Propagation of Seismic Waves Through Earth

Introduction

The analysis of the propagation of seismic waves through the internal structure of Earth is a complex topic. The curvature of Earth, the existence of discontinuities and the variation of the material properties with the depth create a complex interaction that makes the analysis quite challenging. This tutorial presents a method to analyze the propagation of seismic waves through Earth's internal structure. The model uses a 2D axisymmetric geometry to represent the material discontinuities and the variation of properties through the concentric layers of Earth.

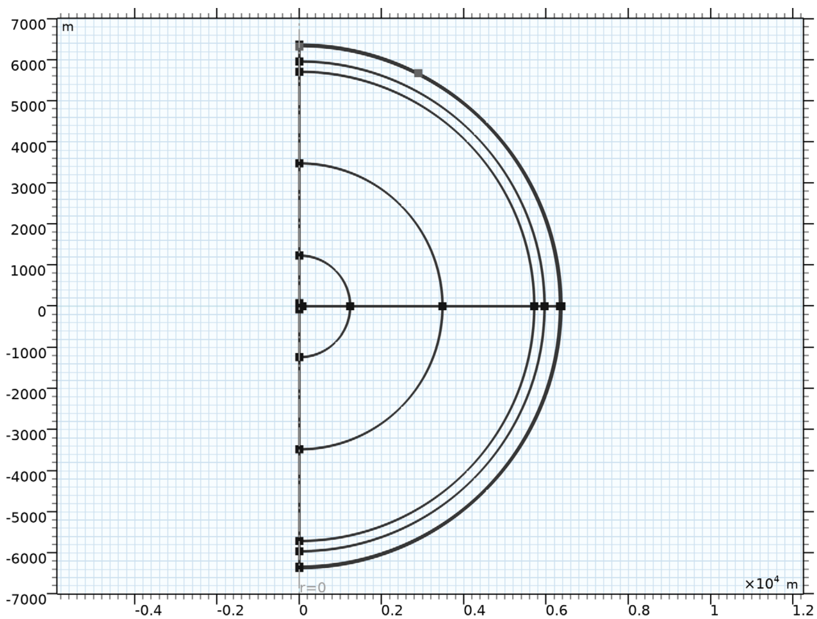


Figure 1: Geometry used in the model

The model uses the *Elastic Waves, Time Explicit* and the *Pressure Acoustics, Time Explicit* interfaces to represent the solid and fluid parts of Earth. The model demonstrates the scalability of time explicit interfaces and their applicability to capture wave propagation in large and very large acoustic models (containing many wavelengths).

Model Definition

A simplified excitation consisting of a pulse is used to study the transmission and propagation of the different pressure and shear waves across the inner structure of Earth.

This pulse, sometimes called a tone burst pulse, uses the product of a sine wave with frequency `f0` multiplied by a sine wave with a frequency of `f0/5`. Figure 2 shows this pulse, with the highest frequency content of 0.1 Hz.

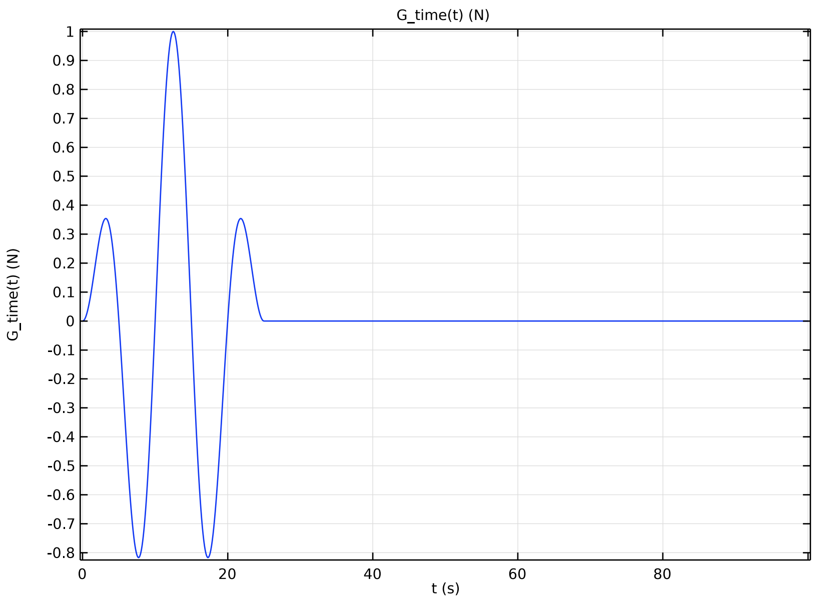


Figure 2: Tone burst load.

The variation of properties through the internal structure of Earth is taken from the Ref. 1which is also reproduced in Ref. 2. Figure 3 shows the variation of density, speed of pressure waves, and speed of shear waves with the depth. Figure 3 shows several discontinuities between the outermost layers of Earth. The area between 2900 km and 5100 km of depth is named the outer core of Earth. This outer core is made of molten iron and nickel, and therefore shear waves do not propagate across it.

d

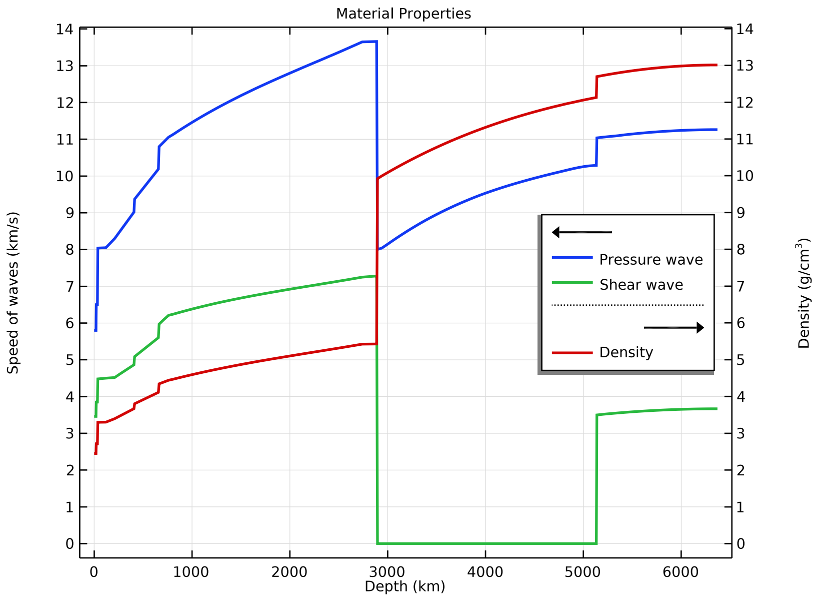


Figure 3: Variation of the material properties with depth.

The model is excited with a vertical force following the pulse previously described. This model disregards the presence of continental and oceanic crust and considers Earth as a perfect sphere with concentric layers. This means that the location (latitude and longitude) of the source is not relevant, as the representation is the same for any point on Earth.

Results and Discussion

Time explicit interfaces impose a strict limit on the time step for stability purposes. This time step is proportional to the overall smallest cell wave time scale (the variable `elte.wtc` or `pate.wtc`), shown on Figure 4. The cell wave time scale is defined as the ratio between the element size and the fastest wave propagating through the element.

The mesh needs to resolve the wavelength of the slowest wave traveling through the element as well. This requirement imposes different mesh size on different layers of Earth, which in turn translates into different cell wave time scales. As shown in Figure 4, due to the slow speed of the shear waves of the core, the elements with the smallest cell wave time scales are located in the inner core of Earth.

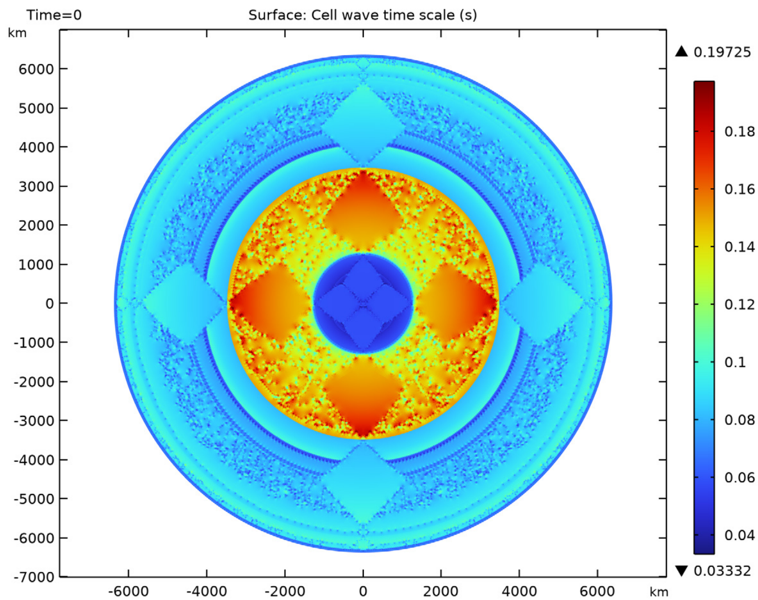


Figure 4: Cell wave time scale.

