Created in COMSOL Multiphysics 6.1

Generation of Lamb Waves for Nondestructive Inspection of Plate Specimens

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Lamb waves are named after Horace Lamb who studied the motion of two-dimensional waves in solids bounded by two parallel planes in his classic work Ref. 1. He obtained two sets of analytical solutions for symmetric and antisymmetric modes propagating in infinite plates with a finite thickness. Lamb waves belong to guided waves that propagate in long solid objects (waveguides), such as rods; plates; or pipes, but also in more complex structures, like rails, where it is impossible to find an analytical solution.

Numerical modeling of Lamb waves is essential for the analysis and design of the structural health monitoring (SHM) systems for long range ultrasonic testing. The design of an SHM system based on guided waves consists of two main parts. In the first place, it is required to know the dispersion curves for the modes that can propagate in the waveguide of a certain cross section. In the second place, the propagation of a chosen mode through the waveguide and its reflection from a possible irregularity in the cross-section area (a fracture or a corrosion defect) is analyzed in the time domain.

This tutorial studies the propagation of guided waves in a steel plate with a finite width and thickness. In the first part, the *Mode Analysis* study for the *Solid Mechanics* physics interface applied on the 2D cross section of the plate computes the propagating modes. This approach is also sometimes referred to as the semi-analytical finite element (SAFE) method. In the second part, an angle beam wedge excites the desired mode in the plate with a flaw, which is modeled in 3D using the *Elastic Wave, Time Explicit* physics interface.

Model Definition

The steel plate used here is 15 mm wide and 5 mm thick. An angle beam wedge transducer placed on top of the plate generates the antisymmetric or flexural zero-order mode (A_0) in the plate. The wedge is made of plastic and has the angle of incidence of 70°. A normal velocity applied on the excitation area of the wedge generates a longitudinal wave that travels through the wedge. The setup is shown in Figure 1. This tutorial thus omits the details on the transducer modeling which are, for example, given in Angle Beam Nondestructive Testing tutorial model.

The source signal is a five-cycle tone burst with the center frequency f_0 = 200 kHz as shown in Figure 2. The longitudinal wave velocity in plastic is c_p = 2080 m/s and the estimated velocity (see the next section) of the refracted A_0-mode in the plate at the chosen frequency is v_Lamb = 2300 m/s. The wave refraction is computed according to Snell’s law as

$$\frac{\sin\alpha}{c_{\mathrm{p}}} = \frac{\sin\beta}{v_{\mathrm{Lamb}}} \tag{1}$$

The refraction angle β = 90°, which yields the critical angle of about α = 65°. Since the angle of incident of the chosen wedge is larger than the critical angle, the wave sent through the wedge will be refracted into the desired mode in the plate.

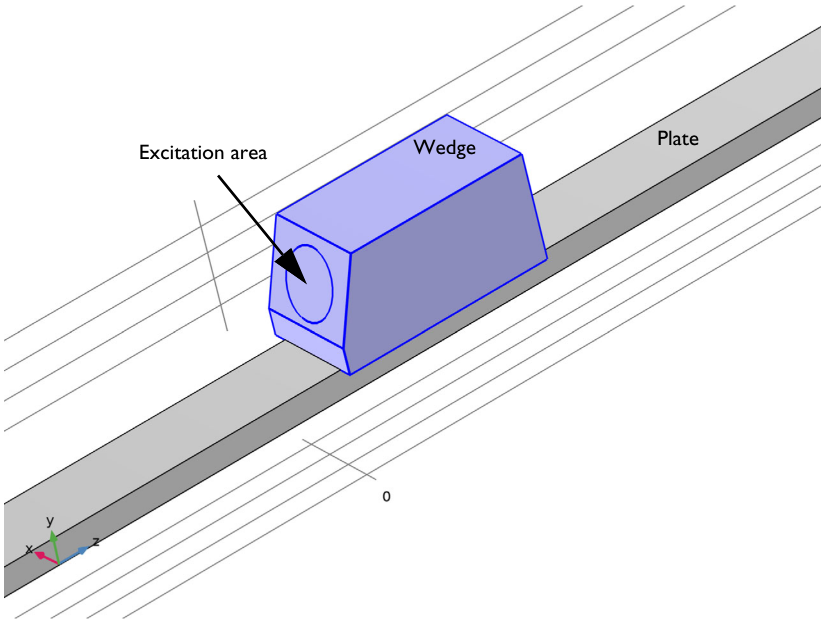

Figure 1: Model setup including wedge and test plate.

