Model created in COMSOL Multiphysics 6.4

Ground Motion After Seismic Event: Scattering off a Small Mountain

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This tutorial studies the evolution of the ground motion after a seismic event 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 the result of the applied load, various types of elastic waves propagate into the bulk and along the top surface of the linear elastic half space. 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 the new envelope of backscattering waves of the same nature.

Model Definition

The half space $y < 0$ with a small mountain on its top is occupied by 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 the Gaussian bell

$$y(x) = 0{,}2e^{-\left(\frac{x}{0{,}3}\right)^2} \text{ km}$$

located 5 km to the left of the vertical line $y = 0$. This setup 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 being at the coordinate system origin (0, 0).

The reflections of the waves that reach the outer boundary are suppressed by adding the Absorbing Layers to the left, right, and bottom sides of the rectangle. The layers thickness is 2 km. In addition, a Low-Reflecting Boundary condition is imposed on the outer boundaries.

The top of the half space is free except for the load at the coordinate system origin. Its distribution in time is given by a Ricker wavelet shown in Figure 1.

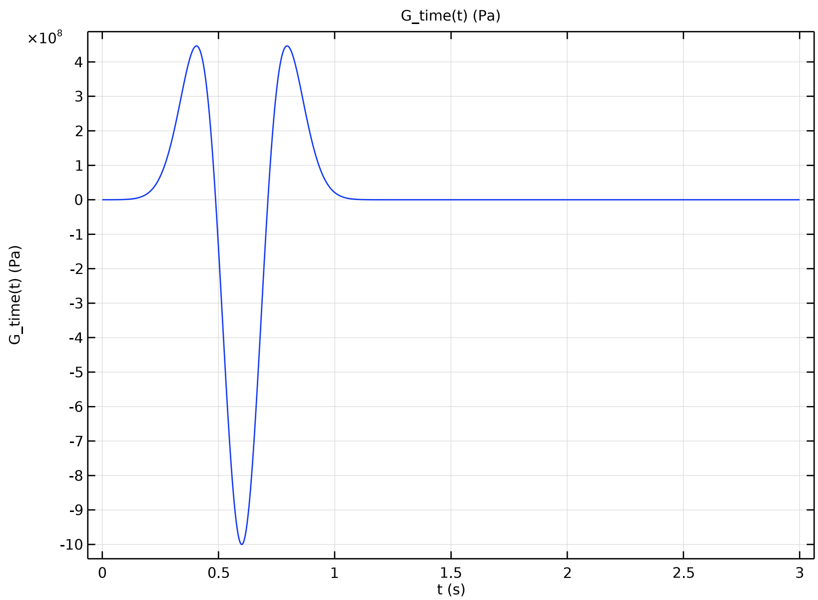


Figure 1: Source distribution in time.

The free top boundary is of particular interest, because it gives rise to Rayleigh waves that propagate in a shallow region under the surface with the speed lower than that of the shear wave. One of the classical estimates of the Rayleigh wave speed, v_R, is

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

where ν is the Poisson's ratio. This 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

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

Results and Discussion

Figure 2 shows the velocity magnitude profiles in the physical domain computed at 3, 6, 9, and 12 s. In the upper-left picture, one can see the propagation of the longitudinal (faster) and the shear (slower) waves in the medium at $t = 3$ s. They reach the mountain, which results in the backscattering of the wave. This is seen in the upper-right picture at $t = 6$ s. The Rayleigh wave also becomes distinguishable in the shallow region below the top of the ground, as it travels slower than the shear wave. One can also see the head (von Schmidt) waves that has a conical shape and propagates at the critical angle β to the ground top, such that $\sin\beta = c_s/c_p$. In the lower-left ($t = 9$ s) and lower-right ($t = 12$ s) pictures, the initial and the scattered waves leave the computational domain.