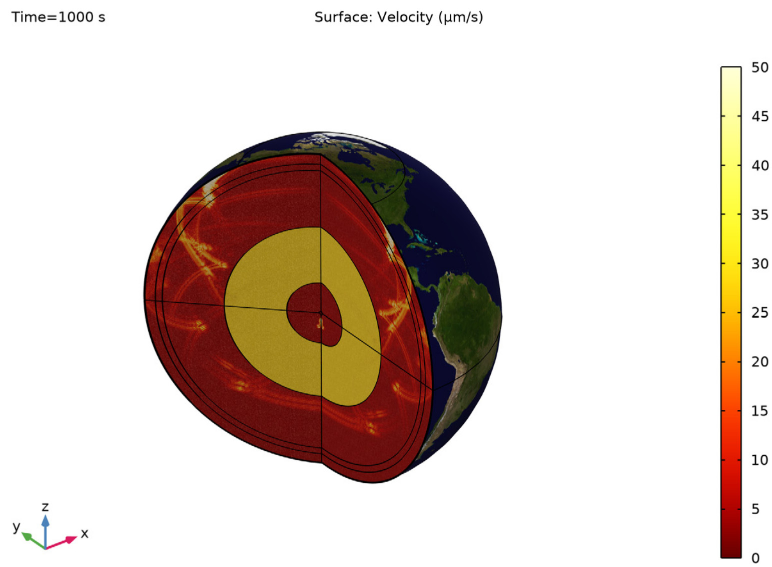


Figure 5: 3D representation of Earth showing the velocity.[1]

The model could be revolved to form a 3D image of Earth, as shown in Figure 5. Then in Figure 6, the velocity at different times of the simulation is depicted. This figure reveals a lot of information about the different waves being transmitted and reflected;

- **a** Note the two different pressure wave pulses, called p-waves in the context of seismology, the first one coming directly from the source while the second pulse coming from its reflection on Earth surface.
- **b** Two pulses of shear waves, called s-waves in the context of seismology, start to develop at the source. Both p-waves and s-waves are generated at the same time, but

1. NASA Goddard Space Flight Center Image by Reto Stöckli (land surface, shallow water, clouds). Enhancements by Robert Simmon (ocean color, compositing, 3D globes, animation). Data and technical support: MODIS Land Group; MODIS Science Data Support Team; MODIS Atmosphere Group; MODIS Ocean Group Additional data: USGS EROS Data Center (topography); USGS Terrestrial Remote Sensing Flagstaff Field Center (Antarctica); Defense Meteorological Satellite Program (city lights).

as s-waves travel approximately at half the speed of the p-waves, after a given time it is possible to see the two distinct wave fronts.

c As the p-waves travel through the mantle and reach the outer core, part of their energy is reflected back as p-waves. This is usually called the reflection from the core-mantle boundary.

d The p-waves reaching the outer core are partially transmitted as p-waves traveling through the liquid. The p-waves that have not been transmitted to the outer core or reflected back continue their travel. This leaves part of Earth without direct p-waves. This is called the *p-waves shadow zone* and it was quite relevant to identify the internal structure of Earth at the early stages of the development of seismology.

e As the s-waves reach the core-mantle boundary, part of their energy is reflected back as p-waves and s-waves. As p-waves travel faster, it is easy to identify them further away from the reflection point.

f The rest of the reflected energy is transmitted as s-waves.

g As s-waves cannot travel through fluids, the s-waves reaching the core-mantle boundary are partially reflected and partially transformed into p-waves traveling through the liquid.

h As the p-waves traveling through the outer core reach the inner core, part of their energy is converted to s-waves transmitted through the inner core. Note the short wavelength of these waves, caused by the slow speed of the s-waves in the inner core.

i The existence of discontinuities in the outer most layers create a series of refracted waves called *head waves* or *von Schmidt waves.*

j The p-waves transmitted through the inner core continue to travel forming a continuous wavefront with the p-waves that have not reached the inner core.

k The p-waves coming directly from the source finally reach the surface of Earth. This point defines one end of the *p-waves shadow zone.*

l The s-waves continue to travel through the mantle without reaching all of Earth. This area where no s-waves are transmitted is called the *s-wave shadow zone*. As the shear waves cannot travel through fluids, these are the only direct s-waves still existing.

m The p-waves reach the core-mantle boundary at the other end of Earth.

n The s-waves finally reach the surface of Earth. This point defines the start of the *s-waves shadow zone*, marked in blue.

o The *Rayleigh waves*, which are slower than the s-waves, can be seen a this point.

p The p-waves that have traveled through the core finally reach the other side of Earth, defining the other end of the *p-waves shadow zone*, marked in red.

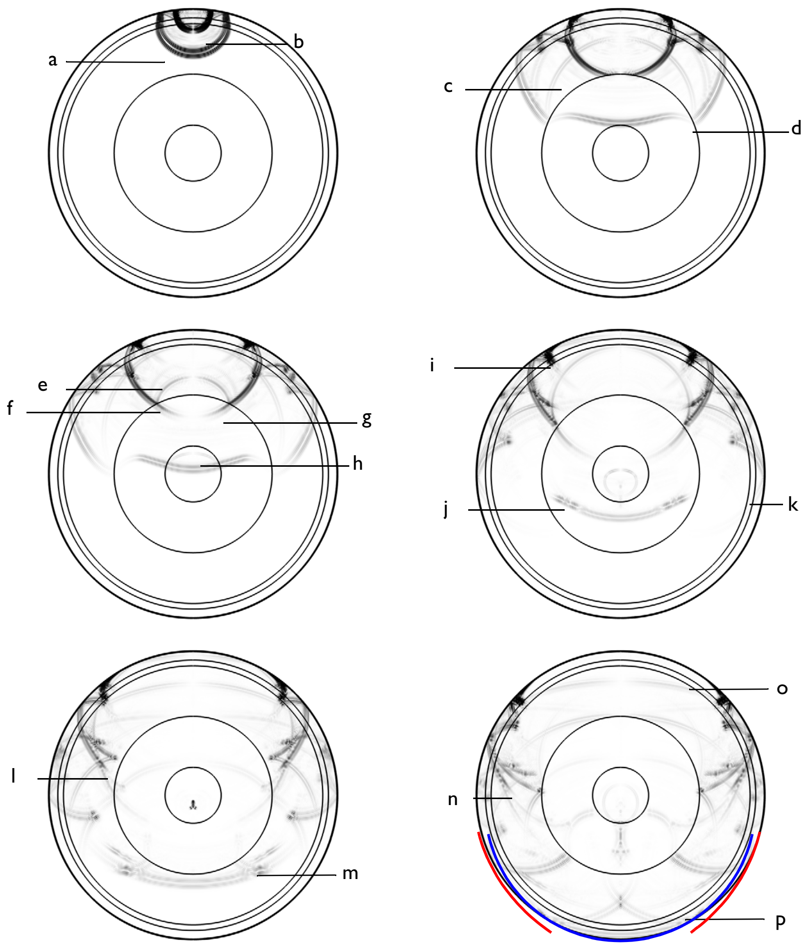


Figure 6: Velocity plot at 200 s, 500 s, 600 s, 800 s, 1000 s, and 1200 s.

The velocity experienced above the source and at the probe location is shown Figure 7. This shows how the relatively simple response at the source is transformed in a complex time signal showing the arrival of the different types of waves to the probe.

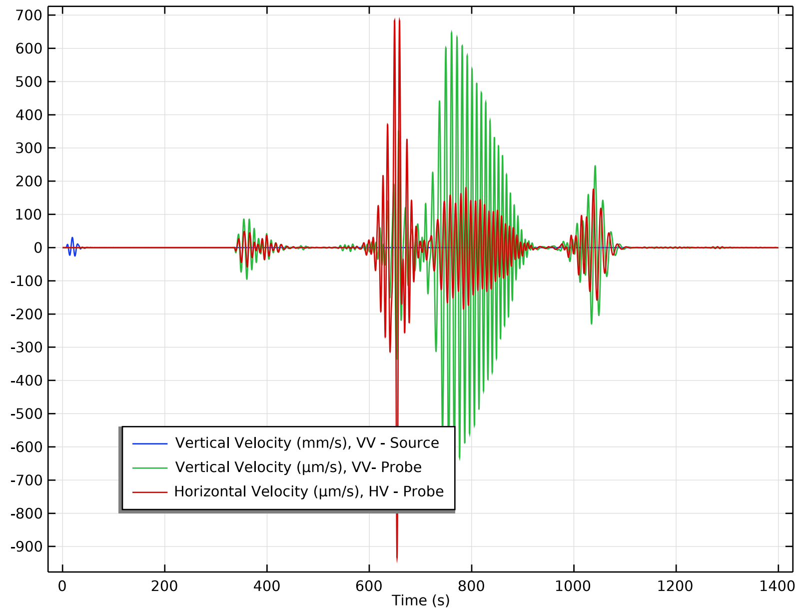


Figure 7: Probe output.

