create the last plot that will help discern the pressure and shear waves.

Pressure and Shear Waves

1 In the **Results** toolbar, click **2D Plot Group**.

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

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

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

5 Select Domain 4 only.

6 Select the **Apply to dataset edges** checkbox.

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

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

9 Click to expand the **Quality** section. From the **Smoothing** list, choose **Inside geometry domains**.

Surface 1

1 Right-click **Pressure and Shear Waves** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.gpeval(sqrt(abs(solid.I2s))*sign(solid.I2s))`.

4 Locate the **Range** section. Select the **Manual color range** checkbox.

5 In the **Minimum** text field, type `-0.24`.

6 In the **Maximum** text field, type `0.3`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Twilight**.

8 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

9 From the **Resolution** list, choose **No refinement**.

10 From the **Smoothing** list, choose **Everywhere**.

11 In the **Pressure and Shear Waves** toolbar, click **Plot**.

Loop through the different times to reproduce Figure 3.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Pressure and Shear Waves**.

4 Locate the **Frames** section. In the **Number of frames** text field, type 13.

5 In the **Frame number** text field, type 13.

Geometry Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Blank Model**.

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

1 In the **Settings** window for **Geometry**, locate the **Units** section.

2 From the **Length unit** list, choose **km**.

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 `40[km]`.

4 In the **Height** text field, type `20[km]`.

5 Locate the **Position** section. In the **x** text field, type `-20[km]`.

6 In the **y** text field, type `-20[km]`.

7 Click to expand the **Layers** section. Select the **Layers to the left** checkbox.

8 Select the **Layers to the right** checkbox.

9 In the table, enter the following settings:

Layer name	Thickness (km)
Layer 1	2[km]

Parametric Curve 1 (pc1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type `2[km]`.

4 Locate the **Expressions** section. In the **x** text field, type `s`.

5 In the **y** text field, type `0.2*(exp(-((s - 1[km])/0.3[km])^2) - exp(-(1[km]/0.3[km])^2))[km]`.

6 Locate the **Position** section. In the **x** text field, type `-6[km]`.

7 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 `2.4[km]`.

4 In the **Height** text field, type `0.7[km]`.

5 Locate the **Position** section. In the **x** text field, type `-6.2[km]`.

Point 1 (pt1)

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

2 In the **Settings** window for **Point**, click **Build Selected**.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **r2** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Click to select the **Activate Selection** toggle button for **Tool objects**.

5 Select the object **pc1** only.

6 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **par1**, select Domain 1 only.

5 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **r1**, select Points 1 and 10 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 1.

5 Click **Build Selected**.

Next create additional domains to improve the mesh quality at the corners.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

x (km)	y (km)
-20	-19
-19.3	-19
-19	-19.3
-19	-20

5 Click **Build Selected**.

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

x (km)	y (km)
-19	-19.3
-18	-19.3

5 Click **Build Selected**.

Polygon 3 (pol3)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

x (km)	y (km)
-19.3	-19
-19.3	-18

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

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

3 Select the **Keep input objects** checkbox.

4 Select the objects **pol1**, **pol2**, and **pol3** only.

5 Click **Build Selected**.

Form Composite Domains 1 (cmd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 On the object **fin**, select Domains 7 and 8 only.

3 In the **Settings** window for **Form Composite Domains**, click **Build Selected**.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **cmd1**, select Boundaries 2, 6, 7, 9, 11, 22, and 26–29 only.

3 In the **Settings** window for **Mesh Control Edges**, click **Build Selected**.