The plate has a flaw, which is a fracture with the dimensions of the plate thickness and half of its width. The orientation of the fracture makes the guided wave hit it at a right angle. Part of the wave will pass further along the plate, and part of it will be reflected back to the wedge. The transmitted and reflected signals are recorded at four observation points placed on the top surface of the plate at equal distance before and after the fracture.

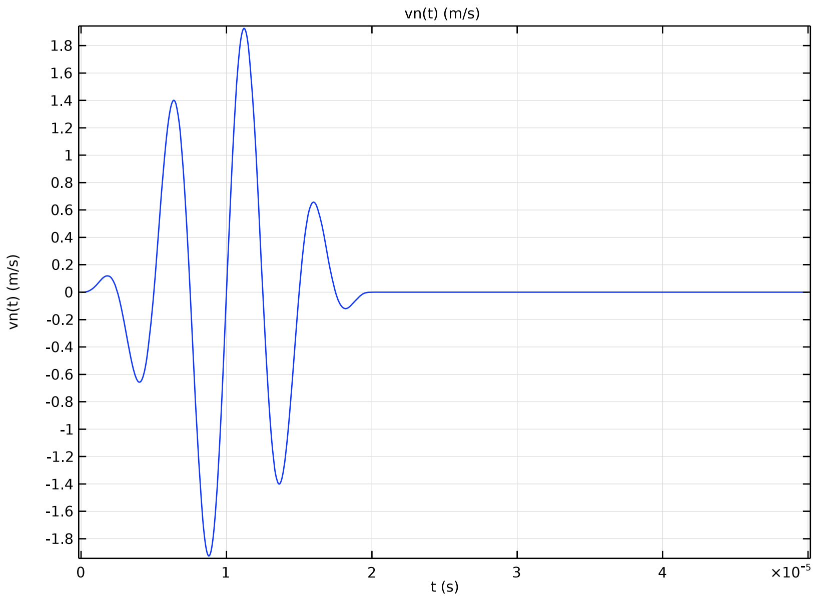

Figure 2: Five-cycle tone burst source signal.

Results and Discussion

First, look at the dispersion diagrams, Figure 3 and Figure 4, obtained from the mode analysis for the plate cross section. The plots show the out-of-plane wave numbers, k_n, and the phase velocities, v_p, of the propagating modes, respectively. The phase velocities are defined as

$$v_p = \frac{\omega}{k_n} = \frac{2\pi f}{k_n}$$

and the group velocities as

$$v_g = \frac{d\omega}{dk_n}$$

Note that the phase velocities in Figure 4 are normalized to the shear wave speed in steel. The red lines depict the wave numbers and normalized velocities of the pure longitudinal and shear waves in steel.

Figure 3 and Figure 4 indicate that each propagating mode exists above a certain nascent frequency (cut on/off frequency of the mode). Four of the computed modes have nascent frequencies of zero and thus exist throughout the whole frequency range. The number of such modes in this example is higher than that from the classic result of Lamb (two zero-order antisymmetric, A_0, and symmetric, S_0, modes), which is due to the finite (and comparable) dimensions of the plate cross section. The more complex the waveguide cross section, the higher number of modes will be able to propagate in it.

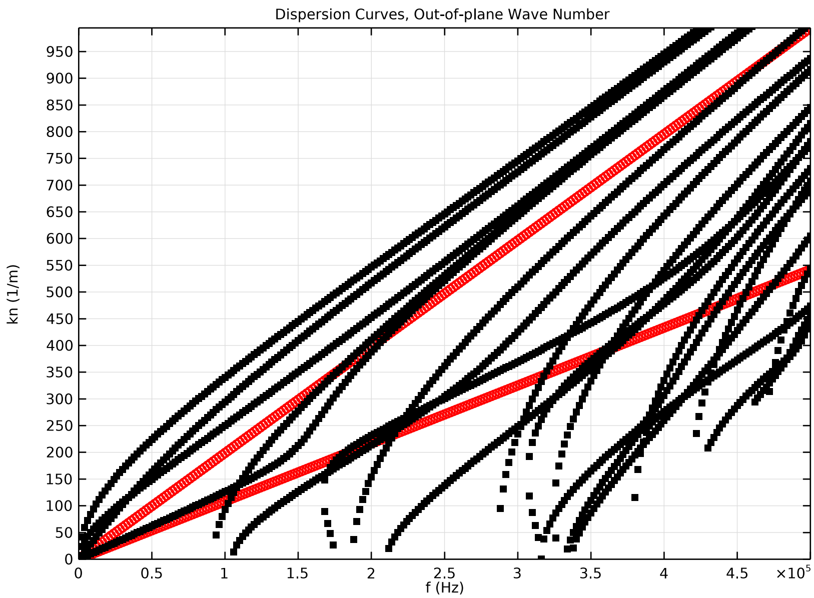

Figure 3: Dispersion curves of Lamb waves in the plate: out-of-plane wave number.