Notes About the COMSOL Implementation

The model uses material properties that vary with depth. A variable called `depth` is used to define the depth of any point in the model. Interpolation curves with linear interpolation are used to define the material properties using the expression `rho3(depth)`, for example.

The *Material Discontinuity* feature is used on all the boundaries between solid layers with sudden changes of material properties.

The mesh uses a different size for each of the layers. This size depends on the speed of the slowest waves transmitted through the layer.

Given the large size of the model, some of the variables are removed from the output. This is done to reduce the size of the model file. Stress variables are also quite useful to discern between *p-waves* and *s-waves*, as demonstrated in the *Ground Motion After Seismic Event: Scattering off a Small Mountain* tutorial.

Probes are a very useful way to obtain output that will not be stored in the solution. It is also a good method to obtain signals with more time resolution than the requested output, as by default they will be populated using all of the times used by the solver.

References

1. B.L.N. Kennett, E.R. Engdahl, and R. Buland. 1995. “Constraints on seismic velocities in the earth from travel times,” *Geophys. J. Int.*, vol. 122, pp. 108–124, https://doi.org/10.1111/j.1365-246X.1995.tb03540.x.

2. Data Services Products: EMC-ak135-f, https://doi.org/10.17611/DP/9991801.

Application Library path: `Acoustics_Module/Elastic_Waves/seismic_waves_earth`

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 Axisymmetric**.
2 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Time Explicit (pate)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Elastic Waves>Elastic Waves, Time Explicit (elte)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Time Dependent**.
8 Click **Done**.

Import the parameters from an external file.

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_waves_earth_parameters.txt`.

Create a Gaussian spatial distribution that will be used in the load.

Space Distribution

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

2 In the **Settings** window for **Analytic**, type `Space Distribution` 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 `10e26/sqrt(pi*dS)*exp(-((r - r0)^2 + (z - z0)^2)/dS)`.

5 In the **Arguments** text field, type `r,z`.

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

Argument	Unit
r	m

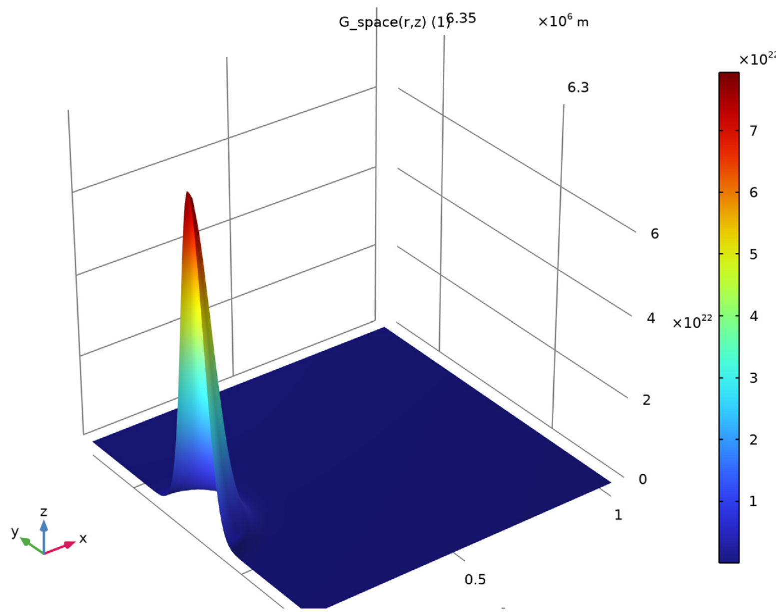
7 In the **Function** text field, type `1`.

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

Argument	Lower limit	Upper limit	Unit
r	0	100000[m]	m
z	z0-50000[m]	z0+50000[m]	

9 Click **Plot**.

The image should look like this.



Create a tone burst that will be used in the load.

Time Distribution

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

2 In the **Settings** window for **Analytic**, type `Time Distribution` 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 `if(t<2.5*T0,sin(2*pi*f0*t)*sin(2*pi*f0*t/5),0)`.

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 `N`.

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

Argument	Lower limit	Upper limit	Unit
t	0	10*T0	s

9 Click **Plot**.

The image should look like Figure 2.

Create and import tables that define the density, speed of pressure waves, and speed of shear waves of the different layers of the Earth.

Rho Layer 3

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

2 In the **Settings** window for **Interpolation**, type `Rho Layer 3` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `rho3`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_rho3.txt`.

6 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Linear**.

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

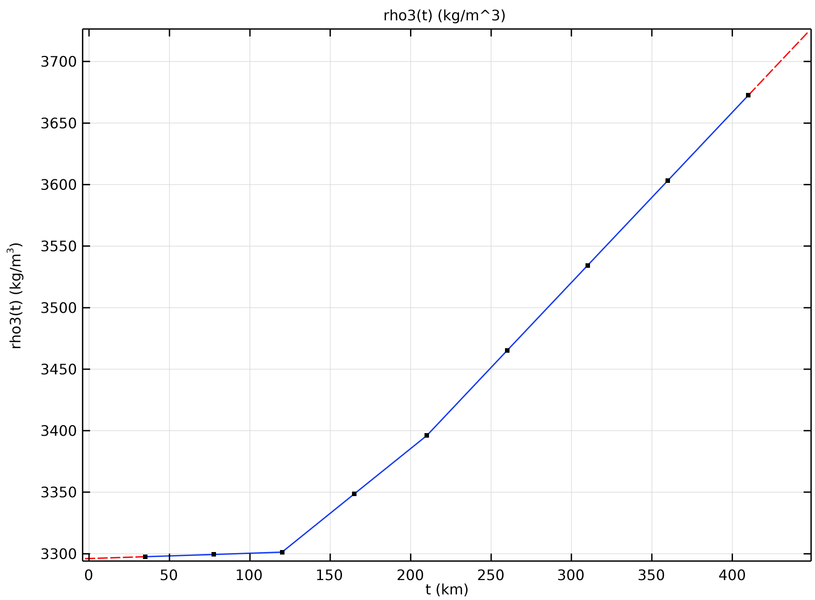
Argument	Unit
t	km

8 In the **Function** table, enter the following settings:

Function	Unit
rho3	kg/m^3

9 Click **Plot**.

The image should look like this.



Cp Layer 3

1 Right-click **Rho Layer 3** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cp Layer 3` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cp3`.

4 Click **Clear Table**.

5 Click **Load from File**.

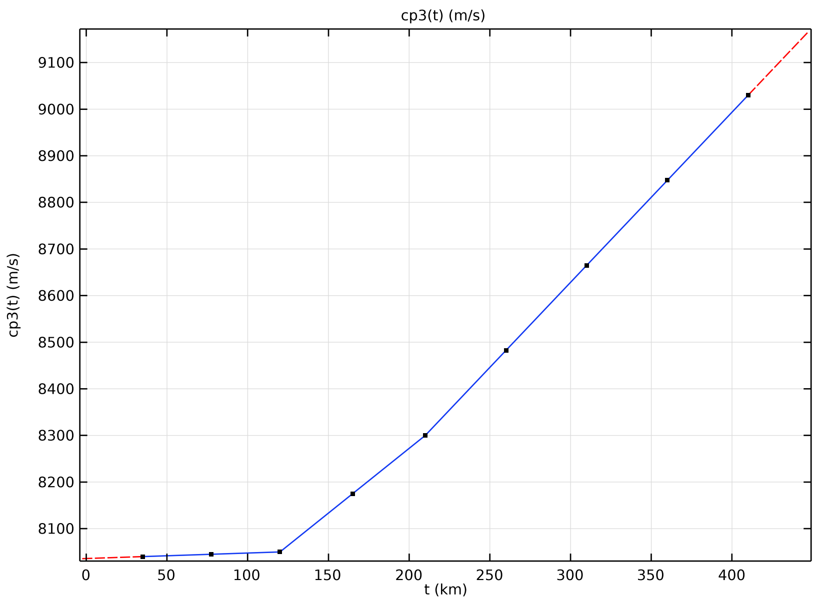
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cp3.txt`.

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

Function	Unit
cp3	m/s

8 Click **Plot**.

The image should look like this.



Cs Layer 3

1 Right-click **Cp Layer 3** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cs Layer 3` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cs3`.

