Model created in COMSOL Multiphysics 6.4

Wave Propagation in Rock Under Blast Loads

Introduction

This example presents a transient analysis of the wave propagation in a rock mass caused by a short duration load on the surface. Such loads are typical during tunnel constructions and other excavations using blasting. It shows the use of the Low-reflecting boundary conditions to truncate the computational domain to a reasonable size. The results are in very good agreement with a published study (see Ref. 1).

As a default, the low-reflecting boundary condition takes the material data from the adjacent domain in an attempt to create a perfect impedance match for both pressure waves and shear waves, so that

$$\sigma \cdot \mathbf{n} = -\rho c_p\left(\frac{\partial \mathbf{u}}{\partial t} \cdot \mathbf{n}\right)\mathbf{n} - \rho c_s\left(\frac{\partial \mathbf{u}}{\partial t} \cdot \mathbf{t}\right)\mathbf{t}$$

where **n** and **t** are the unit normal and tangential vectors at the boundary, respectively, and c_p and c_s are the speeds of the pressure and shear waves in the material. This approach works best when the wave direction in close to the normal at the wall.

More information about modeling using low-reflecting boundary conditions can be found in Ref. 2.

Model Definition

The model geometry is a block. Two of the side walls are symmetry planes. The other two represent the truncation of the computational domain in the directions where the rock has lateral dimensions that are significantly larger than the depth. The size of the block and the material parameters correspond to that studied in Ref. 1.

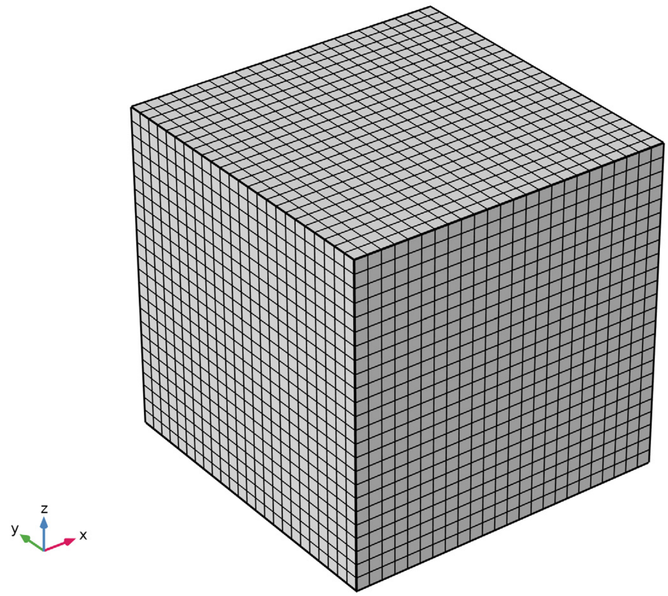

Figure 1: Geometry and mesh.

Thus, the following elastic material data is used: the Young's modulus $E = 50$ GPa, Poisson's ratio $\nu = 2/7$, and density $\rho = 2700\ \mathrm{kg/m^3}$. These values represent a granite rock.

The upper surface is free, and the bottom surface is subjected to loading in form of a finite duration pressure pulse localized near the origin, see Figure 2. The loading is similar to that used in Ref. 1 and represents an explosion within the rock near the surface.

The truncation boundaries are modeled with, and then without, applying the low-reflecting boundary conditions.