The lowermost curve in Figure 4 corresponds to the vertically polarized A_0-mode: its vertical displacements are symmetric while the out-of-plane ones are antisymmetric (see the orientation of the coordinate axes in Figure 1). It is the slowest propagating mode in the plate and is depicted in the upper-left corner in Figure 5. Figure 5 shows the profiles of the modes propagating at f_0 = 200 kHz that are symmetric with respect to the sagittal plane of the system, as those are the only ones that can be excited by the current design of the wedge.

Figure 4 exhibits the known property of Lamb waves: they are dispersive, that is, their speed depends on the frequency. In this regards, the choice of the excitation frequency is no accident For example, the estimated velocity of the A_0-mode at 100 kHz is about 1720 m/s. This value is less than the longitudinal wave speed in the wedge, thus making the desired wave refraction unreachable (see Equation 1) for the current dimension of the plate and the wedge material, and the angle of incidence.

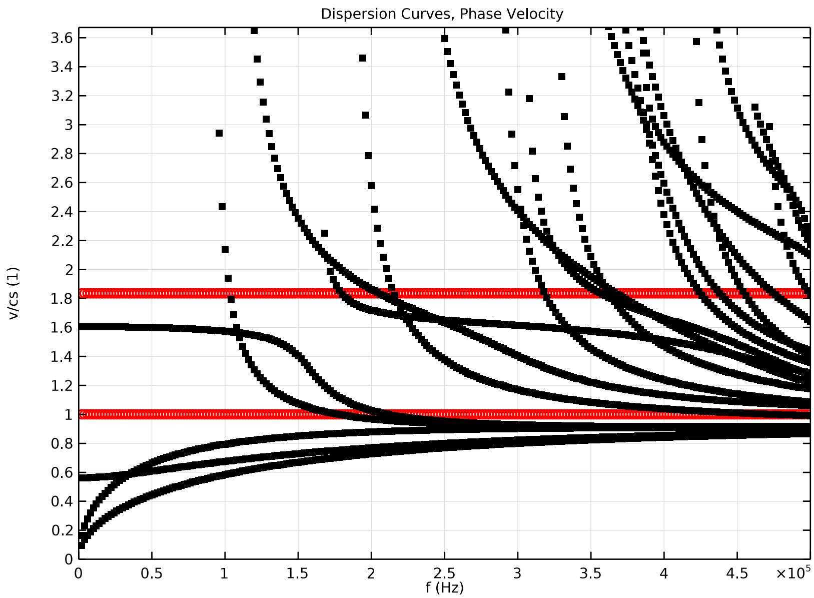

Figure 4: Dispersion curves of Lamb waves in the plate: normalized phase velocity.

The second part of this tutorial is dedicated to the analysis of the selected mode propagation in the 3D plate and its reflection from the flaw. The evolution of the traveling wave is illustrated in Figure 6. Note that Figure 6 shows the velocity magnitude profiles only in the plate, as those in the wedge are of no interest here.

The wave packet travels toward the defect (t = 80 μs) until it hits it (t = 110 μs). Then a portion of the packet passes further down the plate while another portion is reflected back to the wedge (t = 140 μs). The reflected part of the signal travels back toward the wedge (t = 170 μs). Note that the signal is no longer symmetric with respect to the sagittal plane after it has hit the flaw.

The vertical displacements of the signal computed at the listening points placed before and after the fracture are depicted in Figure 7 on the left and on the right, respectively. The blue lines correspond to the side of the plate with the fracture; the green lines, to the opposite side. As the wave is dispersive, the wave packet travels with the group velocity. Measuring the time delay between the incident and reflected waves yields the group velocity of 3153 m/s. The group velocity of the A_0-mode estimated from the dispersion diagram is about 3130 m/s, which is close to one computed from the 3D time-domain analysis.