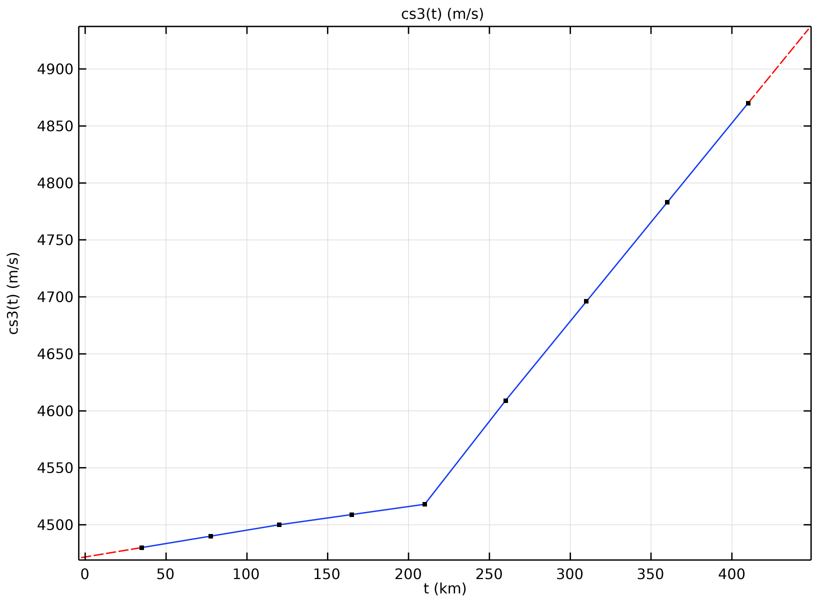
4 Click **Clear Table**.

5 Click **Load from File**.

6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cs3.txt`.

7 Click **Plot**.

The image should look like this.



Rho Layer 4

1 Right-click **Cs Layer 3** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Rho Layer 4` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `rho4`.

4 Click **Clear Table**.

5 Click **Load from File**.

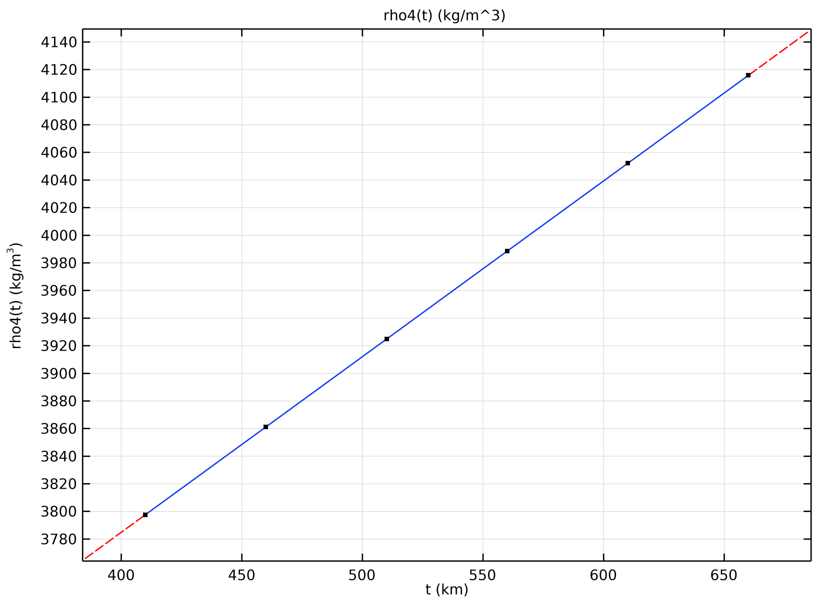
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_rho4.txt`.

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

Function	Unit
rho4	kg/m^3

8 Click **Plot**.

The image should look like this.



Cp Layer 4

1 Right-click **Rho Layer 4** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cp Layer 4` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cp4`.

4 Click **Clear Table**.

5 Click **Load from File**.

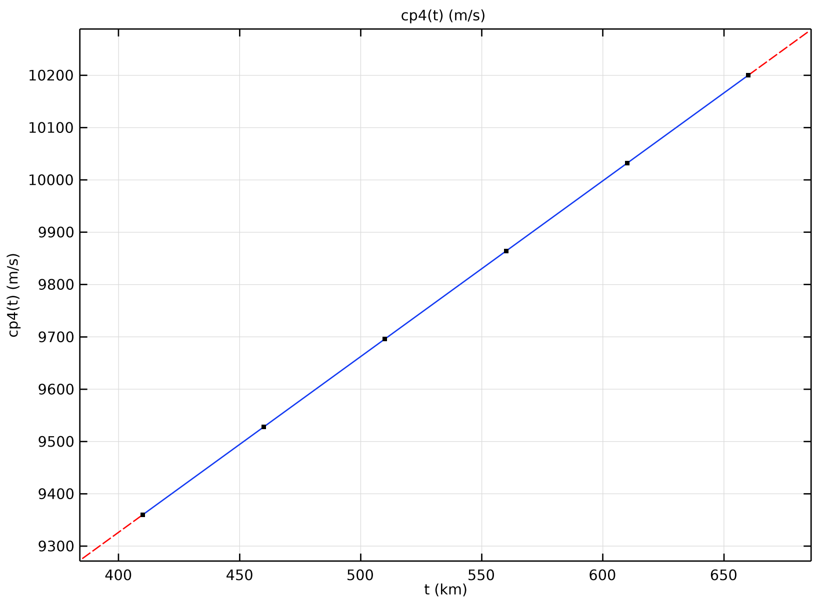
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cp4.txt`.

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

Function	**Unit**
cp4	m/s

8 Click **Plot**.

The image should look like this.



Cs Layer 4

1 Right-click **Cp Layer 4** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cs Layer 4` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cs4`.

4 Click **Clear Table**.

5 Click **Load from File**.

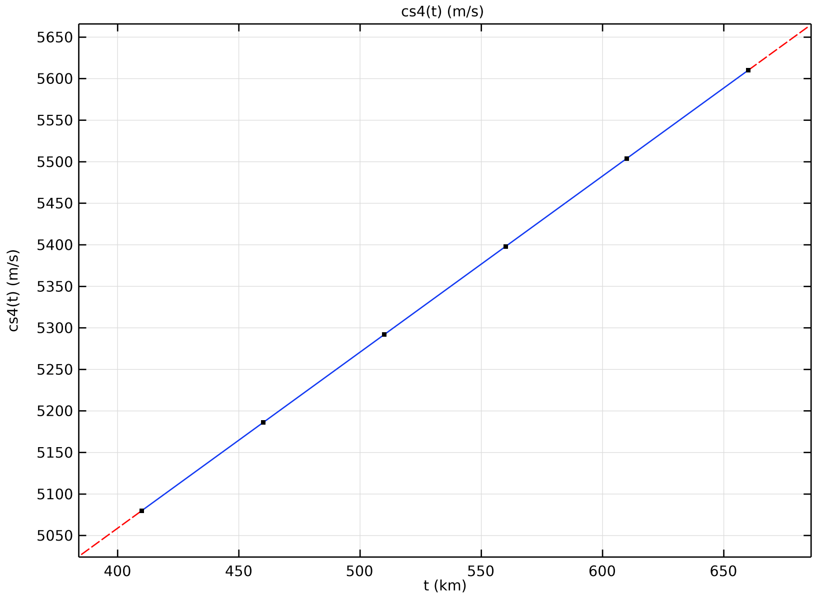
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cs4.txt`.

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

Function	**Unit**
cs4	m/s

8 Click **Plot**.

The image should look like this.



Rho Layer 5

1 Right-click **Cs Layer 4** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Rho Layer 5` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `rho5`.

4 Click **Clear Table**.

5 Click **Load from File**.

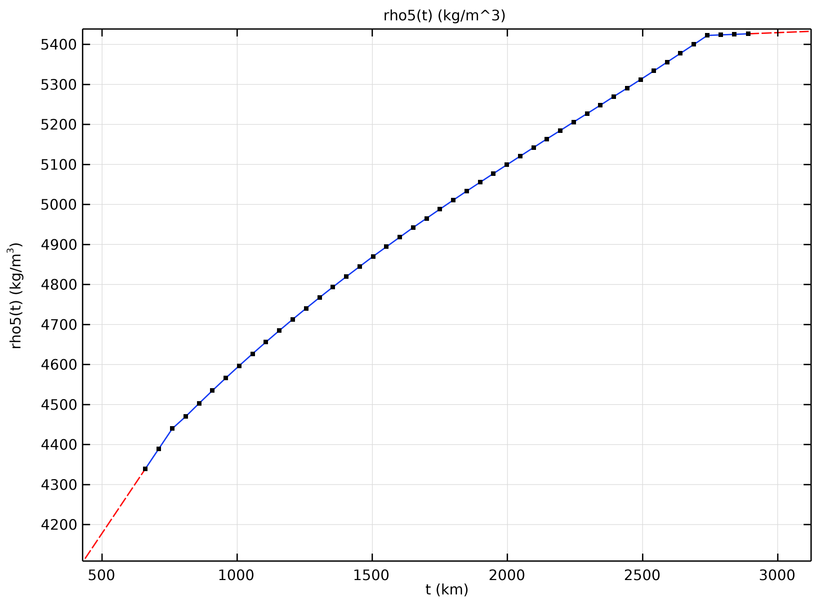
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_rho5.txt`.