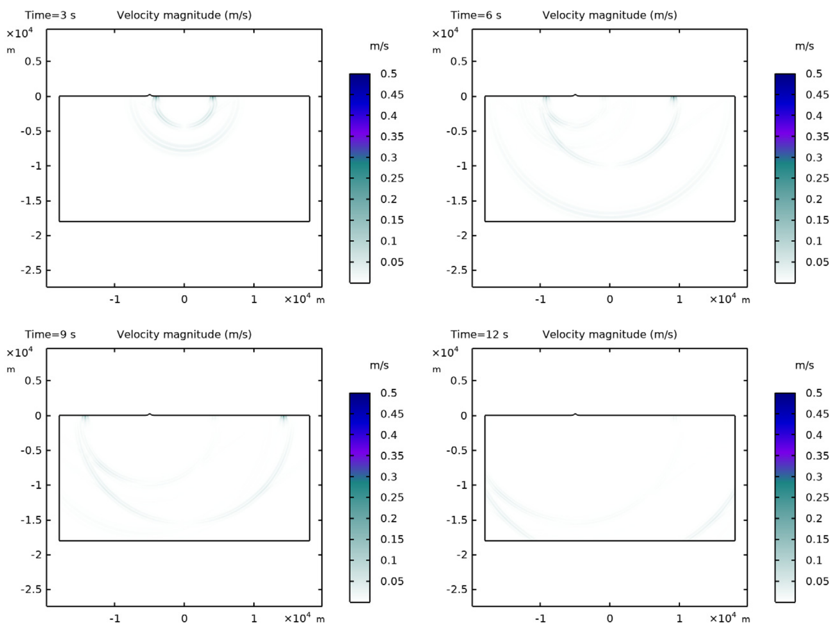


Figure 2: Velocity magnitude profiles at 4 different time steps.

The second principal invariant of the stress tensor contains information on the pressure and shear waves traveling through the solid. Pressure waves present a positive value of the invariant while shear waves present a negative value. As this invariant has units of squared stress, we will take the square root of the absolute value of the invariant and multiply it with the sign of the invariant to create Figure 3.

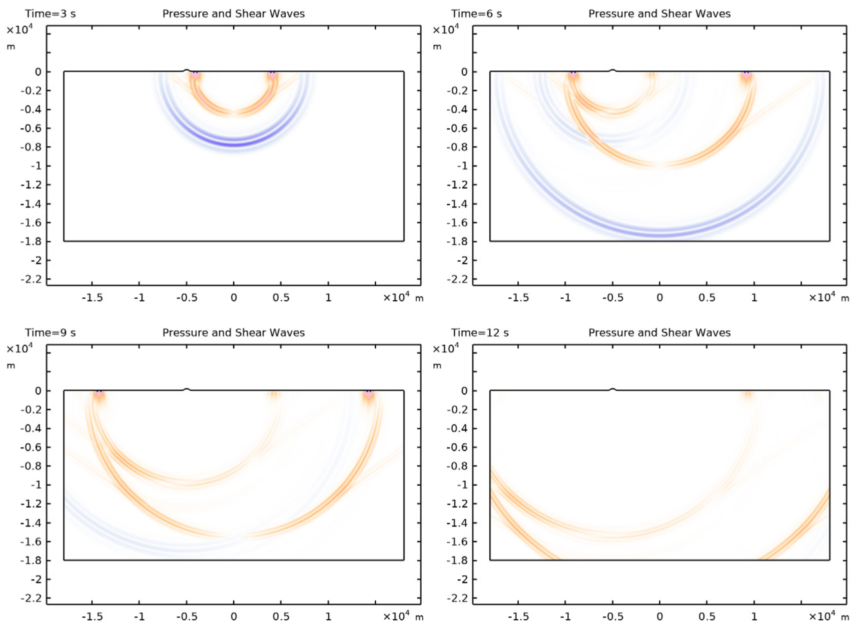


Figure 3: Signed squared root of the second principal invariant at 4 different time steps.

Notes About the COMSOL Implementation

The same notes as in the Isotropic-Anisotropic Sample: Elastic Wave Propagation model apply here.

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: `Acoustics_Module/Elastic_Waves/ground_motion_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 **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 `ground_motion_seismic_event_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 f(X) **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/sqrt(pi*dS)*exp(-x^2/dS)`.
4. Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
x	m

5 In the **Function** text field, type 1.

Analytic 2 (an2)

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 **Function name** text field.