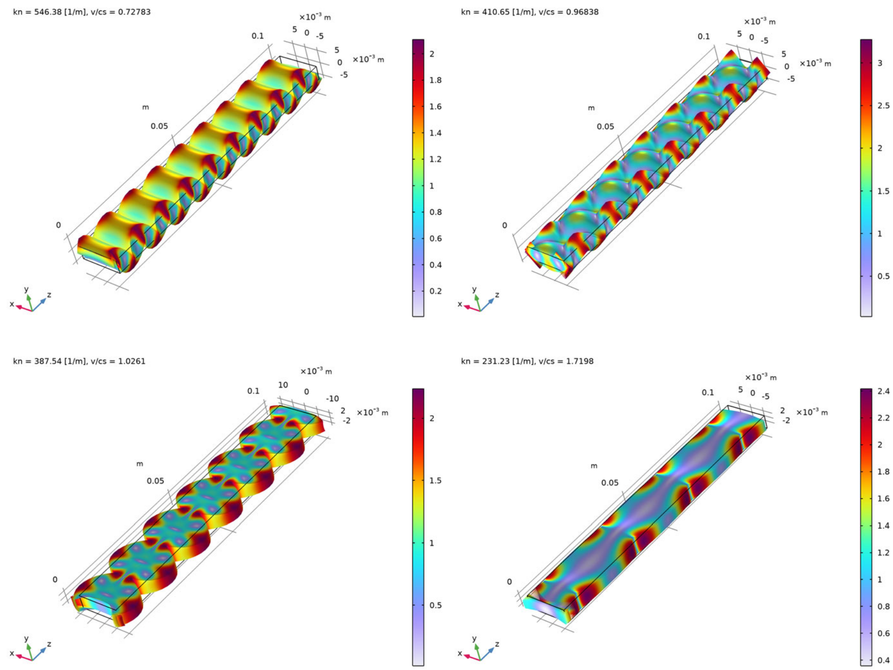

Figure 5: Profiles of the modes propagating at f_0 = 200 kHz and symmetric with respect to the sagittal plane.

The A_0-mode is clearly visible when it propagates in the plate, especially at the times before it has reached the flaw. Although this mode has the highest magnitude, it is not the only refracted wave that is exited by the transducer. The other mode is faster than the A_0-mode and is vaguely seen in Figure 6 at t = 80 μs. Its distinct shape is available from Figure 8 that shows the vertical (y-component) and longitudinal (z-component) velocity components on the top and the bottom of the plate at t = 80 μs. The vertical velocity of the slower A_0-mode is in phase on the top and the bottom of the plate, while that of the faster mode is 180° out of phase. The opposite applies to the longitudinal velocity. The faster mode propagating in the plate is the one depicted in the lower-left corner in Figure 5. It has the phase velocity of 5435 m/s, and its nascent frequency lies between 160 and 170 kHz as seen from Figure 4.