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

Function	Unit
rho5	kg/m^3

8 Click **Plot**.

The image should look like this.



Cp Layer 5

1 Right-click **Rho Layer 5** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cp Layer 5` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cp5`.

4 Click **Clear Table**.

5 Click **Load from File**.

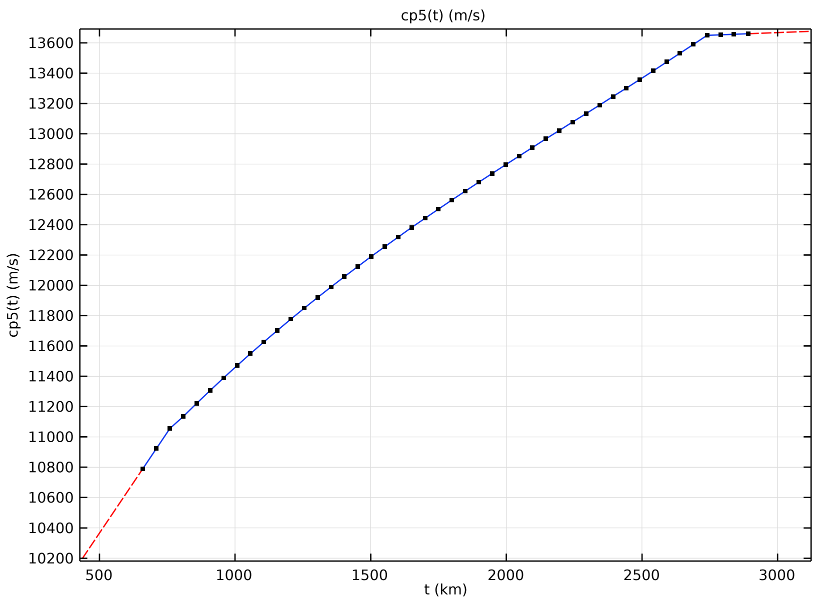
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cp5.txt`.

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

Function	**Unit**
cp5	m/s

8 Click **Plot**.

The image should look like this.



Cs Layer 5

1 Right-click **Cp Layer 5** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cs Layer 5` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cs5`.

4 Click **Clear Table**.

5 Click **Load from File**.

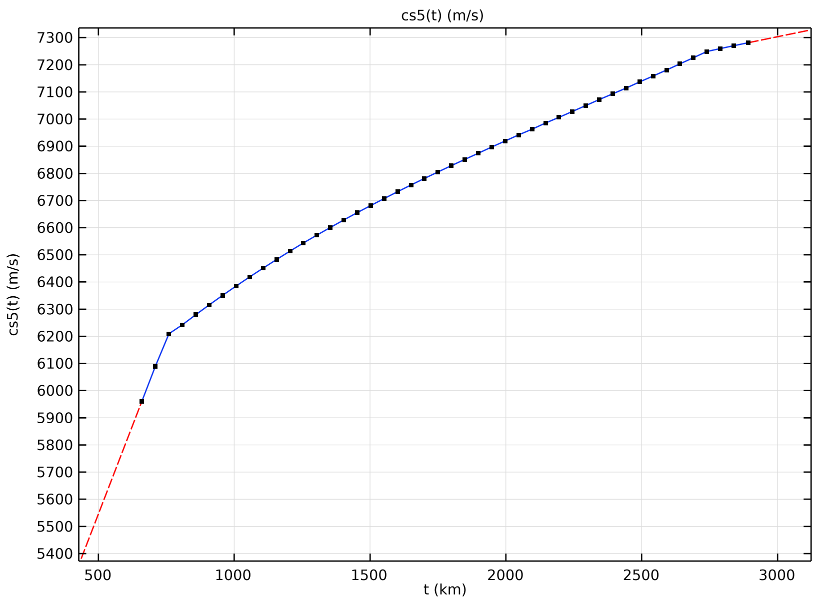
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cs5.txt`.

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

Function	Unit
cs5	m/s

8 Click **Plot**.

The image should look like this.



Rho Layer 6

1 Right-click **Cs Layer 5** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Rho Layer 6` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `rho6`.

4 Click **Clear Table**.

5 Click **Load from File**.

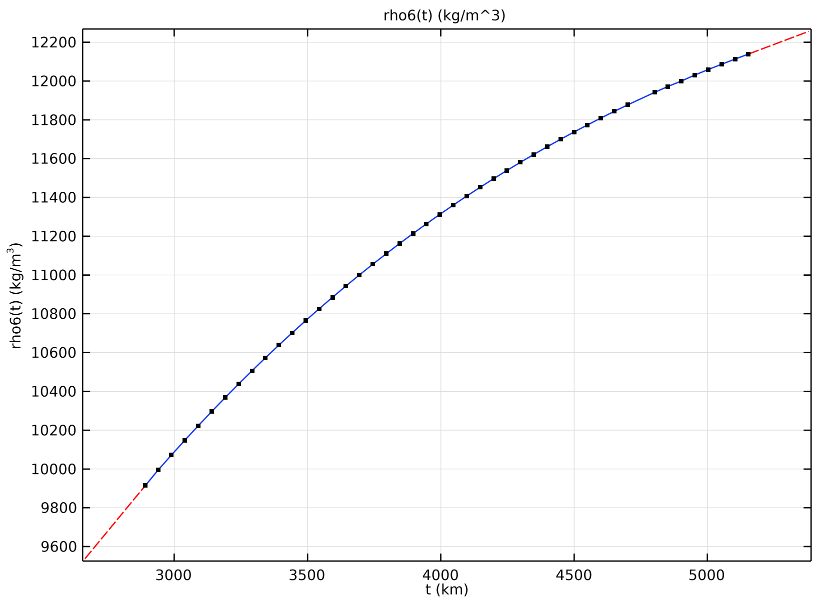
6 Browse to the model’s Application Libraries folder and double-click the file `seismic_waves_earth_rho6.txt`.

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

Function	Unit
rho6	kg/m^3

8 Click **Plot**.

The image should look like this.



Cp Layer 6

1 Right-click **Rho Layer 6** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cp Layer 6` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cp6`.

4 Click **Clear Table**.

5 Click **Load from File**.

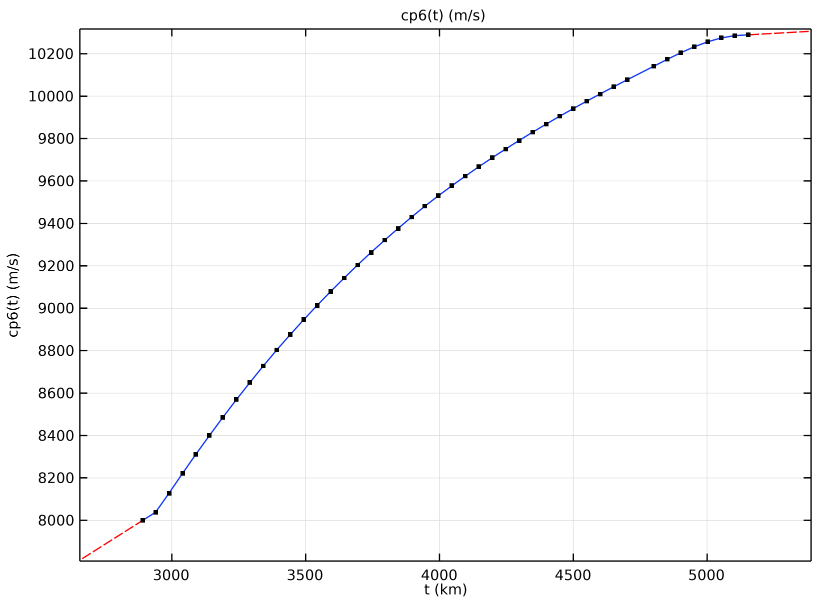
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cp6.txt`.

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

Function	Unit
cp6	m/s

8 Click **Plot**.

The image should look like this.



Rho Layer 7

1 Right-click **Cp Layer 6** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Rho Layer 7` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `rho7`.

