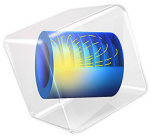

Model created in COMSOL Multiphysics 6.4

Scattered-Field Formulation for Elastic Waves

Introduction

This model showcases how to solve for the scattered field when knowing the incident field for three different types of scatterer, that is, an infinitely rigid one, a cavity, and an elastic inclusion.

This formulation can be useful when the scatterer is in the far field of the source, so that the probing wave resembles a plane wave. In such cases, including the source would require a huge computational domain to be meshed. Both P and S plane waves are thus used as incident fields in the model.

Moreover, the model shows how to numerically compute the field emitted by a point source and how to use the obtained solution as the known incident field for a subsequent study solving the scattering problem.

Model Definition

The model geometry consists of an obstacle shaped as an infinite cylinder embedded in an infinite background consisting of an elastic material. Focusing the attention on P and S waves, the plane-strain assumption holds and you can simply model a plane perpendicular to the axis of the cylindrical object. The infinite extension of the background elastic material can be simulated by adopting a finite computational domain truncated with perfectly matched layers. The radius of the obstacle is arbitrarily selected to be unitary, and the frequency is selected such that the wavelength of the P waves is unitary too.

DOMAIN EQUATIONS

The field equation for the background field reads

$$\rho_b \frac{\partial^2 \mathbf{u}_b}{\partial t^2} = \nabla \cdot (C_b \nabla \mathbf{u}_b)$$

The field equation for the total field reads

$$\rho \frac{\partial^2 \mathbf{u}_t}{\partial t^2} = \nabla \cdot (C \nabla \mathbf{u}_t)$$

where the elasticity tensor and the density are equal to the those of the background outside the scatterer and equal to those of the obstacle inside it. Defining the scattered field as

$$\mathbf{u}_t = \mathbf{u}_b + \mathbf{u}_s$$

and subtracting the field equation for the background from that of the total gives

$$\rho_b \frac{\partial^2 \mathbf{u}_s}{\partial t^2} = \nabla \cdot (C_b \nabla \mathbf{u}_s)$$

outside the scatterer, while

$$\rho_o \frac{\partial^2 \mathbf{u}_s}{\partial t^2} = (\rho_b - \rho_o)\frac{\partial^2 \mathbf{u}_b}{\partial t^2} + \nabla \cdot (C_o \nabla(\mathbf{u}_s + \mathbf{u}_b)) - (\nabla \cdot (C_b \nabla \mathbf{u}_b))$$

inside it. This equation can be rewritten as

$$\rho_o \frac{\partial^2 \mathbf{u}_s}{\partial t^2} = \mathbf{f} + \nabla \cdot (C_o(\nabla \mathbf{u}_s - \varepsilon) + \sigma)$$

Thus, one can solve for the scattered field by including in the standard elastodynamic problem a body force, an initial strain, and an initial stress according to

$$\mathbf{f} = (\rho_b - \rho_o)\frac{\partial^2 \mathbf{u}_b}{\partial t^2}$$

$$\sigma = -C_b \nabla \mathbf{u}_b$$

$$\varepsilon = -\frac{1}{2}(\nabla \mathbf{u}_b + \nabla \mathbf{u}_b^{\mathrm{T}})$$

BOUNDARY CONDITIONS

An infinitely rigid obstacle is one for which no displacements can occur and it can therefore be simulated using a homogeneous Dirichlet (fixed) boundary condition:

$$\mathbf{u}_t = \mathbf{u}_b + \mathbf{u}_s = 0$$

Rewriting this equation in terms of the scattered field, one obtains

$$\mathbf{u}_s = -\mathbf{u}_b$$

A cavity is such that its boundaries are stress free, which corresponds to a homogeneous Neumann condition:

$$\sigma_t \cdot \mathbf{n} = (\sigma_b + \sigma_s) \cdot \mathbf{n} = 0$$

Thus

$$\sigma_s \cdot \mathbf{n} = -(\sigma_b \cdot \mathbf{n})$$

Results

Figure 1 shows the displacement magnitude of the scattered field as obtained when a unitary-amplitude plane wave impinges on the three different types of obstacles.

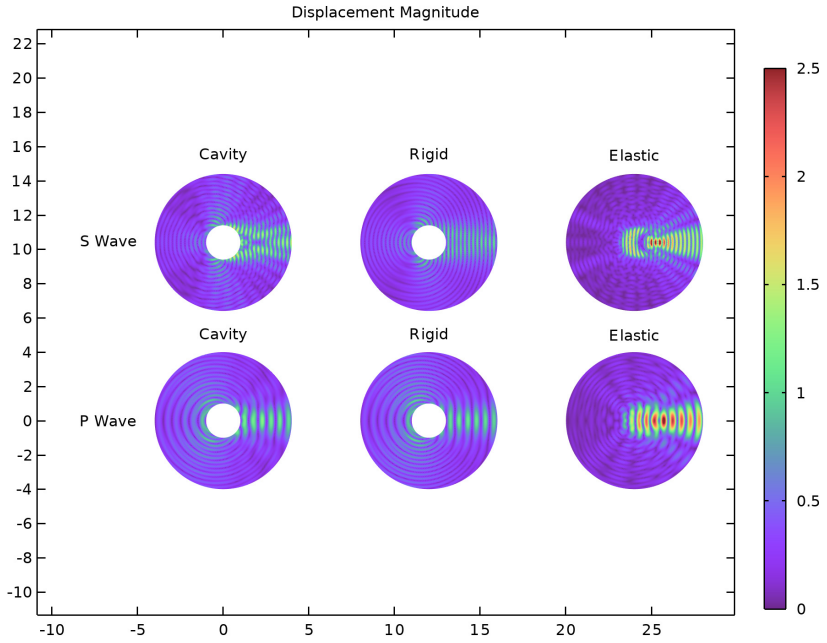

Figure 1: Displacement magnitude of the scattered field for unitary-amplitude incident S (top row) and P (bottom row) plane waves for different types of obstacles.

The top row is the field obtained for an incident S wave, while the bottom row shows the scattered field for a P wave. Exploiting the fact that S waves are divergence free and P waves are curl free, the scattered field can be separated into a P and an S contribution for each analyzed case.

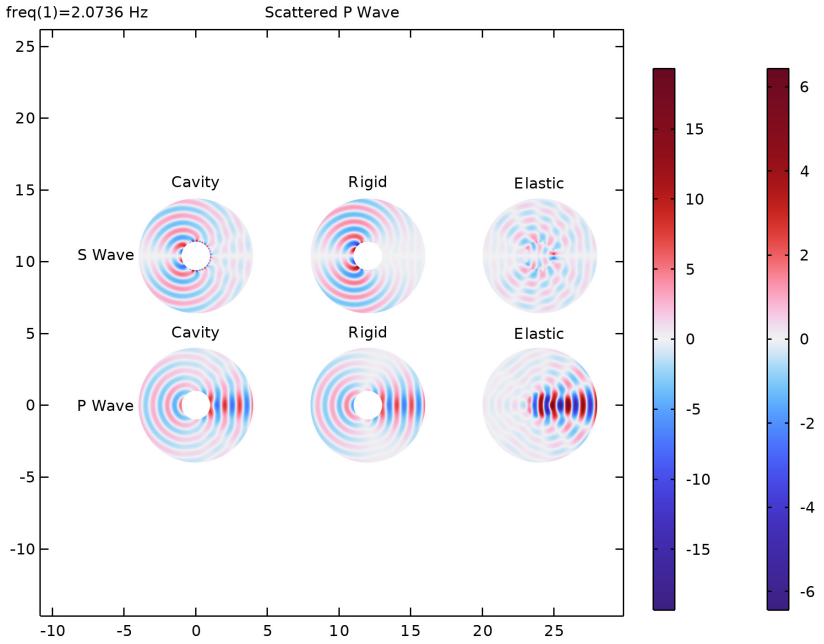

Figure 2: P wave scattered for incident S (top row) and P (bottom row) plane waves for different types of obstacles.

Figure 2 shows the scattered P wave, while Figure 3 shows the scattered S wave.