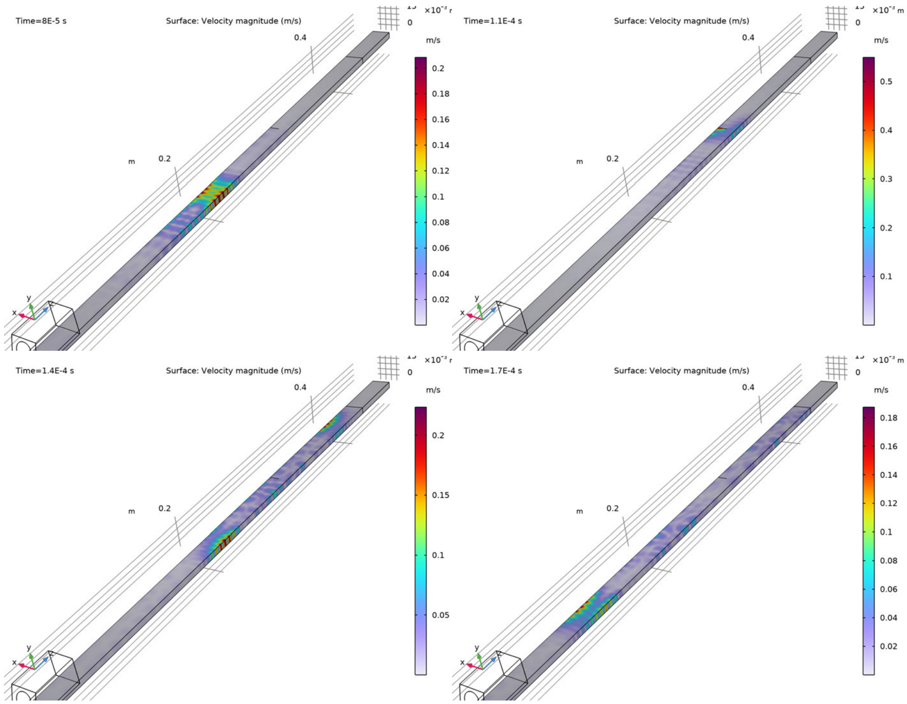

Figure 6: Wave profiles at t = 80, 110, 140, and 170 µ*s.*

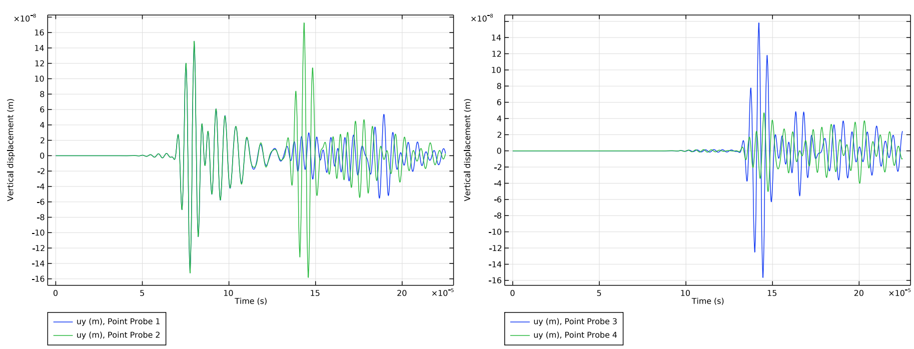

Figure 7: Vertical displacements at the listening points before (left) and after (right) the defect.

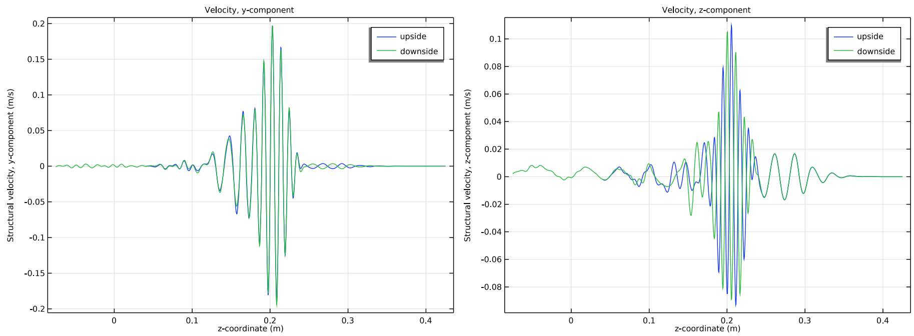

Figure 8: Velocity profiles on the top (green) and the bottom (blue) of the plate at t = 80 µs.

Reference

1. H. Lamb, “On waves in an elastic plate,” *Proc. R. Soc. Lond. A,* vol. 93, issue 648, 1917.

Application Library path: `Acoustics_Module/Ultrasound/lamb_waves_ndt_plate`

Modeling Instructions

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

NEW

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

GLOBAL DEFINITIONS

Parameters: Geometrical Parameters

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

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

3 Click **Load from File**.

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

5 In the **Label** text field, type `Parameters: Geometrical Parameters`.

Parameters: Model Parameters

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

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

3 Click **Load from File**.

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

5 In the **Label** text field, type `Parameters: Model Parameters`.

Steel

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

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

3 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Density**.

4 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho_steel`	kg/m³	Basic

6 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Pressure-Wave and Shear-Wave Speeds**.

7 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	`cp_steel`	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	`cs_steel`	m/s	Pressure-wave and shear-wave speeds

Plastic

1 Right-click **Materials** and choose **Blank Material**.

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

3 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Density**.

4 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Density	rho	rho_plast	kg/m³	Basic

6 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Pressure-Wave and Shear-Wave Speeds**.

7 Click **Add to Material**.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	cp_plast	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	cs_plast	m/s	Pressure-wave and shear-wave speeds

First, compute the modes that can propagate in the waveguide of a rectangular cross section.

ADD COMPONENT

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

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `W_plate`.

4 In the **Height** text field, type `D_plate`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Click **Build All Objects**.

MATERIALS

Steel

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

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

COMPONENT 1 (COMP1)

In the **Home** toolbar, click **Windows** and choose **Add Physics**.

ADD PHYSICS

1 Go to the **Add Physics** window.

2 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add to Component 1** in the window toolbar.

SOLID MECHANICS (SOLID)

In the **Home** toolbar, click **Windows** and choose **Add Physics**.

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

MESH 1

Mapped 1

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

Size

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

RESULTS

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

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** check box.

4 Locate the **Save Data in the Model** section. From the **Save plot data** list, choose **On**.

ROOT

In the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Mode Analysis**.