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

7 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

8 Click **Create Plot**.

The signal should look like the one in Figure 1.

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 (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**.

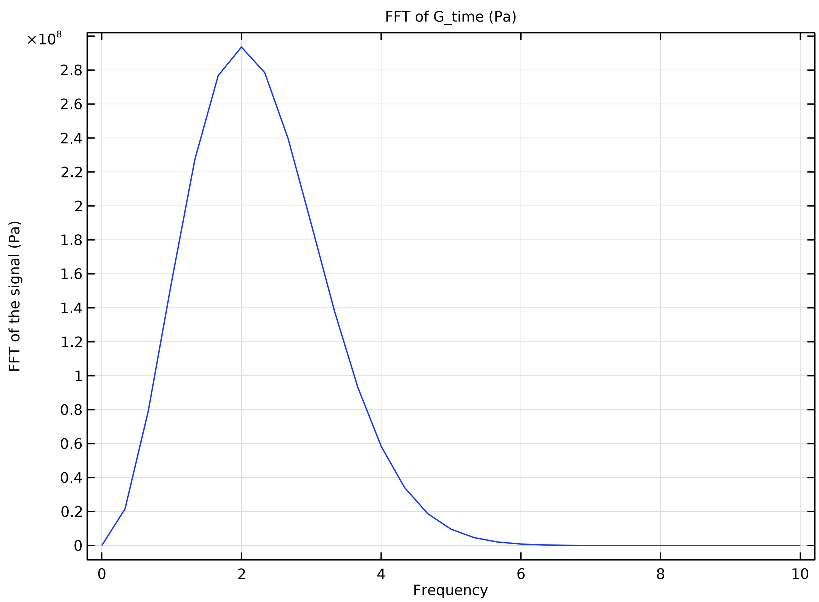
6 In the **Impulse Frequency Content** toolbar, click **Plot**.

7 Select the **Frequency range** checkbox.

8 In the **Maximum** text field, type `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

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]`.

Surround the computational domain by layers from the left, right, and bottom. They will be used to set up the absorbing layers (sponge layers) to mimic the wave propagation in open domain.

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 (m)
`Layer 1`	`2[km]`

Create a 0.2 km high and 1 km wide mountain on the top of the rectangle.

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]`.

Here, the subtraction ensures that the mountain base lies on the line $y = 0$.

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**.

Form Union (fin)

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

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 4 and 5 only.

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

DEFINITIONS

Absorbing Layer 1 (ab1)

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

2 Select Domains 1–3, 5, and 6 only.

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.

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 **Specify** list, choose **Pressure-wave and shear-wave speeds**.

Create an isotropic material with the desired properties.

MATERIALS

Ground Material (mat1)

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

ELASTIC WAVES, TIME EXPLICIT (ELTE)

Boundary Load 1

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

2 Select Boundaries 10 and 11 only.

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

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

0	x
G_space(x)*G_time(t)	y

Low-Reflecting Boundary 1

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

2 Select Boundaries 1–3, 7, 13, 17, and 18 only.

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 **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `cr/(2*f0)/1.5`.
5. Click **Build All**.

STUDY 1

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Output times** text field, type `range(0, 1, 12)`.
4. In the **Study** toolbar, click **Compute**.

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

RESULTS

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Study 1/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 Domain 4 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 (s)** list, choose **3**.

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 **AuroraAustralis**.

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

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.

Change the time and the units of the pressure plot.

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 (s)** list, choose **6**.

Surface 1

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

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

3 From the **Unit** list, choose **MPa**.

4 In the **Pressure (elte)** 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 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

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

Surface 1

1 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 `sqrt(abs(elte.I2s))*sign(elte.I2s)`.

4 From the **Unit** list, choose **MPa**.

5 Select the **Description** checkbox. In the associated text field, type `Signed square root of the second principal invariant`.

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

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

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

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

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

Loop through the different times to reproduce Figure 3.