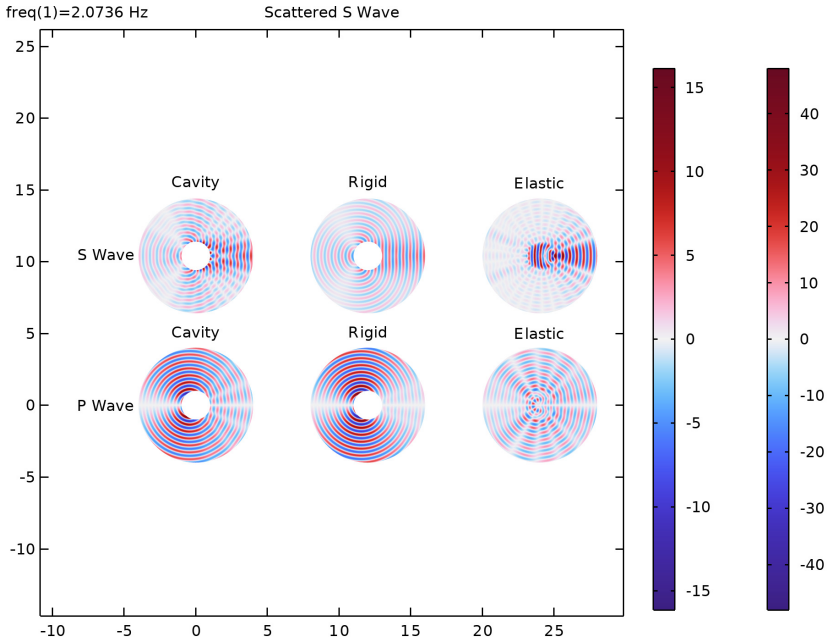

Figure 3: S wave scattered for incident S (top row) and P (bottom row) plane waves for different types of obstacles.

Adding the known incident field to the solved scattered field gives the total field shown in Figure 4, Figure 5, and Figure 6 in terms of the displacement magnitude, P wave, and S wave, respectively.

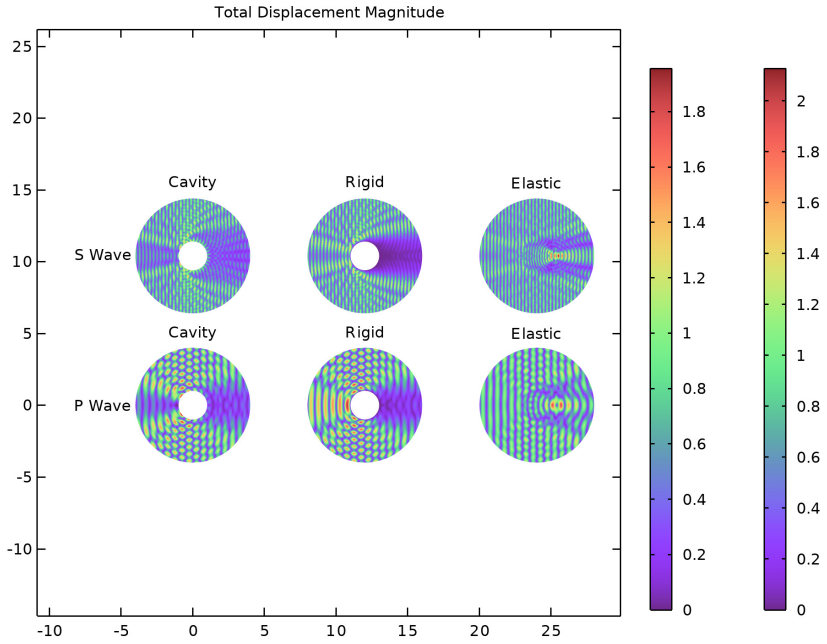

Figure 4: Total displacement magnitude for unitary-amplitude incident S (top row) and P (bottom row) plane waves for different types of obstacles.

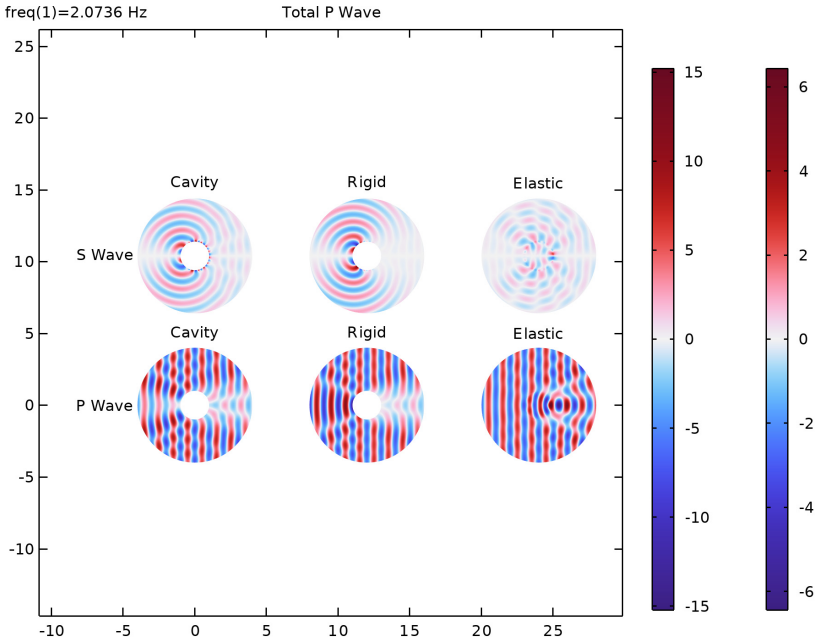

Figure 5: Total P wave field obtained for unitary-amplitude incident S (top row) and P (bottom row) plane waves for different types of obstacles.

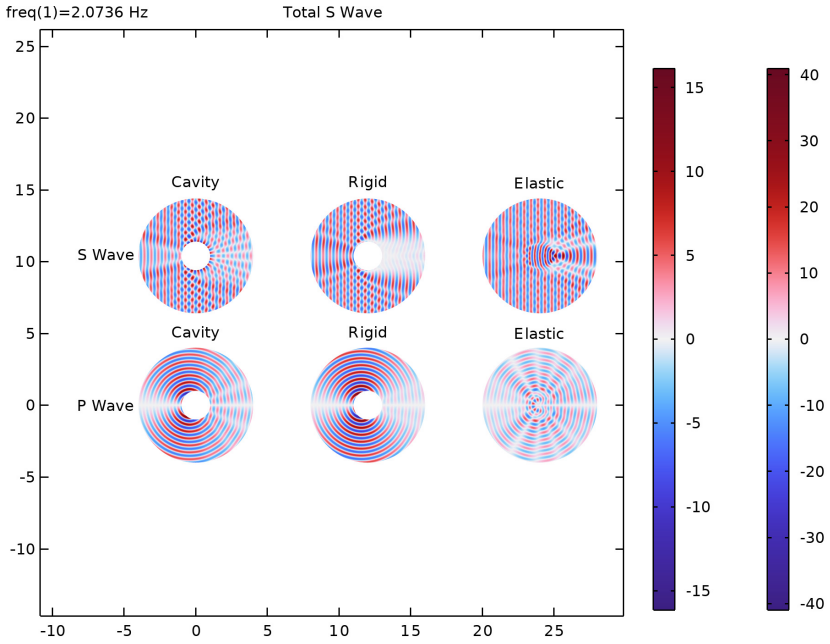

Figure 6: Total S wave field obtained for unitary-amplitude incident S (top row) and P (bottom row) plane waves for different types of obstacles.

Figure 7 shows instead the field produced by a point source in a homogeneous infinitely extended medium. This is used as an incident field to find the fields shown in Figure 8. In Figure 9, the total field obtained summing the scattered field and the incident field shown in Figure 7 is compared to the solution of the same problem solved without the scattered field formulation (SFF) using the point source along with the field equation and boundary conditions for the total field directly.