3 Click **Add Study** in the window toolbar.

STUDY 1 - MODE ANALYSIS

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

2 In the **Settings** window for **Study**, type `Study 1 - Mode Analysis` in the **Label** text field.

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

Step 1: Mode Analysis

1 In the **Model Builder** window, under **Study 1 - Mode Analysis** click **Step 1: Mode Analysis**.

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

3 In the **Mode analysis frequency** text field, type `f0`.

4 Select the **Desired number of modes** check box. In the associated text field, type `20`.

5 Find the **Search region** subsection. In the **Unit** field, type `1/m`.

6 Select the **Search for modes around** check box. In the associated text field, type `ks`.

Search for the modes within the frequency range from 0 to f_{max}.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Operating frequency)	`range(fmax/250, fmax/250, fmax)`	Hz

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

RESULTS

Dispersion Curves, Phase Velocity

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

2 In the **Settings** window for **1D Plot Group**, type Dispersion Curves, Phase Velocity in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Mode Analysis/Parametric Solutions 1 (sol2)**.

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

5 Locate the **Plot Settings** section.

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

7 Select the **y-axis label** check box. In the associated text field, type v/cs (1).

8 Locate the **Axis** section. Select the **Manual axis limits** check box.

9 In the **x minimum** text field, type 0.

10 In the **x maximum** text field, type fmax.

11 In the **y minimum** text field, type 0.

12 In the **y maximum** text field, type 2*cp_steel/cs_steel.

13 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Dispersion Curves, Phase Velocity** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 1 - Mode Analysis/Parametric Solutions 1 (sol2)**.

4 From the **Out-of-plane wave number selection** list, choose **First**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
cp_steel/cs_steel	1	Relative longitudinal wave speed
1	1	Relative shear wave speed

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

7 In the **Expression** text field, type f0.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

10 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

Global 2

1 In the **Model Builder** window, right-click **Dispersion Curves, Phase Velocity** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`omega0/solid.kn/cs_steel`	1	`Relative out-of-plane wave speed`

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

5 In the **Expression** text field, type `f0`.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Color** list, choose **Black**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

Plot the dispersion curves for the modes that propagate in the out-of-plane direction. The wave numbers of such modes will have their imaginary parts close to 0. Plot the curves that lie in the upper half plane.

Filter 1

1 Right-click **Global 2** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(abs(imag(solid.kn))<1)&&(real(solid.kn)>0)`.

4 In the **Dispersion Curves, Phase Velocity** toolbar, click **Plot**.

Dispersion Curves, Out-of-plane Wave Number

1 In the **Model Builder** window, right-click **Dispersion Curves, Phase Velocity** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Dispersion Curves, Out-of-plane Wave Number` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `kn (1/m)`.

4 Locate the **Axis** section. In the **y maximum** text field, type `omegamax/cs_steel`.

Global 1

1 In the **Model Builder** window, expand the **Dispersion Curves, Out-of-plane Wave Number** node, then click **Global 1**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
kp	1/m	Longitudinal wave number in steel
ks	1/m	Shear wave number in steel

Global 2

1 In the **Model Builder** window, click **Global 2**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
solid.kn	1/m	Out-of-plane wave number

4 In the **Dispersion Curves, Out-of-plane Wave Number** toolbar, click **Plot**.

Create an **Extrusion 2D** dataset to visualize the shapes of the modes propagating in the out-of-plane direction.

Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.
2 In the **Settings** window for **Extrusion 2D**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 1 - Mode Analysis/Parametric Solutions 1 (sol2)**.
4 Locate the **Extrusion** section. In the **z maximum** text field, type `10[cm]`.
5 In the **Resolution** text field, type `100`.
6 Click to expand the **Advanced** section. In the **Out-of-plane wave number** text field, type `solid.kn`.

Mode Shape

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Mode Shape` in the **Label** text field.
3 Locate the **Data** section. From the **Parameter value (f0 (Hz))** list, choose **2E5**.
4 From the **Out-of-plane wave number (1/m)** list, choose **546.38**.
5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Parameter indicator** text field, type `kn = eval(real(solid.kn)) [1/m], v/cs = eval(omega0/real(solid.kn)/cs_steel)`.

Surface 1

1 Right-click **Mode Shape** and choose **Surface**.

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

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

5 Click **OK**.

Deformation 1

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

Now, proceed to modeling of the selected mode propagation in the 3D plate.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

GEOMETRY 2

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

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

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

Create a five-cycle tone burst source signal.

DEFINITIONS (COMP2)

Rectangle 1 (rect1)

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

2 In the **Settings** window for **Rectangle**, locate the **Parameters** section.

3 In the **Lower limit** text field, type `0`.

4 In the **Upper limit** text field, type `4*T0`.

5 Click to expand the **Smoothing** section. Clear the **Size of transition zone** check box.

Normal Velocity Source

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

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

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

4 Locate the **Definition** section. In the **Expression** text field, type `sin(omega0*t)*(1 - cos(omega0*t/4))*rect1(t)`.