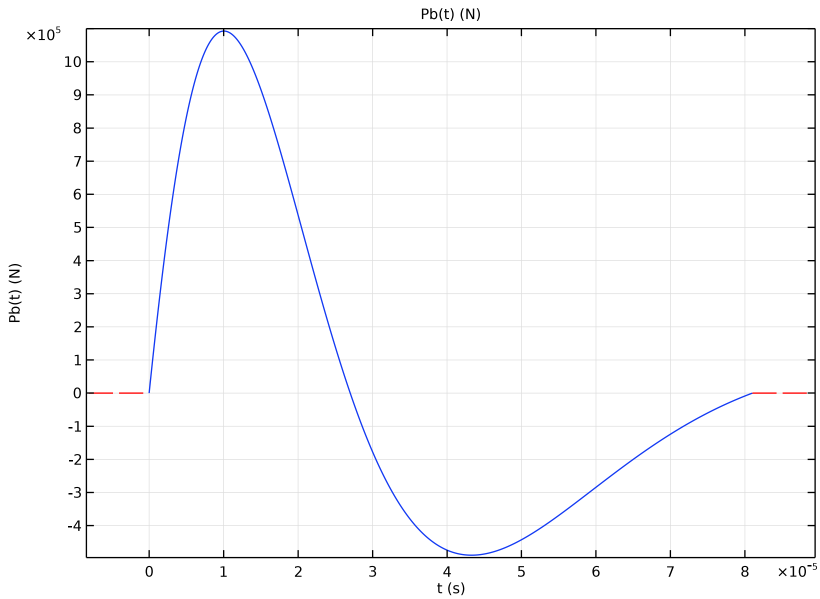

Figure 2: The load function.

Results and Discussion

The wave propagation in the block is modeled via a transient study covering a time interval of 150 µs. The typical wave propagation pattern is shown in Figure 3.

The vertical displacement at the upper surface is shown in Figure 4 for both cases, with and without using the low-reflecting boundary conditions. The analytical estimate for the time when the pressure wave reaches the surface is H/c_p, where H is the height of the block. In case of reflection, the time for the reflected pressure wave to arrive at the sampling point is $\sqrt{5}L/c_p$, where L is the distance of blast point from the truncated boundaries. Both estimates have very good agreement with the computations. Thus, the two responses start to deviate from each other after the estimated time as shown in Figure 4, which is caused by the reflected wave. The wave pattern at the upper surface is analyzed using a spatial fast Fourier transform (FFT). The results are shown in Figure 5 and Figure 6 for two time moments respectively before and after the wave reflection starts. Figure 6 clearly shows significant contributions from the waves reflected from the side boundaries in case when the computations have been performed without using the low-reflecting boundary conditions.

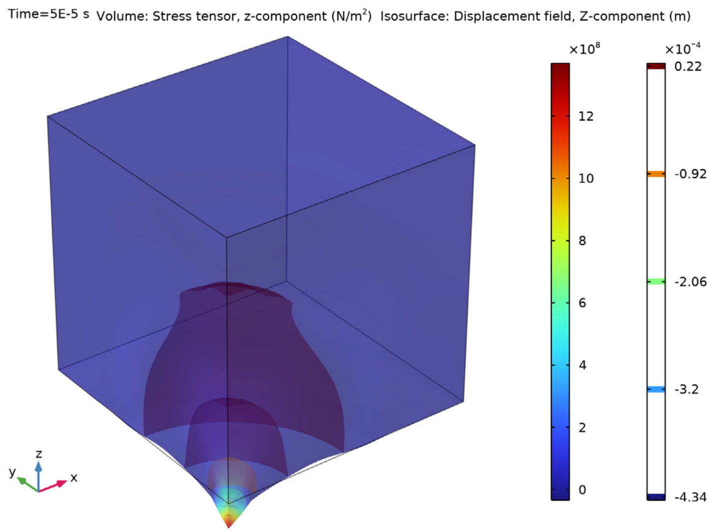

Figure 3: The stress in the block at the early stage of the elastic wave propagation.

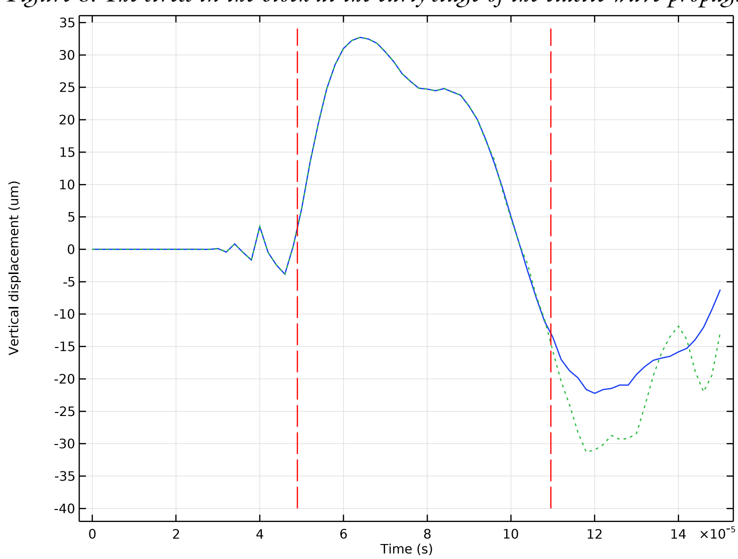

Figure 4: The displacement at the upper surface for the cases with (solid line) and without (dotted line) applying the low-reflecting boundary conditions. The dashed vertical lines represent analytical estimates for time of arrival of incoming and reflected waves, respectively.