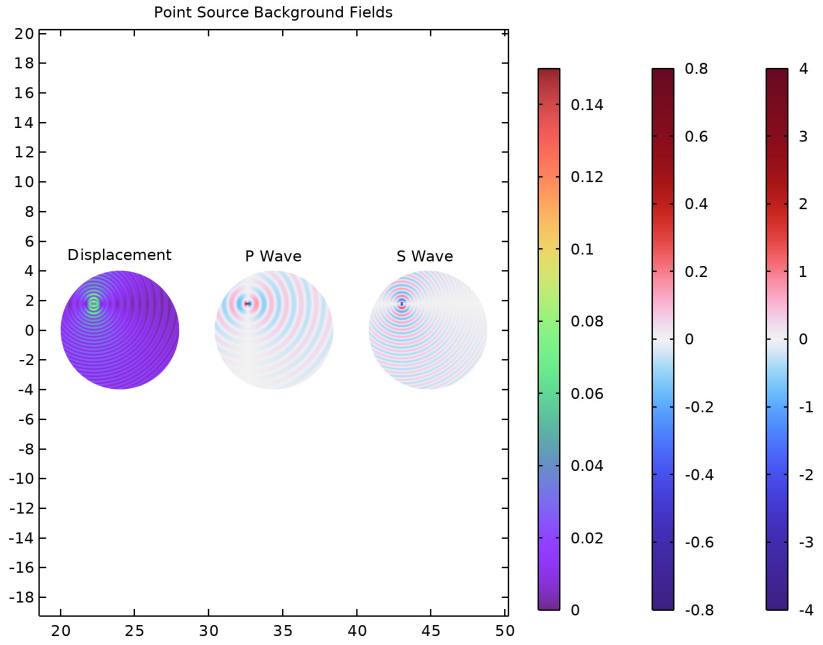

Figure 7: Point source in terms of displacement magnitude and P and S waves.

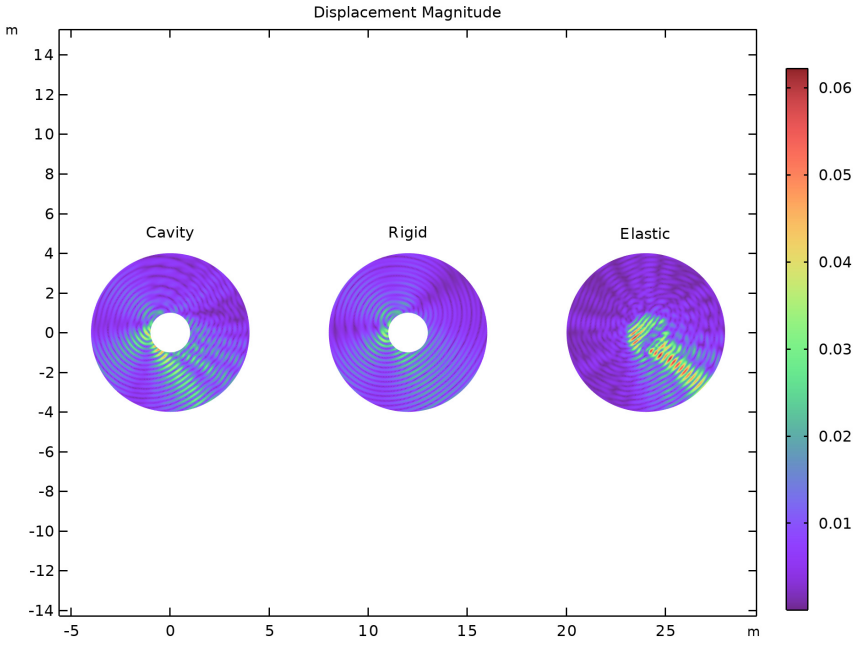

Figure 8: Scattered field produced by a point source for different types of obstacles.

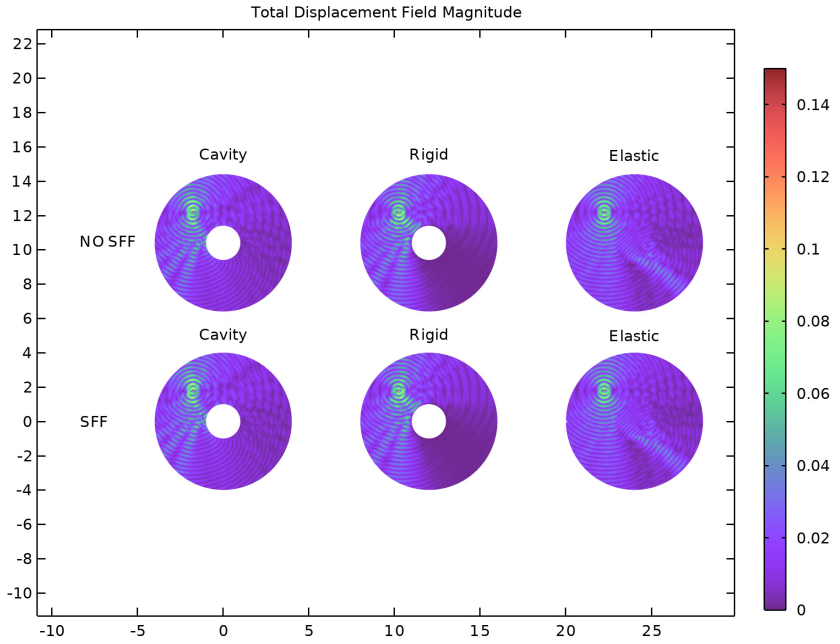

Figure 9: Total field obtained with a point source and different types of obstacles. The top row shows the results obtained without the scattered field formulation (SFF), while the bottom row shows the results obtained summing the incident field to the scattered solved with the SFF.

Application Library path: `Structural_Mechanics_Module/Elastic_Waves/scattered_field_elastic_waves`

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** > **Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Material Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

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

Name	Expression	Value	Description
rho	1[kg/m^3]	1 kg/m³	Density of background material
muLame	1[Pa]	1 Pa	Shear modulus of background material
lambLame	2.3[Pa]	2.3 Pa	First Lamé parameter of background material
rho_o	2[kg/m^3]	2 kg/m³	Density of elastic obstacle
muLame_o	1.1[Pa]	1.1 Pa	Shear modulus of elastic obstacle
lambLame_o	2.5[Pa]	2.5 Pa	First Lamé parameter elastic obstacle

P Wave Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `P Wave Parameters` in the **Label** text field.

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

Name	Expression	Value	Description
cP	sqrt((lambLame+2*muLame)/rho)	2.0736 m/s	Speed of P waves
wlengthP	1[m]	1 m	Wavelength
kP	2*pi[rad]/wlengthP	6.2832 rad/m	Wave number
omega	kP*cP	13.029 rad/s	Angular frequency

S Wave Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `S Wave Parameters` in the **Label** text field.

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

Name	Expression	Value	Description
cS	sqrt(muLame/rho)	1 m/s	Speed of S waves
kS	omega/cS	13.029 rad/m	Wave number
wlengthS	2*pi/kS	0.48224 m	Wavelength

Geometrical Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `Geometrical Parameters` in the **Label** text field.

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

Name	Expression	Value	Description
R	5[m]	5 m	Radius of computational domain
r_o	1[m]	1 m	Radius of inclusion
r_layer	1[m]	1 m	Thickness of PML

GEOMETRY 1

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`.

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

Layer name	Thickness (m)
Layer 1	r_layer
Layer 2	R-r_layer-r_o

5 Click **Build Selected**.

Add more copies of the computational domain in order to solve for cavity inclusion, infinitely rigid obstacle, and elastic inclusion at the same time.

Array 1 (arr1)

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

2 Select the object **c1** only.

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

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `3`.

6 Locate the **Displacement** section. In the **x** text field, type `2*(R+r_o)`.

7 Click **Build Selected**.

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

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 **arr1(1)**, select Domain 9 only.