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

6 Locate the **Units** section. In the **Function** text field, type `m/s`.

7 In the table, enter the following settings:

Argument	Unit
t	s

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

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

9 Click **Plot**.

Wedge

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

2 In the **Settings** window for **Explicit**, type `Wedge` in the **Label** text field.

3 Select Domain 1 only.

Plate

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

2 In the **Settings** window for **Explicit**, type `Plate` in the **Label** text field.

3 Select Domains 2–4 only.

AL

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

2 In the **Settings** window for **Explicit**, type `AL` in the **Label** text field.

3 Select Domains 2 and 4 only.

Probes

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

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

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

4 Select Points 27, 28, 42, and 44 only.

5 In the **Label** text field, type `Probes`.

Exterior Boundaires

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

2 In the **Settings** window for **Adjacent**, type `Exterior Boundaires` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click + **Add**.

4 In the **Add** dialog box, in the **Input selections** list, choose **Wedge** and **Plate**.

5 Click **OK**.

Identity Boundary Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Identity Boundary Pair**.

2 Select Boundary 23 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

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

5 Select Boundary 7 only.

MATERIALS

Steel

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

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

3 From the **Selection** list, choose **Plate**.

4 In the **Label** text field, type `Steel`.

Plastic

1 Right-click **Materials** and choose **More Materials>Material Link**.

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

3 From the **Selection** list, choose **Wedge**.

4 Locate the **Link Settings** section. From the **Material** list, choose **Plastic (mat2)**.

5 In the **Label** text field, type `Plastic`.

COMPONENT 2 (COMP2)

In the **Home** toolbar, click **Windows** and choose **Add Physics**.

ADD PHYSICS

1 Go to the **Add Physics** window.

2 In the tree, select **Acoustics>Elastic Waves>Elastic Waves, Time Explicit (elte)**.

3 Click **Add to Component 2** in the window toolbar.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1 - Mode Analysis**.

ELASTIC WAVES, TIME EXPLICIT (ELTE)

Elastic Waves, Time Explicit Model 1

In the **Model Builder** window, under **Component 2 (comp2)>Elastic Waves, Time Explicit (elte)** click **Elastic Waves, Time Explicit Model 1**.

Compute Displacement 1

1 In the **Physics** toolbar, click **Attributes** and choose **Compute Displacement**.

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

3 From the **Selection** list, choose **Probes**.

Prescribed Velocity 1

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

2 Select Boundary 8 only.

3 In the **Settings** window for **Prescribed Velocity**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Boundary System 2 (sys2)**.

5 Locate the **Prescribed Velocity** section. Specify the $\mathbf{v}_0$ vector as

0	t1
0	t2
vn(t)	n

Low-Reflecting Boundary 1

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

2 Select Boundaries 1, 4, 6, 10, 13, and 20 only.

Fracture 1

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

2 Select Boundary 26 only.

Create four domain probe points along the wave propagation path (two before and two after the defect) where the vertical displacements will be recorded.

DEFINITIONS (COMP2)

Point Probe 1 (point1)

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

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

3 Click **Clear Selection**.

4 Select Point 42 only.

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

6 Select the **Description** check box. In the associated text field, type `uy`.

Point Probe 2 (point2)

1 Right-click **Point Probe 1 (point1)** and choose **Duplicate**.

2 Select Point 27 only.

Point Probe 3 (point3)

1 Right-click **Point Probe 2 (point2)** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Point 44 only.

Point Probe 4 (point4)

1 Right-click **Point Probe 3 (point3)** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Point 28 only.

Point Probe 1 (point1), Point Probe 2 (point2), Point Probe 3 (point3), Point Probe 4 (point4)

1 In the **Model Builder** window, under **Component 2 (comp2)>Definitions**, Ctrl-click to select **Point Probe 1 (point1)**, **Point Probe 2 (point2)**, **Point Probe 3 (point3)**, and **Point Probe 4 (point4)**.

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

y-displacement

In the **Settings** window for **Group**, type `y-displacement` in the **Label** text field.

Absorbing Layer 1 (ab1)

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

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

3 From the **Selection** list, choose **AL**.

MESH 2

Mapped 1

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

2 Select Boundary 13 only.

Swept 1

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

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

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

4 From the **Selection** list, choose **Plate**.

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 `v_lamb/(1.5*2*f0)`.

Free Tetrahedral 1

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

Size 1

1 Right-click **Free Tetrahedral 1** and choose **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.

5 Select the **Maximum element size** check box. In the associated text field, type `cp_plast/(1.5*2*f0)`.

Free Tetrahedral 1

1 In the **Model Builder** window, click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, click to expand the **Element Quality Optimization** section.