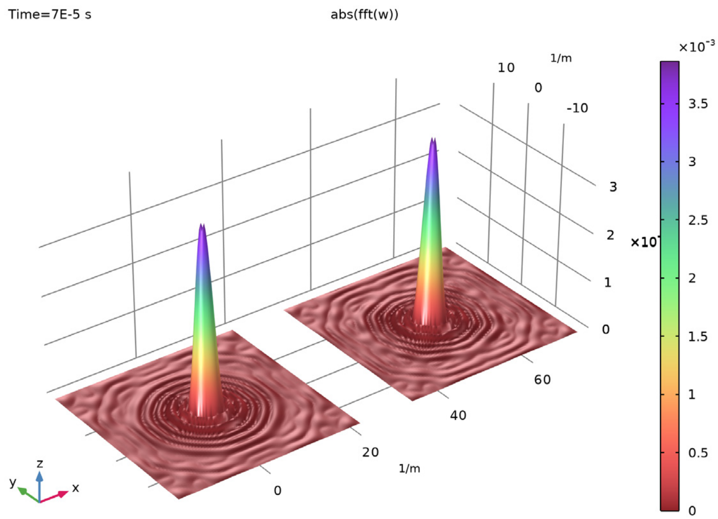

Figure 5: Spatial FFT of the wave pattern the top surface taken at t = 7e-5 s and computed with (left) and without (right) using the low-reflecting boundary conditions.

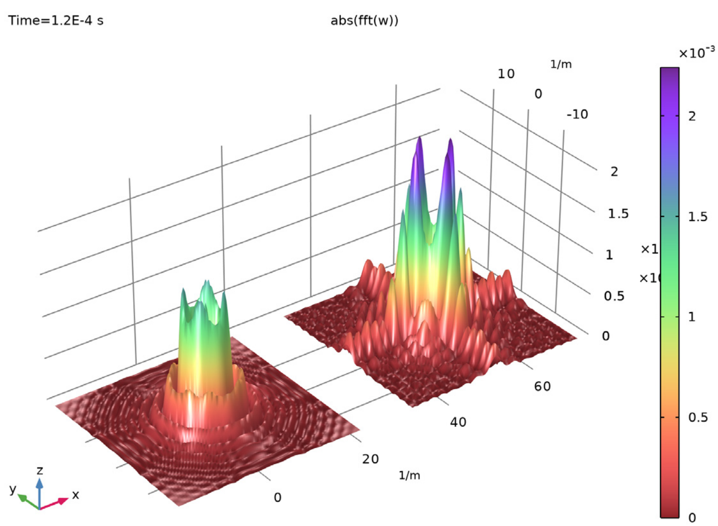

Figure 6: Spatial FFT of the wave pattern at the top surface taken at t = 1.4e-4 s and computed with (left) and without (right) using the low-reflecting boundary conditions.

References

1. H. Sönnerlind, “Beräkningsmetoder för spänningar i explosionsbelastad berg,” rapport nr 001202 (in Swedish), Epsilon HighTech Innovation AB, 2005.

2. M. Cohen and P.C. Jennings, “Silent Boundary Methods for Transient Analysis,” *Computational Methods for Transient Analysis,* vol. 1, T. Belytschko and T.J.R. Hughes, eds., North-Holland, 1983.

Application Library path: `Structural_Mechanics_Module/Elastic_Waves/blasting_rock`

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 **3D**.

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** > **Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Define the geometry and loading parameters.

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 In the table, enter the following settings:

Name	Expression	Value	Description
H	240[mm]	0.24 m	Depth
L	H	0.24 m	Width
L1	L/24	0.01 m	Load extension
Q	1[g]	0.001 kg	Amount of explosive
P0	140e6[N]*(Q/1[kg])^(2/3)	1.4E6 N	Load magnitude
t0	0.81e-3[s]*(Q/1[kg])^(1/3)	8.1E-5 s	Loading duration
gamma	1.86	1.86	Decay rate
u0	50[um]	5E-5 m	Displacement scale

DEFINITIONS

Next, define the loading function.