5 On the object **arr1(2)**, select Domain 9 only.

6 Click **Build All Objects**.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Perfectly Matched Layer**.

3 Select Domains 1, 2, 5, and 8 only.

4 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

5 From the **Type** list, choose **Cylindrical**.

ARTIFICIAL DOMAINS

Perfectly Matched Layer 2 (pml2)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 9, 10, 13, and 16 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

Perfectly Matched Layer 3 (pml3)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 17, 18, 21, and 24 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

First, set up the simulation for an incident P wave.

DEFINITIONS

Incident P Wave

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

2 In the **Settings** window for **Variables**, type `Incident P Wave` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
uP	`exp(-1i*kP*x-1i*pi/2+1i*phase)[m]`	m	`Incident wave: u field`
vP	`0[m]`	m	`Incident wave: v field`
eps11P	`d(uP,x)`		`Incident wave: strain tensor, 11 component`
eps22P	`d(vP,y)`		`Incident wave: strain tensor, 22 component`
eps12P	`0.5*(d(uP,y)+d(vP,x))`		`Incident wave: strain tensor, 12 component`
s11P	`(lambLame+2*muLame)*eps11P+lambLame*eps22P`	Pa	`Incident wave: stress tensor, 11 component`
s22P	`lambLame*eps11P+(lambLame+2*muLame)*eps22P`	Pa	`Incident wave: stress tensor, 22 component`
s12P	`2*muLame*eps12P`	Pa	`Incident wave: stress tensor, 12 component`

The internal variable **phase** is used to synchronize the incident field with the solved scattered field in plots and animations over the dynamic data extension.

Background Material

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

3 Locate the **Linear Elastic Material** section. From the **Specify** list, choose **Pressure-wave and shear-wave speeds**.

4 From the c_p list, choose **User defined**. In the associated text field, type `cP`.

5 From the c_s list, choose **User defined**. In the associated text field, type `cS`.

6 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

Add the boundary condition for a cavity inclusion.

Cavity Inclusion, P Wave

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

2 In the **Settings** window for **Boundary Load**, type `Cavity Inclusion, P Wave` in the **Label** text field.

3 Select Boundaries 29, 30, 33, and 34 only.

4 Locate the **Force** section. Specify the $\mathbf{f}_A$ vector as

`-(s11P*solid.nx+s12P*solid.ny)`	`x`
`-(s12P*solid.nx+s22P*solid.ny)`	`y`

Now add the boundary condition for an infinitely rigid inclusion.

Infinitely Rigid Inclusion, P Wave

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 In the **Settings** window for **Prescribed Displacement**, type `Infinitely Rigid Inclusion, P Wave` in the **Label** text field.

3 Select Boundaries 41, 42, 45, and 46 only.

4 Locate the **Prescribed Displacement** section. From the **Displacement in x direction** list, choose **Prescribed**.

5 In the u_{0x} text field, type `-uP`.

6 From the **Displacement in y direction** list, choose **Prescribed**.

7 In the u_{0y} text field, type `-vP`.

Finally, set up the domain equation for the elastic inclusion.

Elastic Inclusion, P Wave

1 In the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 In the **Settings** window for **Linear Elastic Material**, type `Elastic Inclusion, P Wave` in the **Label** text field.

3 Select Domain 25 only.

4 Locate the **Linear Elastic Material** section. From the **Specify** list, choose **Lamé parameters**.

5 From the λ list, choose **User defined**. In the associated text field, type `lambLame_o`.

6 From the μ list, choose **User defined**. In the associated text field, type `muLame_o`.

7 From the ρ list, choose **User defined**. In the associated text field, type `rho_o`.

Add the initial stress and strain computed from the background field.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 Specify the S_0 matrix as

`-s11P`	`-s12P`	0
`-s12P`	`-s22P`	0
0	0	0

4 Specify the ε_0 matrix as

`-eps11P`	`-eps12P`	0
`-eps12P`	`-eps22P`	0
0	0	0

Add also the body force.

Body Load (Elastic Inclusion), P Wave

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

2 In the **Settings** window for **Body Load**, type `Body Load (Elastic Inclusion), P Wave` in the **Label** text field.

3 Select Domain 25 only.

4 Locate the **Force** section. Specify the $\mathbf{f}_V$ vector as

`(rho-rho_o)*(-omega^2)*uP`	x
`(rho-rho_o)*(-omega^2)*vP`	y

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, 5, 8–10, 13, 16–18, 21, and 24 only.

Free Triangular 1

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

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 `wlengthS/6`.

5 In the **Minimum element size** text field, type `wlengthS/8`.

6 Click  **Build All**.

P WAVE

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

2 In the **Settings** window for **Study**, type `P Wave` in the **Label** text field.

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **P Wave** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `omega/2/pi[rad]`.

P WAVE

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

2 Right-click **P Wave** and choose **Compute**.

RESULTS

Scattered u Field

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

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

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

4 In the **Title** text area, type `Displacement Field, X-component`.

5 Clear the **Parameter indicator** text field.

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **Scattered u Field** and choose **Surface**.

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

3 In the **Expression** text field, type `u`.

4 Locate the **Coloring and Style** section. From the **Scale** list, choose **Linear symmetric**.

Selection 1

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

2 Select Domains 3, 4, 6, 7, 11, 12, 14, 15, 19, 20, 22, 23, and 25 only.

Scattered u Field

In the **Model Builder** window, under **Results** click **Scattered u Field**.

Table Annotation 1

1 In the **Scattered u Field** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	5	Cavity
12	5	Rigid
24	5	Elastic

5 In the **Scattered u Field** toolbar, click **Plot**.

6 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Center**.

7 Clear the **Show point** checkbox.

Surface 1

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

2 In the **Model Builder** window, collapse the **Surface 1** node.

RESULTS

Scattered u Field

In the **Model Builder** window, collapse the **Results** > **Scattered u Field** node.

Scattered v Field

1 In the **Model Builder** window, right-click **Scattered u Field** and choose **Duplicate**.

2 In the **Model Builder** window, click **Scattered u Field 1**.

3 In the **Settings** window for **2D Plot Group**, type `Scattered v Field` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Displacement Field, Y-component`.

Surface 1

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

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

3 In the **Expression** text field, type `v`.

4 In the **Scattered v Field** toolbar, click  **Plot**.

RESULTS

Scattered v Field

In the **Model Builder** window, collapse the **Results** > **Scattered v Field** node.

Scattered Displacement Field Magnitude

1 In the **Model Builder** window, right-click **Scattered v Field** and choose **Duplicate**.

2 In the **Model Builder** window, click **Scattered v Field 1**.

3 In the **Settings** window for **2D Plot Group**, type `Scattered Displacement Field Magnitude` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Displacement Magnitude`.

Surface 1

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

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

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

4 In the **Scattered Displacement Field Magnitude** toolbar, click **Plot**.

5 Locate the **Coloring and Style** section. From the **Scale** list, choose **Linear**.

6 From the **Color table** list, choose **SpectrumLight**.

Scattered Displacement Field Magnitude 1

In the **Model Builder** window, right-click **Scattered Displacement Field Magnitude** and choose **Duplicate**.

Scattered Displacement Field Magnitude

In the **Model Builder** window, collapse the **Results** > **Scattered Displacement Field Magnitude** node.

Scattered P Wave

Apply the divergence to the displacement field to highlight P waves only.

1 In the **Model Builder** window, under **Results** click **Scattered Displacement Field Magnitude 1**.

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

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Surface 1

1 In the **Model Builder** window, expand the **Scattered P Wave** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `d(u,x)+d(v,y)`.

4 In the **Scattered P Wave** toolbar, click **Plot**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

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

7 In the **Scattered P Wave** toolbar, click **Plot**.

Apply the curl to the displacement field to highlight S waves only.

Scattered S Wave

1 In the **Model Builder** window, right-click **Scattered P Wave** and choose **Duplicate**.

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

Scattered P Wave

In the **Model Builder** window, collapse the **Results** > **Scattered P Wave** node.

Surface 1

1 In the **Model Builder** window, expand the **Results** > **Scattered S Wave** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Displacement** > **Curl of displacement (material and geometry frames)** > **solid.curlUZ - Curl of displacement, Z-component**.