4 Click **Clear Table**.

5 Click **Load from File**.

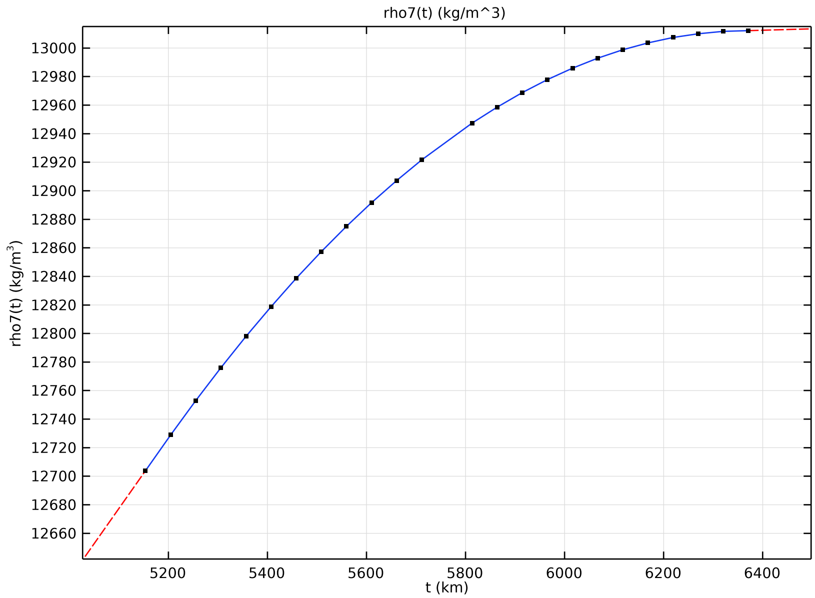
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_rho7.txt`.

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

Function	**Unit**
rho7	kg/m^3

8 Click **Plot**.

The image should look like this.



Cp Layer 7

1 Right-click **Rho Layer 7** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cp Layer 7` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cp7`.

4 Click **Clear Table**.

5 Click **Load from File**.

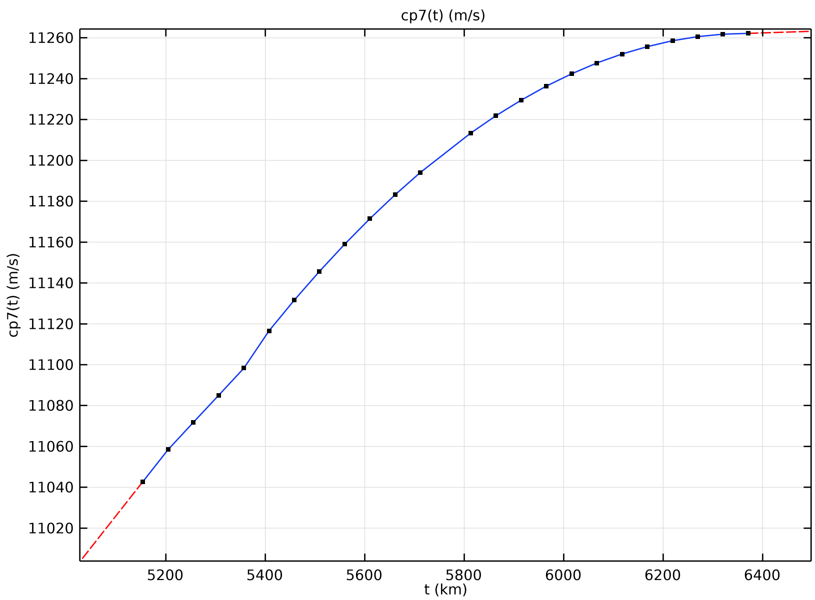
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cp7.txt`.

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

Function	Unit
cp7	m/s

8 Click **Plot**.

The image should look like this.



Cs Layer 7

1 Right-click **Cp Layer 7** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Cs Layer 7` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `cs7`.

4 Click **Clear Table**.

5 Click **Load from File**.

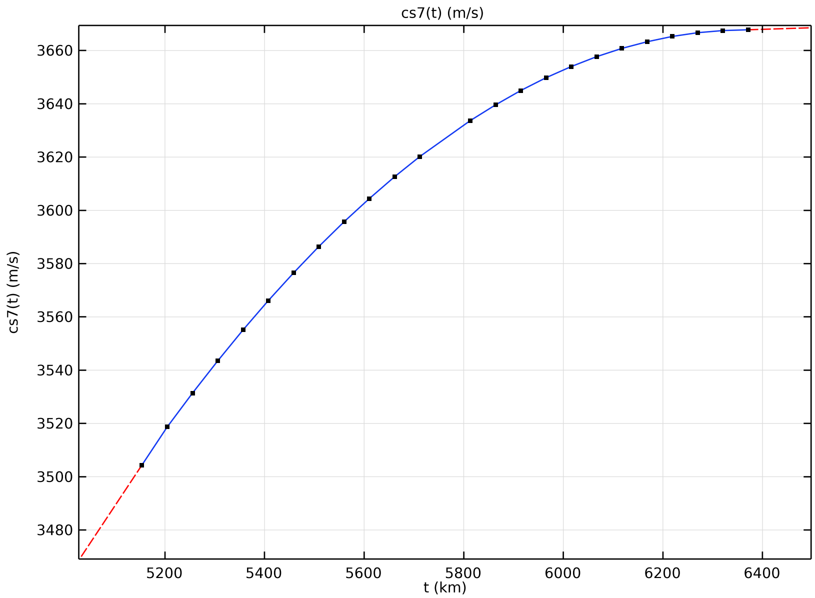
6 Browse to the model's Application Libraries folder and double-click the file `seismic_waves_earth_cs7.txt`.

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

Function	Unit
cs7	m/s

8 Click **Plot**.

The image should look like this.



Group the interpolation curves to facilitate the navigation through the model.

Cp Layer 3 (cp3), Cs Layer 3 (cs3), Rho Layer 3 (rho3)

1 In the **Model Builder** window, under **Global Definitions**, Ctrl-click to select **Rho Layer 3 (rho3)**, **Cp Layer 3 (cp3)**, and **Cs Layer 3 (cs3)**.

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

Layer 3 Properties

In the **Settings** window for **Group**, type `Layer 3 Properties` in the **Label** text field.

Cp Layer 4 (cp4), Cs Layer 4 (cs4), Rho Layer 4 (rho4)

1 In the **Model Builder** window, under **Global Definitions**, Ctrl-click to select **Rho Layer 4 (rho4)**, **Cp Layer 4 (cp4)**, and **Cs Layer 4 (cs4)**.

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

Layer 4 Properties

In the **Settings** window for **Group**, type `Layer 4 Properties` in the **Label** text field.

Cp Layer 5 (cp5), Cs Layer 5 (cs5), Rho Layer 5 (rho5)

1 In the **Model Builder** window, under **Global Definitions**, Ctrl-click to select **Rho Layer 5 (rho5)**, **Cp Layer 5 (cp5)**, and **Cs Layer 5 (cs5)**.

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

Layer 5 Properties

In the **Settings** window for **Group**, type `Layer 5 Properties` in the **Label** text field.

Cp Layer 6 (cp6), Rho Layer 6 (rho6)

1 In the **Model Builder** window, under **Global Definitions**, Ctrl-click to select **Rho Layer 6 (rho6)** and **Cp Layer 6 (cp6)**.

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

Layer 6 Properties

In the **Settings** window for **Group**, type `Layer 6 Properties` in the **Label** text field.

Cp Layer 7 (cp7), Cs Layer 7 (cs7), Rho Layer 7 (rho7)

1 In the **Model Builder** window, under **Global Definitions**, Ctrl-click to select **Rho Layer 7 (rho7)**, **Cp Layer 7 (cp7)**, and **Cs Layer 7 (cs7)**.

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

Layer 7 Properties

In the **Settings** window for **Group**, type `Layer 7 Properties` in the **Label** text field.

Now that the material properties have been added to the model, proceed to generate the geometry.

GEOMETRY 1

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

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

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

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_earth`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (km)
Layer 1	th1
Layer 2	th2
Layer 3	th3
Layer 4	th4
Layer 5	th5
Layer 6	th6
Layer 7	th7

7 Click **Build Selected**.

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

Add a point at the source location to make sure that the mesh captures this point.

Point 1 (pt1)

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

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

3 In the **r** text field, type `r0`.

4 In the **z** text field, type `z0`.

5 Click **Build Selected**.

Add a point away from the source that will be used as a probe to output the resulting pressure and velocity.

Point 2 (pt2)

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

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

3 In the **r** text field, type `r_earth*sin(phi0)`.

4 In the **z** text field, type `r_earth*cos(phi0)`.

5 Click **Build All Objects**.

The geometry should look like Figure 1.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
depth	r_earth-sqrt(r^2+z^2)	m	Depth

This variable defines the depth. The material models require this depth to obtain the material properties at a given point.

Given the size of the model, some of the variables will not be stored in the output. Create a few probes where all variables can be obtained, even those not stored in the output.

VV - Source

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type `VV - Source` in the **Label** text field.

3 Locate the **Source Selection** section. Click **Clear Selection**.

4 Select Point 18 only.

5 Locate the **Expression** section. In the **Expression** text field, type `v2z`.

Note that the probes at the source will experience a much severe movement, so it makes sense to adapt the units of the probes to account for this.