Piecewise 1 (pw1)

1 In the **Definitions** toolbar, click **Piecewise**.

2 In the **Settings** window for **Piecewise**, type Pb in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type t.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	t0	P0*exp(-gamma*t/t0)*sin(4*pi/(1+t0/t))

5 Locate the **Units** section. In the **Arguments** text field, type s.

6 In the **Function** text field, type N.

Plot the function which should look similar to that shown in Figure 2.

7 Click **Plot**.

GEOMETRY 1

Block 1 (blk1)

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

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

3 In the **Width** text field, type L.

4 In the **Depth** text field, type L.

5 In the **Height** text field, type H.

6 Click **Build Selected**.

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

Enter the material parameters, which correspond to granite rock.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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**, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated text field, type `50[GPa]`.

4 From the ν list, choose **User defined**. In the associated text field, type `2/7`.

5 From the ρ list, choose **User defined**. In the associated text field, type `2700`.

Because of the symmetry, you model one quarter of the geometry. This also explains the factor of 0.25 used in the load expression, since the load is given as a total force.

Symmetry 1

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

2 Select Boundaries 1 and 2 only.

Boundary Load 1

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

2 Select Boundary 3 only.

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

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

0	x
0	y
0.25*Pb(t)/L1^2*(X<=L1)*(Y<=L1)	z

Low-Reflecting Boundary 1

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

2 Select Boundaries 5 and 6 only.

Use linear elements to reduce the numerical dispersion of the wavefront.

3 In the **Model Builder** window, click **Solid Mechanics (solid)**.

4 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

5 From the **Displacement field** list, choose **Linear**.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Select Boundary 3 only.

The number of elements in the **Distribution** node should be written as `floor(L/L1)` in order to have the input in an integer format.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 2, 3, 7, and 10 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `floor(L/L1)`.

5 Click **Build Selected**.

Swept 1

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

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `floor(L/L1)`.

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

The resulting mesh should be similar to that shown in Figure 1.

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(0,2e-6,1.5e-4)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

   The default scale for the displacement is based on the size of the geometry. Change this to a scale more suitable for the wave propagation analysis.

3. In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node, then click **Displacement Field (comp1.u)**.
4. In the **Settings** window for **Field**, locate the **Scaling** section.
5. In the **Scale** text field, type `u0`.
6. In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.
7. In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
8. In the **Amplification for high frequency** text field, type `0.5`.

   This change in the damping will help to suppress numerical artifacts in the transient solution.

RESULTS

Before solving the problem, prepare a plot of the vertical displacement component. This will be shown and updated during the computations.

Displacement

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Displacement` in the **Label** text field.
3. Click to expand the **Title** section. From the **Title type** list, choose **None**.

Point Graph 1

1. Right-click **Displacement** and choose **Point Graph**.
2. Select Point 2 only.
3. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `w`.
5. In the **Unit** field, type `um`.

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

3 Select the **Plot** checkbox.

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

RESULTS

Add a volume plot of the stress similar to that shown in Figure 3.

3D Plot Group 2

In the **Results** toolbar, click **3D Plot Group**.

Volume 1

1 Right-click **3D Plot Group 2** and choose **Volume**.

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

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

Deformation 1

Right-click **Volume 1** and choose **Deformation**.

3D Plot Group 2

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

2 From the **Time (s)** list, choose **5E-5**.

3 Click the **Transparency** button in the **Graphics** toolbar.

4 In the **3D Plot Group 2** toolbar, click **Plot**.

Isosurface 1

1 Right-click **3D Plot Group 2** and choose **Isosurface**.

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

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

Deformation 1

1 Right-click **Isosurface 1** and choose **Deformation**.

2 In the **3D Plot Group 2** toolbar, click **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

ROOT

Next, add a new study, for which the low-reflecting boundary conditions will be disabled.

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** > **Time Dependent**.

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

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

STUDY 2

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,2e-6,1.5e-4)`.

3 Locate the **Results While Solving** section. Select the **Plot** checkbox.

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)** > **Low-Reflecting Boundary 1**.