3 In the **Scattered S Wave** toolbar, click **Plot**.

RESULTS

Scattered S Wave

In the **Model Builder** window, collapse the **Results** > **Scattered S Wave** node.

Total Displacement Field Magnitude

1 In the **Model Builder** window, right-click **Scattered Displacement Field Magnitude** and choose **Duplicate**.

Add the incident field to the scattered to obtain the total displacement.

2 In the **Settings** window for **2D Plot Group**, type `Total Displacement Field Magnitude` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Total Displacement Magnitude`.

Surface 1

1 In the **Model Builder** window, expand the **Total Displacement Field Magnitude** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `sqrt((real(u+uP))^2+(real(v+vP))^2)`.

4 In the **Total Displacement Field Magnitude** toolbar, click **Plot**.

RESULTS

Total Displacement Field Magnitude

In the **Model Builder** window, collapse the **Results** > **Total Displacement Field Magnitude** node.

Total P Wave

1 In the **Model Builder** window, right-click **Scattered P Wave** and choose **Duplicate**.

2 In the **Model Builder** window, click **Scattered P Wave 1**.

3 In the **Settings** window for **2D Plot Group**, type `Total P Wave` in the **Label** text field.

Surface 1

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

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

3 In the **Expression** text field, type `d(u+uP,x)+d(v+vP,y)`.

RESULTS

Total P Wave

In the **Model Builder** window, collapse the **Results** > **Total P Wave** node.

Total S Wave

1 In the **Model Builder** window, right-click **Scattered S Wave** and choose **Duplicate**.

2 In the **Model Builder** window, click **Scattered S Wave 1**.

3 In the **Settings** window for **2D Plot Group**, type `Total S Wave` in the **Label** text field.

Surface 1

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

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

3 In the **Expression** text field, type `d(v+vP,x)-d(u+uP,y)`.

4 In the **Total S Wave** toolbar, click  **Plot**.

Now analyze the case of impinging S wave.

DEFINITIONS

Incident S Wave

1 In the **Model Builder** window, right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Variables**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
uS	0[m]	m	Incident wave: u field
vS	exp(-1i*kS*x-1i*pi/2+1i*phase)[m]	m	Incident wave: v field
eps11S	d(uS,x)		Incident wave: strain tensor, 11 component
eps22S	d(vS,y)		Incident wave: strain tensor, 22 component
eps12S	0.5*(d(uS,y)+d(vS,x))		Incident wave: strain tensor, 12 component
s11S	(lambLame+2*muLame)*eps11S+lambLame*eps22S	Pa	Incident wave: stress tensor, 11 component
s22S	lambLame*eps11S+(lambLame+2*muLame)*eps22S	Pa	Incident wave: stress tensor, 22 component
s12S	2*muLame*eps12S	Pa	Incident wave: stress tensor, 12 component

SOLID MECHANICS (SOLID)

Body Load (Elastic Inclusion), P Wave, Cavity Inclusion, P Wave, Elastic Inclusion, P Wave, Infinitely Rigid Inclusion, P Wave

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)**, Ctrl-click to select **Cavity Inclusion, P Wave**, **Infinitely Rigid Inclusion, P Wave**, **Elastic Inclusion, P Wave**, and **Body Load (Elastic Inclusion), P Wave**.

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

P Wave

In the **Settings** window for **Group**, type `P Wave` in the **Label** text field.

P Wave 1

Right-click **P Wave** and choose **Duplicate**.

P Wave

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **P Wave** node.

S Wave

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **P Wave 1**.

2 In the **Settings** window for **Group**, type `S Wave` in the **Label** text field.

Cavity Inclusion, S Wave

1 In the **Model Builder** window, expand the **S Wave** node, then click **Cavity Inclusion, P Wave 1**.

2 In the **Settings** window for **Boundary Load**, type `Cavity Inclusion, S Wave` in the **Label** text field.

3 Locate the **Force** section. Specify the $\mathbf{f}_A$ vector as

`-(s11S*solid.nx+s12S*solid.ny)`	x
`-(s12S*solid.nx+s22S*solid.ny)`	y

Infinitely Rigid Inclusion, S Wave

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave** click **Infinitely Rigid Inclusion, P Wave 1**.

2 In the **Settings** window for **Prescribed Displacement**, type `Infinitely Rigid Inclusion, S Wave` in the **Label** text field.

3 Locate the **Prescribed Displacement** section. In the u_{0x} text field, type `-uS`.

4 In the u_{0y} text field, type `-vS`.

Elastic Inclusion, S Wave

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave** > **Elastic Inclusion, P Wave 1** node, then click **Elastic Inclusion, P Wave 1**.

2 In the **Settings** window for **Linear Elastic Material**, type `Elastic Inclusion, S Wave` in the **Label** text field.

Initial Stress and Strain

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave** > **Elastic Inclusion, S Wave** click **Initial Stress and Strain 1**.

2 In the **Settings** window for **Initial Stress and Strain**, type `Initial Stress and Strain` in the **Label** text field.

3 Locate the **Initial Stress and Strain** section. Specify the S_0 matrix as

`-s11S`	`-s12S`	0
-s12S	`-s22S`	0
0	0	0

4 Specify the ε_0 matrix as

-eps11S	-eps12S	0
-eps12S	-eps22S	0
0	0	0

Body Load (Elastic Inclusion), S Wave

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave** click **Body Load (Elastic Inclusion), P Wave 1**.

2 In the **Settings** window for **Body Load**, type `Body Load (Elastic Inclusion), S Wave` in the **Label** text field.

3 Locate the **Force** section. Specify the $\mathbf{f}_V$ vector as

(rho-rho_o)*(-omega^2)*uS	x
(rho-rho_o)*(-omega^2)*vS	y

Elastic Inclusion, S Wave

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave** > **Elastic Inclusion, S Wave** node.

SOLID MECHANICS (SOLID)

S Wave

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave** node.

Modify the previous study to include only the desired boundary conditions for future reruns.

P WAVE

Step 1: Frequency Domain

1 In the **Model Builder** window, under **P Wave** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave**.

5 Click **Disable**.

P WAVE

In the **Model Builder** window, collapse the **P Wave** node.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

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

S WAVE

1 In the **Settings** window for **Study**, type `S Wave` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

1 In the **Model Builder** window, under **S Wave** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `omega/2/pi[rad]`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **P Wave**.

6 Click **Disable**.

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

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

RESULTS

Add the results for the incident S wave near those obtained for the P wave.

Scattered u Field

1 In the **Model Builder** window, under **Results** click **Scattered u Field**.

2 In the **Settings** window for **2D Plot Group**, click to expand the **Plot Array** section.

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, expand the **Scattered u Field** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

4 In the **Scattered u Field** toolbar, click **Plot**.

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

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Scattered u Field

In the **Model Builder** window, click **Scattered u Field**.

Table Annotation 2

1 In the **Scattered u Field** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Scattered u Field** toolbar, click **Plot**.

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

Incident P Wave

1 In the **Model Builder** window, click **Incident P Wave**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** checkbox.

4 In the **Minimum** text field, type `-1.5`.

5 In the **Maximum** text field, type `1.5`.

Incident S Wave

1 In the **Model Builder** window, click **Incident S Wave**.

2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Incident P Wave**.

4 In the **Scattered u Field** toolbar, click **Plot**.

Scattered u Field

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

2 In the **Model Builder** window, click **Scattered u Field**.

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

4 From the **View** list, choose **New view**.

5 In the **Scattered u Field** toolbar, click **Plot**.

Scattered v Field

1 In the **Model Builder** window, click **Scattered v Field**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, expand the **Scattered v Field** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

Incident P Wave

1 In the **Model Builder** window, click **Incident P Wave**.

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

3 Select the **Manual color range** checkbox.

4 In the **Minimum** text field, type `-1.5`.

5 In the **Maximum** text field, type `1.5`.

Incident S Wave

1 In the **Model Builder** window, click **Incident S Wave**.

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

3 From the **Plot** list, choose **Incident P Wave**.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Scattered v Field

In the **Model Builder** window, click **Scattered v Field**.

