Created in COMSOL Multiphysics 6.2

Angle Beam Nondestructive Testing

Introduction

Angle beam ultrasonic units are used for nondestructive testing (NDT) of solid objects, such as metal pipes. They are especially useful for detecting flaws in and around welding areas, such as pores, small cracks, lack of fusion, and so on. Angle beam NDT is often used where the straight beam testing struggles to find defects, for example, when the cracks are vertical and thin and thus not detectable because of small amount of reflection. The operating principle of angle beam NDT lies in the conversion of a longitudinal (compression) wave sent by the transducer into a refracted shear (transverse) wave in the test sample. The shear wave is then reflected by the flaws in the test object. Compared to longitudinal waves, shear waves have lower attenuation and shorter wavelength, which makes them capable of detecting smaller defects.

In this tutorial, the *Elastic Waves, Time Explicit* physics interfaces is used to model wave propagation in linear elastic media. The interface solves the linear elastic wave equation written in the velocity-strain form using the discontinuous Galerkin finite element method (dG-FEM) and an explicit time integration scheme. This approach is well suited for solving large-scale transient problems. The piezoelectric part of the transducer is modeled with the *Piezoelectric Effect, Time Explicit* multiphysics feature that couples the *Elastic Waves, Time Explicit* and the *Electrostatics* physics interfaces. The model takes advantage of a geometry assembly and a nonconforming mesh.

Model Definition

The angle beam transducers are designed to generate a refracted shear wave in the test sample that propagates at a certain angle to the sample surface, typically, from 40° to 70°. The angle or refraction depends on the material properties of the pair transducer wedge/test sample and the angle of incidence of the ultrasonic signal. The incident longitudinal wave that passes through the wedge with the speed c_{p1} results in refracted longitudinal and shear waves that propagate in the test sample with the speeds c_{p2} and c_{s2}, respectively (see Figure 1). The angles of refraction are defined according to Snell's law as

$$\frac{\sin\alpha}{c_{\mathrm{p1}}} = \frac{\sin\beta}{c_{\mathrm{p2}}} = \frac{\sin\gamma}{c_{\mathrm{s2}}}$$

The first and second critical angles are those that yield $\beta = 90°$ and $\gamma = 90°$, respectively. For the pair of plastic wedge with c_{p1} = 2080 m/s and aluminum test object with c_{p2} = 6200 m/s and c_{s2} = 3120 m/s, this yields first and second critical angles of about 19.5° and 44.7°, respectively. In this tutorial, the shear wave refraction angle is $\gamma = 45°$, which results in the angle of incidence $\alpha = 28°$. The angle of incidence lies between the first and the second critical angles, and therefore a refracted longitudinal wave will skim along the surface of the test sample.

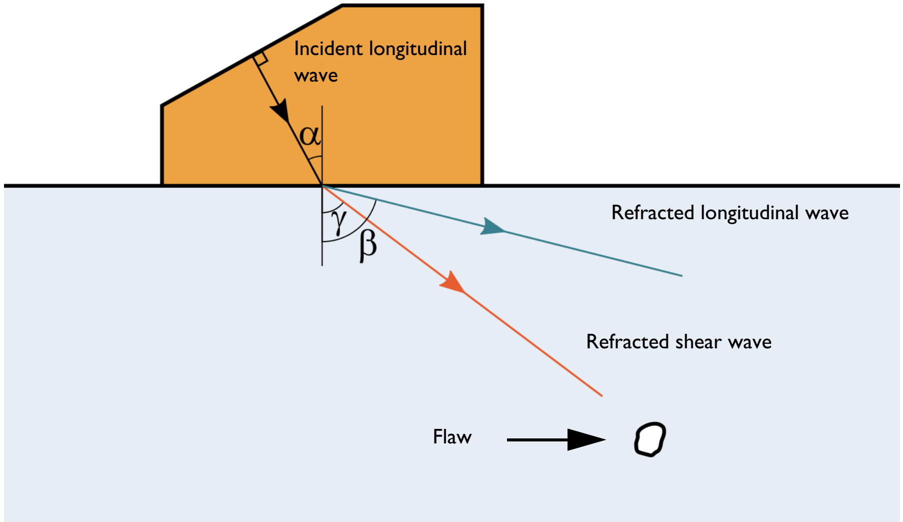


Figure 1: Wave refraction at the wedge/test sample boundary.

The angle beam NDT units usually operate at frequencies between 1 and 10 MHz. The unit is driven by voltage signal applied to the piezoelectric transducer. In this tutorial, the center frequency of the signal f_0 = 1 MHz. The transducer is attached to the wedge via a matching layer. The transducer and the matching layer are half and one-quarter wavelength thick, respectively. The transducer is surrounded by a backing layer block (also called damping block) at the back. The main parts of the setup are shown in Figure 2.

The matching layer and the damping block materials should be chosen in such a way that the acoustic impedance of the former is close to that of the transducer, while the impedance of the latter is close to the geometric mean of those of the transducer and the wedge. That is,

$$Z_{\text{matching}} = \sqrt{Z_{\text{transducer}} Z_{\text{wedge}}} \quad .$$

Typical materials used for the matching and backing layers are alumina/epoxy or tungsten/epoxy composites. The desired acoustical properties are achieved by the amount of the alumina or tungsten powder in the composite. The elastic properties of such a two-phase composite may be defined based on the Devaney model (see Ref. 1). The properties of the angle beam unit components used in this model are listed in Table 1.