6 Right-click and choose **Disable**.

7 In the **Model Builder** window, click **Study 2**.

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

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

Configure the solver in the same way as for the first study.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Dependent Variables 1** node, then click **Displacement Field (comp1.u)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `u0`.

6 In the **Model Builder** window, under **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** click **Time-Dependent Solver 1**.

7 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

8 In the **Amplification for high frequency** text field, type `0.5`.

RESULTS

Point Graph 2

1 In the **Model Builder** window, under **Results** > **Displacement** right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

STUDY 2

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

RESULTS

Displacement

Add the following line plots to show the estimated times at which the incoming and reflected pressure waves hit the block surface.

Point Graph 3

1 In the **Model Builder** window, right-click **Displacement** and choose **Point Graph**.

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

3 In the **Expression** text field, type `(t/t0-1)*40`.

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

5 In the **Expression** text field, type `L/solid.cp`.

6 Select Point 2 only.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

8 From the **Color** list, choose **Red**.

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `L*sqrt(5)/solid.cp`.

Displacement

1 In the **Model Builder** window, click **Displacement**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **x-axis label** checkbox. In the associated text field, type `Time (s)`.
4 Select the **y-axis label** checkbox. In the associated text field, type `Vertical displacement (um)`.
5 In the **Displacement** toolbar, click **Plot**.

   The final plot should look similar to that shown in Figure 4.

Investigate the wave pattern using the spatial FFT.

Mirror 3D 1

In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

Mirror 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.
2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 1**.
4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.
2 In the **Settings** window for **Cut Plane**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 2**.
4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
5 In the **z-coordinate** text field, type `H`.

Spatial FFT 1

1 In the **Results** toolbar, click **More Datasets** and choose **Spatial FFT**.
2 In the **Settings** window for **Spatial FFT**, locate the **Data** section.
3 From the **Dataset** list, choose **Cut Plane 1**.
4 Locate the **Transformation** section. Find the **Spatial resolution** subsection. From the **Resolution** list, choose **Manual**.
5 Find the **Sampling resolution** subsection. In the N_x text field, type `20`.
6 Find the **Spatial resolution** subsection. In the N_y text field, type `20`.
7 Find the **Spatial layout** subsection. From the **Layout** list, choose **Use zero padding**.

8 In the **x padding** text field, type 40.

9 In the **y padding** text field, type 40.

Cut Plane 1, Mirror 3D 1, Mirror 3D 2, Spatial FFT 1

1 In the **Model Builder** window, under **Results** > **Datasets**, Ctrl-click to select **Mirror 3D 1**, **Mirror 3D 2**, **Cut Plane 1**, and **Spatial FFT 1**.

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

Mirror 3D 3

1 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Mirror 3D 4

1 In the **Model Builder** window, click **Mirror 3D 4**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 3**.

Cut Plane 2

1 In the **Model Builder** window, click **Cut Plane 2**.

2 In the **Settings** window for **Cut Plane**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 4**.

Spatial FFT 2

1 In the **Model Builder** window, click **Spatial FFT 2**.

2 In the **Settings** window for **Spatial FFT**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 2**.

2D Plot Group 3

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

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

3 From the **Dataset** list, choose **Spatial FFT 1**.

4 From the **Time (s)** list, choose **7E-5**.

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

6 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

Surface 1

1 Right-click **2D Plot Group 3** and choose **Surface**.

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

3 In the **Expression** text field, type `abs(fft(w))`.

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

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

Height Expression 1

Right-click **Surface 1** and choose **Height Expression**.

Surface 2

1 In the **Model Builder** window, under **Results** > **2D Plot Group 3** right-click **Surface 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Spatial FFT 2**.

4 From the **Time (s)** list, choose **7E-5**.

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

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 In the **2D Plot Group 3** toolbar, click **Plot**.

2D Plot Group 4

1 In the **Model Builder** window, under **Results** right-click **2D Plot Group 3** and choose **Duplicate**.

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

3 From the **Time (s)** list, choose **1.2E-4**.

Surface 2

1 In the **Model Builder** window, expand the **2D Plot Group 4** node, then click **Surface 2**.

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

3 From the **Time (s)** list, choose **1.2E-4**.

4 In the **2D Plot Group 4** toolbar, click **Plot**.