Table Annotation 2

1 In the **Scattered v Field** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Scattered v Field** toolbar, click **Plot**.

Scattered v Field

1 In the **Model Builder** window, click **Scattered v Field**.

2 Click **Plot**.

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

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

5 From the **View** list, choose **View 2D 2**.

Scattered Displacement Field Magnitude

1 In the **Model Builder** window, click **Scattered Displacement Field Magnitude**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, expand the **Scattered Displacement Field Magnitude** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

Incident P Wave

1 In the **Model Builder** window, click **Incident P Wave**.

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

3 Select the **Manual color range** checkbox.

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

5 In the **Maximum** text field, type `2.5`.

Incident S Wave

1 In the **Model Builder** window, click **Incident S Wave**.

2 In the **Settings** window for **Surface**, click to collapse the **Inherit Style** section.

3 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Incident P Wave**.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Scattered Displacement Field Magnitude

In the **Model Builder** window, click **Scattered Displacement Field Magnitude**.

Table Annotation 2

1 In the **Scattered Displacement Field Magnitude** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Scattered Displacement Field Magnitude** toolbar, click **Plot**.

Scattered Displacement Field Magnitude

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

2 In the **Model Builder** window, click **Scattered Displacement Field Magnitude**.

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

4 From the **View** list, choose **View 2D 2**.

The resulting plot is shown in Figure 1.

Scattered P Wave

1 In the **Model Builder** window, expand the **Results** > **Scattered P Wave** node, then click **Scattered P Wave**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, under **Results** > **Scattered P Wave** click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Scattered P Wave

In the **Model Builder** window, click **Scattered P Wave**.

Table Annotation 2

1 In the **Scattered P Wave** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Scattered P Wave** toolbar, click **Plot**.

Scattered P Wave

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

2 In the **Model Builder** window, click **Scattered P Wave**.

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

4 From the **View** list, choose **View 2D 2**.

The resulting plot is shown in Figure 2.

Scattered S Wave

1 In the **Model Builder** window, click **Scattered S Wave**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, expand the **Scattered S Wave** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

4 In the **Scattered S Wave** toolbar, click **Plot**.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Scattered S Wave

In the **Model Builder** window, click **Scattered S Wave**.

Table Annotation 2

1 In the **Scattered S Wave** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Scattered S Wave** toolbar, click **Plot**.

Scattered S Wave

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

2 In the **Model Builder** window, click **Scattered S Wave**.

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

4 From the **View** list, choose **View 2D 2**.

The resulting plot is shown in Figure 3.

Total Displacement Field Magnitude

1 In the **Model Builder** window, expand the **Results** > **Total Displacement Field Magnitude** node, then click **Total Displacement Field Magnitude**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, under **Results** > **Total Displacement Field Magnitude** click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `sqrt((real(u+uS))^2+(real(v+vS))^2)`.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Total Displacement Field Magnitude

In the **Model Builder** window, click **Total Displacement Field Magnitude**.

Table Annotation 2

1 In the **Total Displacement Field Magnitude** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Total Displacement Field Magnitude** toolbar, click **Plot**.

Total Displacement Field Magnitude

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

2 In the **Model Builder** window, click **Total Displacement Field Magnitude**.

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

4 From the **View** list, choose **View 2D 2**.

The resulting plot is shown in Figure 4.

Total P Wave

1 In the **Model Builder** window, expand the **Results** > **Total P Wave** node, then click **Total P Wave**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, under **Results** > **Total P Wave** click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `d(u+uS,x)+d(v+vS,y)`.

5 In the **Total P Wave** toolbar, click **Plot**.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Total P Wave

In the **Model Builder** window, click **Total P Wave**.

Table Annotation 2

1 In the **Total P Wave** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Total P Wave** toolbar, click **Plot**.

Total P Wave

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

2 In the **Model Builder** window, click **Total P Wave**.

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

4 From the **View** list, choose **View 2D 2**.

The resulting plot is shown in Figure 5.

Total S Wave

1 In the **Model Builder** window, click **Total S Wave**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

Incident P Wave

1 In the **Model Builder** window, under **Results** > **Total S Wave** click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Incident P Wave` in the **Label** text field.

Incident S Wave

1 Right-click **Incident P Wave** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Incident S Wave` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **S Wave/Solution 2 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `d(v+vS,x)-d(u+uS,y)`.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Total S Wave

In the **Model Builder** window, click **Total S Wave**.

Table Annotation 2

1 In the **Total S Wave** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	P Wave
-9	10.5	S Wave

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Total S Wave** toolbar, click **Plot**.

Total S Wave

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

2 In the **Model Builder** window, click **Total S Wave**.

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

4 From the **View** list, choose **View 2D 2**.

The resulting plot is shown in Figure 6.

5 In the **Model Builder** window, collapse the **Total S Wave** node.

Total P Wave

In the **Model Builder** window, collapse the **Results** > **Total P Wave** node.

Total Displacement Field Magnitude

In the **Model Builder** window, collapse the **Results** > **Total Displacement Field Magnitude** node.

Scattered S Wave

In the **Model Builder** window, collapse the **Results** > **Scattered S Wave** node.

Scattered P Wave

In the **Model Builder** window, collapse the **Results** > **Scattered P Wave** node.

Scattered Displacement Field Magnitude

In the **Model Builder** window, collapse the **Results** > **Scattered Displacement Field Magnitude** node.

Scattered v Field

In the **Model Builder** window, collapse the **Results** > **Scattered v Field** node.

Scattered u Field

In the **Model Builder** window, collapse the **Results** > **Scattered u Field** node.

Scattered Displacement Field Magnitude, Scattered P Wave, Scattered S Wave, Scattered u Field, Scattered v Field, Total Displacement Field Magnitude, Total P Wave, Total S Wave

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Scattered u Field**, **Scattered v Field**, **Scattered Displacement Field Magnitude**, **Scattered P Wave**, **Scattered S Wave**, **Total Displacement Field Magnitude**, **Total P Wave**, and **Total S Wave**.

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

Plane Wave

In the **Settings** window for **Group**, type `Plane Wave` in the **Label** text field.

Add a point where to apply the point source.

GEOMETRY 1

Point 1 (pt1)

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

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

3 In the **x** text field, type `-((R-r_layer-r_o)/2+r_o)*cos(pi/4)`.

4 In the **y** text field, type `((R-r_layer-r_o)/2+r_o)*sin(pi/4)`.

5 Click **Build Selected**.

6 Drag and drop below **Circle 1 (c1)**.

7 Click **Build Selected**.

Array 1 (arr1)

1 In the **Model Builder** window, click **Array 1 (arr1)**.

2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Array**, click **Build All Objects**.

SOLID MECHANICS (SOLID)

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 29 only.

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

4 Specify the $\mathbf{F}_P$ vector as

1	x

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

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

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

POINT SOURCE INCIDENT FIELD

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

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

3 In the **Label** text field, type `Point Source Incident Field`.

P WAVE

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **P Wave** node, then click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Load 1**.

4 Click **Disable**.

P WAVE

In the **Model Builder** window, collapse the **P Wave** node.

S WAVE

1 In the **Model Builder** window, under **S Wave** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Click **Disable**.

S WAVE

In the **Model Builder** window, collapse the **S Wave** node.

POINT SOURCE INCIDENT FIELD

1 In the **Model Builder** window, under **Point Source Incident Field** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `omega/2/pi[rad]`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **P Wave**.

6 Click **Disable**.

7 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave**.

8 Click **Disable**.

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

Plot the field generated by the point source, that will be used as incident field.

RESULTS

Point Source Background Fields

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

2 In the **Settings** window for **2D Plot Group**, type `Point Source Background Fields` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Point Source Incident Field/ Solution 3 (sol3)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

Displacement Magnitude

1 Right-click **Point Source Background Fields** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Displacement Magnitude` in the **Label** text field.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **SpectrumLight**.