6 From the **Table and plot unit** list, choose **mm/s**.

7 Select the **Description** check box. In the associated text field, type `Vertical Velocity`.

VV- Probe

1 Right-click **VV - Source** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `VV- Probe` in the **Label** text field.

3 Locate the **Source Selection** section. Click **Clear Selection**.

4 Select Point 21 only.

5 Locate the **Expression** section. In the **Expression** text field, type `v2r*sin(phi0)+v2z*cos(phi0)`.

Change the unit of this probe as the expected pressure is a thousand times lower than the one in the source.

6 From the **Table and plot unit** list, choose **µm/s**.

HV - Probe

1 Right-click **VV- Probe** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `HV - Probe` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `v2r*cos(phi0)-v2z*sin(phi0)`.

4 Select the **Description** check box. In the associated text field, type `Horizontal Velocity`.

The outer core of the Earth is liquid, so it is captured in the model through the **Pressure Acoustics, Time Explicit** physics.

PRESSURE ACOUSTICS, TIME EXPLICIT (PATE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Time Explicit (pate)**.

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

3 Click **Clear Selection**.

4 Select Domains 6 and 10 only.

The rest of the layers of the Earth are solid, so they are captured in the model through the **Elastic Waves, Time Explicit** physics.

ELASTIC WAVES, TIME EXPLICIT (ELTE)

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

2 Select Domains 1–5, 7–9, and 11–15 only.

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 a body load that will be used as an earthquake in the model.

Body Load 1

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

2 Select Domain 13 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 $\mathbf{F}_{tot}$ vector as

`0`	`r`
`G_space(r,z)*G_time(t)`	`z`

Use the **Material Discontinuity** feature on each of the interior boundaries where there is a sudden change in material properties.

Material Discontinuity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Material Discontinuity**.

2 Select Boundaries 26–29 and 36–39 only.

Proceed to create the different materials existing in the model. The outermost and innermost layers have constant properties, while the rest of the materials will have properties that depend on depth.

MATERIALS

Layer 1

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 `Layer 1` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domains 1 and 15 only.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	`cp1`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs1`	m/s	Pressure-wave and shear-wave speeds
Density	rho	`rho1`	kg/m³	Basic

Layer 2

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

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

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domains 2 and 14 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho2`	kg/m³	Basic
Pressure-wave speed	cp	`cp2`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs2`	m/s	Pressure-wave and shear-wave speeds

Layer 3

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

2 In the **Settings** window for **Material**, type `Layer 3` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domains 3 and 13 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho3(depth)`	kg/m³	Basic
Pressure-wave speed	cp	`cp3(depth)`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs3(depth)`	m/s	Pressure-wave and shear-wave speeds

Layer 4

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

2 In the **Settings** window for **Material**, type `Layer 4` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domains 4 and 12 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`rho4(depth)`	kg/m^3	Basic
Pressure-wave speed	cp	`cp4(depth)`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs4(depth)`	m/s	Pressure-wave and shear-wave speeds

Layer 5

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

2 In the **Settings** window for **Material**, type `Layer 5` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domains 5 and 11 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	`rho5(depth)`	kg/m^3	Basic
Pressure-wave speed	cp	`cp5(depth)`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs5(depth)`	m/s	Pressure-wave and shear-wave speeds

Layer 6

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Layer 6` in the **Label** text field.

3 Select Domains 6 and 10 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho6(depth)`	kg/m³	Basic
Speed of sound	c	`cp6(depth)`	m/s	Basic

Layer 7

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Layer 5 (mat5)** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Layer 7` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domains 7 and 9 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho7(depth)`	kg/m³	Basic
Pressure-wave speed	cp	`cp7(depth)`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs7(depth)`	m/s	Pressure-wave and shear-wave speeds

Layer 8

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Layer 8` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domain 8 only.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	cp8	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	cs8	m/s	Pressure-wave and shear-wave speeds
Density	rho	rho8	kg/m³	Basic

Add the **Acoustic-Structure Boundary, Time Explicit** multiphysics coupling to connect both physics.

MULTIPHYSICS

Acoustic-Structure Boundary, Time Explicit 1 (asbte1)

1. In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic-Structure Boundary, Time Explicit**.
2. Select Boundaries 30, 31, 34, and 35 only.

Proceed to generate the mesh. To limit the size of the model, each of the layers will use a different size that is driven by the wavelength of the slowest wave traveling through that layer divided by 1.5.

MESH 1

Mapped 1

1. In the **Mesh** toolbar, click **Mapped**.
2. In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1, 2, 14, and 15 only.

Size 1

1. Right-click **Mapped 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.

   Use a slightly finer mesh in the outermost layer, as the Rayleigh waves are slightly slower than the shear waves.
4. Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `cs1/f0/2.0`.

6 Select the **Minimum element size** check box. In the associated text field, type `cs1/f0/2.0`.

7 Click **Build Selected**.

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 From the **Predefined** list, choose **Coarser**.

Convert 1

1 In the **Mesh** toolbar, click **Modify** and choose **Convert**.

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

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

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

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

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

4 Select Domains 3 and 13 only.

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

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

7 Select the **Maximum element size** check box. In the associated text field, type `cs3(th1+th2)/f0/1.5`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Domains 4 and 12 only.

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

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

7 Select the **Maximum element size** check box. In the associated text field, type `cs4(th1+th2+th3)/f0/1.5`.

Size 3

1. Right-click **Free Triangular 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 5 and 11 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `cs5(th1+th2+th3+th4)/f0/1.5`.

Size 4

1. Right-click **Free Triangular 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 6 and 10 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `cp6(th1+th2+th3+th4+th5)/f0/1.5`.

Size 5

1. Right-click **Free Triangular 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 7 and 9 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `cs7(th1+th2+th3+th4+th5+th6)/f0/1.5`.

Size 6

1. Right-click **Free Triangular 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 8 only.

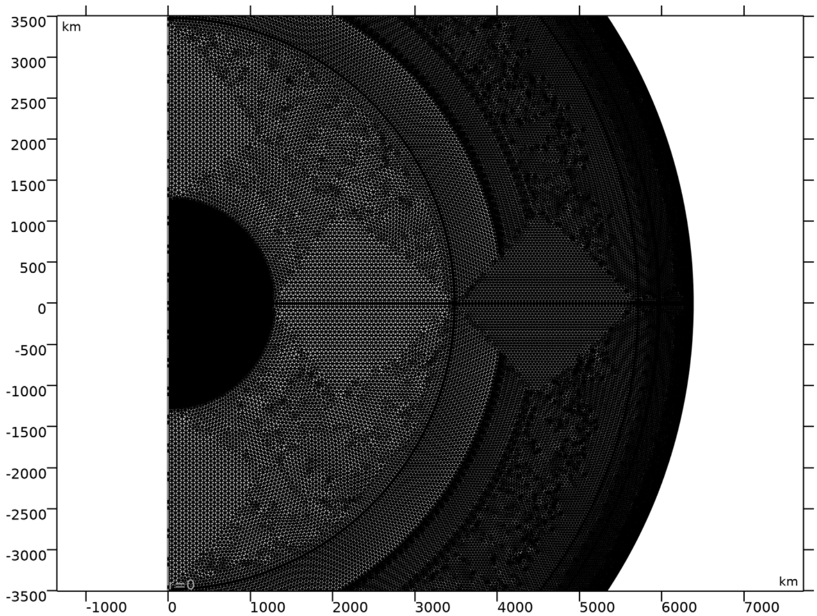
5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** check box. In the associated text field, type `cs8/f0/1.5`.

8 Click **Build All**.

The mesh should look like this.



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(0,100,1400)`.

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

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

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

7 Clear the **Generate convergence plots** check box.

8 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Results**.

2 In the **Settings** window for **Results**, locate the **Save Data in the Model** section.

3 From the **Save plot data** list, choose **On**.

Study 1/Solution 1 (3) (sol1)

1 In the **Model Builder** window, under **Results>Datasets** right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

This dataset is created only for postprocessing purposes.

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 **Boundary**.

4 Select Boundaries 25–31 and 34–41 only.

Mirror 2D 1

In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

Mirror 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

Revolution 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.

3 In the **Start angle** text field, type `-90`.

4 In the **Revolution angle** text field, type `225`.

Probe Plot

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

2 In the **Settings** window for **1D Plot Group**, type `Probe Plot` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Manual**.

4 In the **x-position** text field, type `0.15`.

5 In the **y-position** text field, type `0.1`.

Cell wave time scale

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Cell wave time scale` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.
4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.
5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.
6 Click to expand the **Number Format** section. Select the **Manual color legend settings** check box.
7 In the **Precision** text field, type `6`.

Surface 1

1 Right-click **Cell wave time scale** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `if(isnan(pate.wtc),elte.wtc,pate.wtc)`.