3 From the **Optimization level** list, choose **High**.

4 Select the **Avoid too small elements** check box.

5 Click **Build All**.

ROOT

In the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Time Dependent**.

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

STUDY 2 - WAVE PROPAGATION

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

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2 - Wave Propagation** 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, T0, 45*T0)`.

4 Click to expand the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, select **Exterior Boundaires** in the **Selections** list.

7 Click **OK**.

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

RESULTS

Surface 1

In the **Model Builder** window, expand the **Results>Velocity Magnitude (elte)** node, then click **Surface 1**.

Selection 1

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

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

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

4 From the **Selection** list, choose **Plate**.

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 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

5 Click **OK**.

Probe Plot Displacements

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 6**.

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Vertical displacement (m)`.

5 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

6 From the **Position** list, choose **Bottom**.

7 In the **Number of rows** text field, type `2`.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Plot Displacements** node, then click **Probe Table Graph 1**.

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

3 In the **Columns** list, choose **uy (m), Point Probe 3** and **uy (m), Point Probe 4**.

4 In the **Columns** list, choose **uy (m), Point Probe 1** and **uy (m), Point Probe 2**.

Velocity, y-component

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

2 In the **Settings** window for **1D Plot Group**, type `Velocity, y-component` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Wave Propagation/ Solution 253 (4) (sol253)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **8E-5**.

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

Line Graph 1

1 Right-click **Velocity, y-component** and choose **Line Graph**.

2 Select Edges 43, 45, and 46 only.

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

4 In the **Expression** text field, type `v2y`.

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

6 In the **Expression** text field, type `z`.

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

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

9 In the table, enter the following settings:

Legends
upside

Line Graph 2

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

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

3 Click **Clear Selection**.

4 Select Edge 29 only.

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

Legends
downside

6 In the **Velocity, y-component** toolbar, click **Plot**.

Appendix — Geometry 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 Click **Done**.

GLOBAL DEFINITIONS

Geometry Parameters

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

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

3 Click **Load from File**.

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

5 In the **Label** text field, type `Geometry Parameters`.

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `W_plate`.

4 In the **Height** text field, type `D_plate`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Extrude 1 (ext1)

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
`0.1*L_plate`
`1.1*L_plate`
`1.2*L_plate`

Move 1 (mov1)

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

2 Select the object **ext1** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **x** text field, type `W_plate/2`.

5 In the **y** text field, type `-D_plate/2`.

6 In the **z** text field, type `-L_plate/4`.

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 `W_wedge`.

4 In the **Depth** text field, type `H_wedge`.

5 In the **Height** text field, type `L_wedge`.

6 Locate the **Position** section. In the **x** text field, type `(W_plate - W_wedge)/2`.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Normal vector**.

4 Find the **Normal vector** subsection. In the **y** text field, type `cos(alpha)`.

5 In the **z** text field, type `-sin(alpha)`.

6 Find the **Point on plane** subsection. In the **y** text field, type `H_wedge - L_slope*sin(alpha)`.

Work Plane 2 (wp2)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

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

4 Locate the **Position** section. In the **xw** text field, type `-L_slope*sin(alpha)/2`.

5 In the **yw** text field, type `-W_plate/2`.

Partition Objects 1 (par1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Booleans and Partitions>Partition Objects**.

2 Select the object **blk1** only.

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

4 From the **Partition with** list, choose **Work plane**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

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

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

4 On the object **par1**, select Domain 2 only.

Parametric Surface 1 (ps1)

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

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

3 Find the **First parameter** subsection. In the **Minimum** text field, type `W_plate/2`.

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

5 Find the **Second parameter** subsection. In the **Minimum** text field, type `-D_plate`.

6 In the **Maximum** text field, type `0`.

7 Locate the **Expressions** section. In the **x** text field, type `s1`.

8 In the **y** text field, type `s2`.

9 In the **z** text field, type `0.3`.

10 Click **Build Selected**.

Point 1 (pt1)

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

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

3 In the **z** text field, type `0.2`.

Point 2 (pt2)

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

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

3 In the **z** text field, type `0.4`.

Copy 1 (copy1)

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

2 Select the objects **pt1** and **pt2** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `W_plate`.

Union 1 (uni1)

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

2 Select the objects **copy1(1)**, **copy1(2)**, **mov1**, **ps1**, **pt1**, and **pt2** only.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Select the **Create imprints** check box.

6 Click **Build Selected**.

Ignore Faces 1 (igf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Faces**.

2 On the object **fin**, select Boundary 31 only.

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