Selection 1

1 Right-click **Displacement Magnitude** and choose **Selection**.

2 Select Domains 19, 20, 22, 23, and 25 only.

3 In the **Point Source Background Fields** toolbar, click **Plot**.

Displacement Magnitude

1 In the **Model Builder** window, click **Displacement Magnitude**.

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

3 Select the **Manual color range** checkbox.

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

5 In the **Maximum** text field, type `0.15`.

6 In the **Point Source Background Fields** toolbar, click **Plot**.

P Wave

1 Right-click **Displacement Magnitude** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `P Wave` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `d(u,x)+d(v,y)`.

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

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

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

7 In the **Minimum** text field, type `-0.8`.

8 In the **Maximum** text field, type `0.8`.

9 In the **Point Source Background Fields** toolbar, click **Plot**.

P Wave 1

Right-click **P Wave** and choose **Duplicate**.

S Wave

1 In the **Model Builder** window, expand the **Results** > **Point Source Background Fields** > **P Wave** node, then click **Results** > **Point Source Background Fields** > **P Wave 1**.

2 In the **Settings** window for **Surface**, type `S Wave` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `d(v,x)-d(u,y)`.

4 In the **Point Source Background Fields** toolbar, click **Plot**.

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

6 Locate the **Range** section. In the **Minimum** text field, type `-4`.

7 In the **Maximum** text field, type `4`.

8 In the **Point Source Background Fields** toolbar, click **Plot**.

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

Point Source Background Fields

In the **Model Builder** window, click **Point Source Background Fields**.

Table Annotation 1

1 In the **Point Source Background Fields** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
24	5	Displacement
34.4	5	P Wave
44.6	5	S Wave

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 From the **Anchor point** list, choose **Center**.

7 In the **Point Source Background Fields** toolbar, click **Plot**.

The resulting plot is shown in Figure 7.

Point Source Background Fields

1 In the **Model Builder** window, click **Point Source Background Fields**.

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Point Source Background Fields`.

5 Clear the **Parameter indicator** text field.

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

View 2D 3

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 2D**.

Point Source Background Fields

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

2 From the **View** list, choose **View 2D 3**.

P Wave

In the **Model Builder** window, collapse the **Results** > **Point Source Background Fields** > **P Wave** node.

Displacement Magnitude

In the **Model Builder** window, collapse the **Results** > **Point Source Background Fields** > **Displacement Magnitude** node.

Point Source Background Fields

In the **Model Builder** window, collapse the **Results** > **Point Source Background Fields** node.

Set the results of the previous study to be the new incident field.

DEFINITIONS

Point Source

1 In the **Model Builder** window, right-click **Incident S Wave** and choose **Duplicate**.

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

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
uPS	`withsol('sol3',u)* exp(1i*phase)`	m	`Incident wave: u field`
vPS	`withsol('sol3',v)* exp(1i*phase)`	m	`Incident wave: v field`
eps11PS	`withsol('sol3', solid.el11)*exp(1i* phase)`		`Incident wave: strain tensor, 11 component`
eps22PS	`withsol('sol3', solid.el22)*exp(1i* phase)`		`Incident wave: strain tensor, 22 component`
eps12PS	`withsol('sol3', solid.el12)*exp(1i* phase)`		`Incident wave: strain tensor, 12 component`
s11PS	`withsol('sol3', solid.sl11)*exp(1i* phase)`	N/m^2	`Incident wave: stress tensor, 11 component`
s22PS	`withsol('sol3', solid.sl22)*exp(1i* phase)`	N/m^2	`Incident wave: stress tensor, 22 component`
s12PS	`withsol('sol3', solid.sl12)*exp(1i* phase)`	N/m^2	`Incident wave: stress tensor, 12 component`

Note that the point source background field is only defined on the third computational domain. Add two **General Extrusion** operators in order to make the point source background field available also for the other two computational domains.

General Extrusion 1 (genext1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.

2 Select Domains 17–25 only.

3 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

4 In the **x-expression** text field, type `x+24`.

General Extrusion 2 (genext2)

1 Right-click **General Extrusion 1 (genext1)** and choose **Duplicate**.

2 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

3 In the **x-expression** text field, type `x+12`.

SOLID MECHANICS (SOLID)

Point Source

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **P Wave** > **Cavity Inclusion, P Wave** node.

2 Right-click **S Wave** and choose **Duplicate**.

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

Cavity Inclusion, Point Source

1 In the **Model Builder** window, expand the **Point Source** node, then click **Cavity Inclusion, S Wave 1**.

2 In the **Settings** window for **Boundary Load**, type `Cavity Inclusion, Point Source` in the **Label** text field.

3 Locate the **Force** section. Specify the $\mathbf{f}_A$ vector as

`-(genext1(s11PS)*solid.nx+genext1(s12PS)*solid.ny)`	x
`-(genext1(s12PS)*solid.nx+genext1(s22PS)*solid.ny)`	y

Infinitely Rigid Inclusion, Point Source

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** click **Infinitely Rigid Inclusion, S Wave 1**.

2 In the **Settings** window for **Prescribed Displacement**, type `Infinitely Rigid Inclusion, Point Source` in the **Label** text field.

3 Locate the **Prescribed Displacement** section. In the u_{0x} text field, type `-genext2(uPS)`.

4 In the u_{0y} text field, type `-genext2(vPS)`.

Elastic Inclusion, Point Source

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** > **Elastic Inclusion, S Wave 1** node, then click **Elastic Inclusion, S Wave 1**.

2 In the **Settings** window for **Linear Elastic Material**, type `Elastic Inclusion, Point Source` in the **Label** text field.

Initial Stress and Strain

1 In the **Model Builder** window, click **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 Specify the S_0 matrix as

-s11PS	-s12PS	0
-s12PS	-s22PS	0
0	0	0

4 Specify the ε_0 matrix as

-eps11PS	-eps12PS	0
-eps12PS	-eps22PS	0
0	0	0

Body load (Elastic Inclusion), Point Source

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** click **Body Load (Elastic Inclusion), S Wave 1**.

2 In the **Settings** window for **Body Load**, type `Body load (Elastic Inclusion), Point Source` in the **Label** text field.

3 Locate the **Force** section. Specify the $\mathbf{f}_V$ vector as

(rho-rho_o)*(-omega^2)*uPS	x
(rho-rho_o)*(-omega^2)*vPS	y

Elastic Inclusion, Point Source

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** > **Elastic Inclusion, Point Source** node.

SOLID MECHANICS (SOLID)

Point Source

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** node.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

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

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

POINT SOURCE SCATTERED FIELD

1 In the **Settings** window for **Study**, type `Point Source Scattered Field` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

P WAVE

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **P Wave** node, then click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source**.

4 Click **Disable**.

S WAVE

1 In the **Model Builder** window, expand the **S Wave** node, then click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source**.

4 Click **Disable**.

POINT SOURCE INCIDENT FIELD

1 In the **Model Builder** window, under **Point Source Incident Field** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source**.

4 Click **Disable**.

POINT SOURCE SCATTERED FIELD

1 In the **Model Builder** window, under **Point Source Scattered Field** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `omega/2/pi[rad]`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **P Wave**.

6 Click **Disable**.

7 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave**.

8 Click **Disable**.

9 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Load 1**.

10 Click **Disable**.

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

RESULTS

Point Source Scattered Displacement Field Magnitude

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

2 In the **Settings** window for **2D Plot Group**, type `Point Source Scattered Displacement Field Magnitude` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Displacement Magnitude`.

5 Clear the **Parameter indicator** text field.

6 Locate the **Data** section. From the **Dataset** list, choose **Point Source Scattered Field/ Solution 4 (sol4)**.

Surface 1

Right-click **Point Source Scattered Displacement Field Magnitude** and choose **Surface**.

Selection 1

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

2 Select Domains 3, 4, 6, 7, 11, 12, 14, 15, 19, 20, 22, 23, and 25 only.

Surface 1

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

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

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

Point Source Scattered Displacement Field Magnitude

1 In the **Model Builder** window, click **Point Source Scattered Displacement Field Magnitude**.

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

3 Clear the **Plot dataset edges** checkbox.

Table Annotation 1

1 In the **Point Source Scattered Displacement Field Magnitude** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	5	Cavity
12	5	Rigid
24	5	Elastic

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 From the **Anchor point** list, choose **Center**.