   This expressions shows the cell wave time scale for both physics in a single plot.
4 Select the **Description** check box. In the associated text field, type `Cell wave time scale`.
5 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.
6 In the **Cell wave time scale** toolbar, click **Plot**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

   The image should look like Figure 4.

Velocity

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

Surface 1

1 Right-click **Velocity** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 2D 1**.

4. Locate the **Expression** section. In the **Expression** text field, type `if(isnan(pate.v_inst),elte.vel,pate.v_inst)`.

   This expressions shows the velocity for both physics in a single plot.
5. From the **Unit** list, choose **µm/s**.
6. Select the **Description** check box. In the associated text field, type `Velocity`.
7. Click to expand the **Range** section. Select the **Manual color range** check box.
8. In the **Minimum** text field, type `0`.
9. In the **Maximum** text field, type `50`.
10. Locate the **Coloring and Style** section. Click **Change Color Table**.
11. In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.
12. Click **OK**.
13. In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
14. From the **Color table transformation** list, choose **Reverse**.
15. Locate the **Quality** section. From the **Resolution** list, choose **Custom**.
16. In the **Element refinement** text field, type `6`.
17. From the **Smoothing** list, choose **Everywhere**.
18. In the **Velocity** toolbar, click **Plot**.
19. Click the **Zoom Extents** button in the **Graphics** toolbar.
20. Click the **Zoom Extents** button in the **Graphics** toolbar.

Velocity (3D)

1. In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Velocity (3D)` in the **Label** text field.

Surface 1

1. Right-click **Velocity (3D)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `0`.
4. Click to expand the **Title** section. From the **Title type** list, choose **None**.
5. Click to expand the **Quality** section. From the **Resolution** list, choose **Custom**.
6. In the **Element refinement** text field, type `6`.

Image 1

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

2 In the **Settings** window for **Image**, locate the **File** section.

3 In the **Filename** text field, type `data:///physics/images/earth.jpg`.

4 Locate the **Mapping** section. From the **Mapping** list, choose **Spherical**.

5 Find the **Axis** subsection. From the **Axis type** list, choose **Cartesian**.

6 In the **x** text field, type `0.18`.

7 In the **y** text field, type `0.45`.

8 In the **z** text field, type `1`.

9 Find the **Angle** subsection. In the **Rotation** text field, type `270`.

10 In the **Velocity (3D)** toolbar, click **Plot**.

11 Click the **Show Grid** button in the **Graphics** toolbar.

Selection 1

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

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

3 Clear the **Evaluate the end caps** check box.

Surface 2

1 In the **Model Builder** window, right-click **Velocity (3D)** and choose **Surface**.

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

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

4 Select the **Description** check box. In the associated text field, type `Velocity`.

5 From the **Unit** list, choose **µm/s**.

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

7 In the **Minimum** text field, type `0`.

8 In the **Maximum** text field, type `50`.

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

10 In the **Color Table** dialog box, select **Thermal>Thermal** in the tree.

11 Click **OK**.

12 In the **Settings** window for **Surface**, locate the **Quality** section.

13 From the **Resolution** list, choose **Custom**.

14 In the **Element refinement** text field, type `6`.

Selection 1

Right-click **Surface 2** and choose **Selection**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Soil**.

5 Locate the **Color** section. Select the **Use the plot's color** check box.

Selection 1

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

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

3 Clear the **Evaluate the mantle** check box.

Surface 3

1 In the **Model Builder** window, right-click **Velocity (3D)** and choose **Surface**.

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

3 In the **Expression** text field, type `pate.v_inst`.

4 From the **Unit** list, choose **µm/s**.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Range** section. Select the **Manual color range** check box.

7 In the **Maximum** text field, type `50`.

8 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Gradient**.

9 From the **Top color** list, choose **Black**.

10 From the **Bottom color** list, choose **Custom**.

11 Click **Define custom colors**.

12 Set the RGB values to 253, 185, and 19, respectively.

13 Click **Add to custom colors**.

14 Click **Show color palette only** or **OK** on the cross-platform desktop.

15 Clear the **Color legend** check box.

Selection 1

Right-click **Surface 3** and choose **Selection**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 3** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Soil**.

5 Locate the **Color** section. Select the **Use the plot's color** check box.

Selection 1

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

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

3 Clear the **Evaluate the mantle** check box.

Probe Table 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Probe Table 1**.

2 In the **Settings** window for **Table**, locate the **Storage** section.

3 In the **Maximum number of rows** text field, type 20000.

Increase the table size as there will be more than 10000 cells in the probe table.

STUDY 1

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node.

Turn off storage of all the variables except for the velocity in both physics.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Strain tensor, Voigt notation (comp1.e)**.

3 In the **Settings** window for **Field**, locate the **General** section.

4 Clear the **Store in output** check box.

5 In the **Model Builder** window, click **Eigenvectors, structural (comp1.asbte1.veig)**.

6 In the **Settings** window for **Field**, locate the **General** section.

7 Clear the **Store in output** check box.

8 In the **Model Builder** window, click **Eigenvalues, structural (comp1.asbte1.eig)**.

9 In the **Settings** window for **Field**, locate the **General** section.

10 Clear the **Store in output** check box.

11 In the **Model Builder** window, click **Acoustic pressure (comp1.p)**.

12 In the **Settings** window for **Field**, locate the **General** section.

13 Clear the **Store in output** check box.

14 In the **Model Builder** window, click **Eigenvalues, downside (comp1.elte.mde1.eigd)**.

15 In the **Settings** window for **Field**, locate the **General** section.

16 Clear the **Store in output** check box.

17 In the **Model Builder** window, click **Eigenvalues, upside (comp1.elte.mde1.eigu)**.

18 In the **Settings** window for **Field**, locate the **General** section.

19 Clear the **Store in output** check box.

20 In the **Model Builder** window, click **Eigenvectors, downside (comp1.elte.mde1.veigd)**.

21 In the **Settings** window for **Field**, locate the **General** section.

22 Clear the **Store in output** check box.

23 In the **Model Builder** window, click **Eigenvectors, upside (comp1.elte.mde1.veigu)**.

24 In the **Settings** window for **Field**, locate the **General** section.

25 Clear the **Store in output** check box.

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**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

Plot the velocity while the analysis is computing, as this will not slow down the calculation and will allow you to check how the simulation progresses.

4 From the **Plot group** list, choose **Velocity (3D)**.

The analysis takes around 14 hours in a workstation. Time Explicit physics are quite well suited to parallel run, so it is highly recommended to run this model in a cluster using several nodes.

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

RESULTS

Material Properties

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

2 In the **Settings** window for **1D Plot Group**, type `Material Properties` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Last**.

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

5 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

6 Select the **x-axis label** check box. In the associated text field, type Depth (km).

7 Select the **y-axis label** check box. In the associated text field, type Speed of waves (km/s).

8 Select the **Secondary y-axis label** check box. In the associated text field, type Density (g/cm³).

9 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Line Graph 1

1 Right-click **Material Properties** and choose **Line Graph**.

2 Select Boundaries 1–8 only.

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

4 In the **Expression** text field, type if(isnan(pate.c),elte.cp,pate.c).

5 From the **Unit** list, choose **km/s**.

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

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type depth.

9 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

10 Click to expand the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
Pressure wave

13 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

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

15 In the **Material Properties** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type if(isnan(pate.c),elte.cs,0).

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

Legends
`Shear wave`

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `if(isnan(pate.rho),elte.rho,pate.rho)`.

4 From the **Unit** list, choose **g/cm^3**.

5 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** check box.

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

Legends
`Density`

Annotation 1

1 In the **Model Builder** window, right-click **Material Properties** and choose **Annotation**.

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

3 Select the **Plot on secondary y-axis** check box.

4 Locate the **Coloring and Style** section. Clear the **Show point** check box.

5 In the **Material Properties** toolbar, click **Plot**.

Annotation 2

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

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

3 In the **z** text field, type `13.65`.

4 In the **Material Properties** toolbar, click **Plot**.

The image should look like Figure 3.

Cycle through the different plot groups to reproduce the figures in the results section.

Probe Plot

1 In the **Model Builder** window, under **Results** click **Probe Plot**.

2 In the **Probe Plot** toolbar, click **Plot**.

The results should look like Figure 7.

Surface 1

1 In the **Model Builder** window, under **Results>Velocity** click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Solution parameters** list, choose **From parent**.
4 In the **Velocity** toolbar, click **Plot**.

   Cycle through the different times to reproduce the Figure 6.

Velocity

1 In the **Model Builder** window, click **Velocity**.
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3 From the **Time (s)** list, choose **1400**.
4 In the **Velocity** toolbar, click **Plot**.

Velocity (3D)

1 In the **Model Builder** window, click **Velocity (3D)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Time (s)** list, choose **1000**.
4 In the **Velocity (3D)** toolbar, click **Plot**.

   The results should look like Figure 5.