The resulting plot is shown in Figure 8.

View 2D 4

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 2D**.

Point Source Scattered Displacement Field Magnitude

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

2 From the **View** list, choose **View 2D 4**.

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

Surface 1

In the **Model Builder** window, collapse the **Results** > **Point Source Scattered Displacement Field Magnitude** > **Surface 1** node.

Point Source Scattered Displacement Field Magnitude 1

Right-click **Results** > **Point Source Scattered Displacement Field Magnitude** and choose **Duplicate**.

Point Source Scattered Displacement Field Magnitude

In the **Model Builder** window, collapse the **Results** > **Point Source Scattered Displacement Field Magnitude** node.

Point Source Total Displacement Field Magnitude

1 In the **Model Builder** window, under **Results** click **Point Source Scattered Displacement Field Magnitude 1**.

2 In the **Settings** window for **2D Plot Group**, type `Point Source Total Displacement Field Magnitude` in the **Label** text field.

3 In the **Model Builder** window, expand the **Point Source Total Displacement Field Magnitude** node.

Scattered Field Formulation

1 In the **Model Builder** window, expand the **Results** > **Point Source Total Displacement Field Magnitude** > **Surface 1** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, type `Scattered Field Formulation` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `if(x>20, sqrt((real(u+uPS))^2+(real(v+vPS))^2),if(x<5,sqrt((real(u+genext1(uPS)))^2+(real(v+genext1(vPS)))^2),sqrt((real(u+genext2(uPS)))^2+(real(v+genext2(vPS)))^2)))`.

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

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

6 In the **Maximum** text field, type `0.15`.

7 In the **Point Source Total Displacement Field Magnitude** toolbar, click  **Plot**.

RESULTS

Point Source Total Displacement Field Magnitude

1 In the **Model Builder** window, collapse the **Results** > **Point Source Total Displacement Field Magnitude** node.

2 In the **Model Builder** window, click **Point Source Total Displacement Field Magnitude**.

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

4 In the **Title** text area, type `Total Displacement Field Magnitude`.

Compute the total field without using the scattered field formulation, that is, adopting the actual boundary conditions and field equation for the total field.

SOLID MECHANICS (SOLID)

Point Load 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Point Load 1**.

2 Select Points 3, 16, and 29 only.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 41, 42, 45, and 46 only.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

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

POINT SOURCE TOTAL FIELD (NO SCATTERED FIELD FORMULATION)

1 In the **Settings** window for **Study**, type `Point Source Total Field (NO Scattered Field Formulation)` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

P WAVE

Step 1: Frequency Domain

1 In the **Model Builder** window, under **P Wave** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint 1**.

4 Click **Disable**.

S WAVE

1 In the **Model Builder** window, under **S Wave** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint 1**.

4 Click **Disable**.

POINT SOURCE INCIDENT FIELD

1 In the **Model Builder** window, under **Point Source Incident Field** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint 1**.

4 Click **Disable**.

POINT SOURCE SCATTERED FIELD

1 In the **Model Builder** window, under **Point Source Scattered Field** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Click **Disable**.

POINT SOURCE TOTAL FIELD (NO SCATTERED FIELD FORMULATION)

1 In the **Model Builder** window, under **Point Source Total Field (NO Scattered Field Formulation)** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `omega/2/pi[rad]`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **P Wave**.

6 Click **Disable**.

7 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **S Wave**.

8 Click **Disable**.

9 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** > **Cavity Inclusion, Point Source**.

10 Click **Disable**.

11 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** > **Infinitely Rigid Inclusion, Point Source**.

12 Click **Disable**.

13 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** > **Elastic Inclusion, Point Source** > **Initial Stress and Strain**.

14 Click **Disable**.

15 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Point Source** > **Body load (Elastic Inclusion), Point Source**.

16 Click **Disable**.

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

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

19 In the **Model Builder** window, collapse the **Point Source Total Field (NO Scattered Field Formulation)** node.

Plot the results obtained next to those obtained with the scattered field formulation for comparison.

RESULTS

Point Source Total Displacement Field Magnitude

1 In the **Model Builder** window, under **Results** click **Point Source Total Displacement Field Magnitude**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

NO Scattered Field Formulation

1 In the **Model Builder** window, expand the **Point Source Total Displacement Field Magnitude** node.

2 Right-click **Scattered Field Formulation** and choose **Duplicate**.

3 In the **Settings** window for **Surface**, locate the **Data** section.

4 From the **Dataset** list, choose **Point Source Total Field (NO Scattered Field Formulation)/ Solution 5 (sol5)**.

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

6 In the **Point Source Total Displacement Field Magnitude** toolbar, click **Plot**.

7 Locate the **Inherit Style** section. From the **Plot** list, choose **Scattered Field Formulation**.

8 In the **Label** text field, type `NO Scattered Field Formulation`.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.

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

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0	15.5	Cavity
12	15.5	Rigid
24	15.5	Elastic

Point Source Total Displacement Field Magnitude

In the **Model Builder** window, click **Point Source Total Displacement Field Magnitude**.

Table Annotation 2

1 In the **Point Source Total Displacement Field Magnitude** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
-9	0	SFF
-9	10.5	NO SFF

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Middle left**.

6 Clear the **Show point** checkbox.

7 In the **Point Source Total Displacement Field Magnitude** toolbar, click **Plot**.

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

Point Source Total Displacement Field Magnitude

1 In the **Model Builder** window, click **Point Source Total Displacement Field Magnitude**.

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

3 From the **View** list, choose **New view**.

4 In the **Point Source Total Displacement Field Magnitude** toolbar, click **Plot**.

The resulting plot is shown in Figure 9.

Point Source Total P Wave

1 Right-click **Point Source Total Displacement Field Magnitude** and choose **Duplicate**.

2 In the **Model Builder** window, click **Point Source Total Displacement Field Magnitude 1**.

3 In the **Settings** window for **2D Plot Group**, type `Point Source Total P Wave` in the **Label** text field.

Scattered Field Formulation

1 In the **Model Builder** window, click **Scattered Field Formulation**.

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

3 In the **Expression** text field, type `if(x>20,d(u+uPS,x)+d(v+vPS,y),if(x<5,d(u+genext1(uPS),x)+d(v+genext1(vPS),y),d(u+genext2(uPS),x)+d(v+genext2(vPS),y)))`.

4 In the **Point Source Total P Wave** toolbar, click **Plot**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

6 Locate the **Range** section. In the **Minimum** text field, type `-0.5`.

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

8 In the **Point Source Total P Wave** toolbar, click **Plot**.

NO Scattered Field Formulation

1 In the **Model Builder** window, click **NO Scattered Field Formulation**.

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

3 In the **Expression** text field, type `d(u,x)+d(v,y)`.

4 In the **Point Source Total P Wave** toolbar, click **Plot**.

Point Source Total S Wave

1 In the **Model Builder** window, right-click **Point Source Total P Wave** and choose **Duplicate**.

2 In the **Model Builder** window, click **Point Source Total P Wave 1**.

3 In the **Settings** window for **2D Plot Group**, type `Point Source Total S Wave` in the **Label** text field.

Scattered Field Formulation

1 In the **Model Builder** window, click **Scattered Field Formulation**.

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

3 In the **Expression** text field, type `if(x>20,-d(u+uPS,y)+d(v+vPS,x),if(x<5,-d(u+genext1(uPS),y)+d(v+genext1(vPS),x),-d(u+genext2(uPS),y)+d(v+genext2(vPS),x)))`.

4 Locate the **Range** section. In the **Minimum** text field, type `-1`.

5 In the **Maximum** text field, type `1`.

6 In the **Point Source Total S Wave** toolbar, click  **Plot**.

NO Scattered Field Formulation

1 In the **Model Builder** window, click **NO Scattered Field Formulation**.

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

3 In the **Expression** text field, type `-d(u,y)+d(v,x)`.

4 In the **Point Source Total S Wave** toolbar, click  **Plot**.