The NDT unit is put upon an aluminum test specimen that has a defect in the form of a zero-thickness fracture as shown in Figure 2.

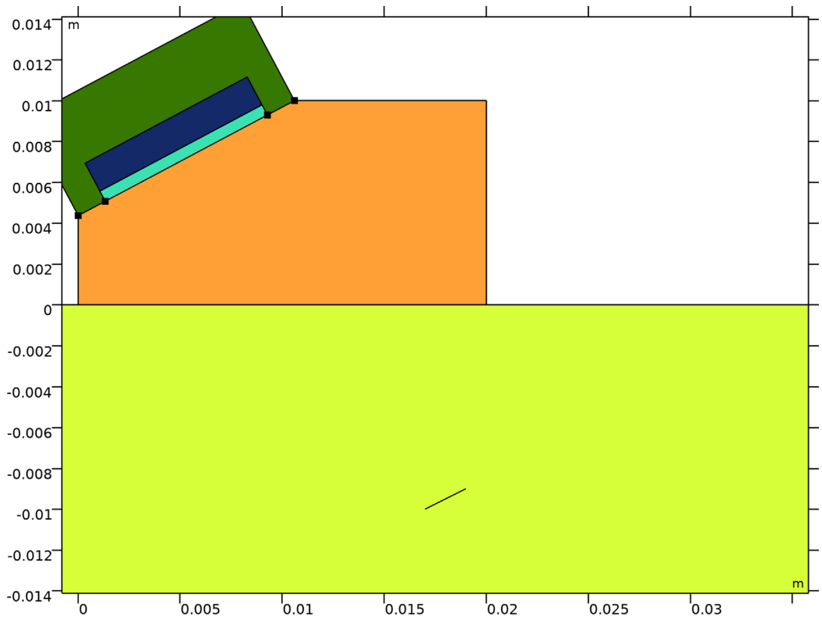


Figure 2: Angle beam NDT setup: test sample (yellow), wedge (orange), piezoelectric transducer (dark blue), matching (cyan) and backing (green) layers.

TABLE 1: PROPERTIES OF THE NDT UNIT.

Part	**Material**	**Density, kg/m3**	**Longitudinal wave speed, m/s**	**Shear wave speed, m/s**
Transducer	PZT-5H	7500	4620	1750
Wedge	Acrylic plastic	1190	2080	1000
Matching layer	Alumina/Epoxy	2280	3400	1920
Damping block	Tungsten/Epoxy	6580	1500	775

Results and Discussion

The evolution of the signal traveling from the transducer to the defect in the test sample is illustrated in Figure 3. Note that the longitudinal and shear waves are separated: the former are shown in blue and the latter in orange. The longitudinal wave generated by the transducer travels through the wedge (t = 4 μs). Then the wave hits the surface of the test object and a refracted shear wave begins to propagate through it (t = 6 μs). The shear wave travels toward the defect (t = 8 μs), hits it and becomes reflected. The reflected wave travels back toward the testing unit (t = 12 μs).

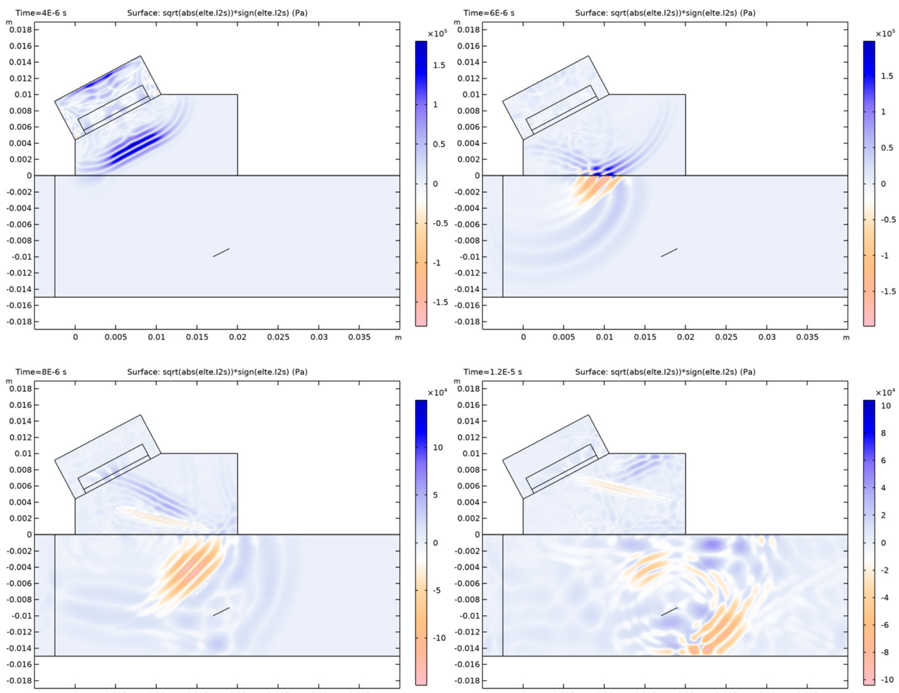


Figure 3: Wave profiles at t = 4, 6, 8, and 12 μs.

Figure 4 shows the voltage signal on the transducer terminal for the test sample with defect on top of the reference signal recorded when the inspected object has no defects. The signals zoomed around the time when the reflected wave reached the transducer are depicted in Figure 5. It is seen that the received signal slowly starts to deviate from the reference soon after t = 15 μs, while the main portion of the reflected signal arrives at the transducer after t = 17 μs.

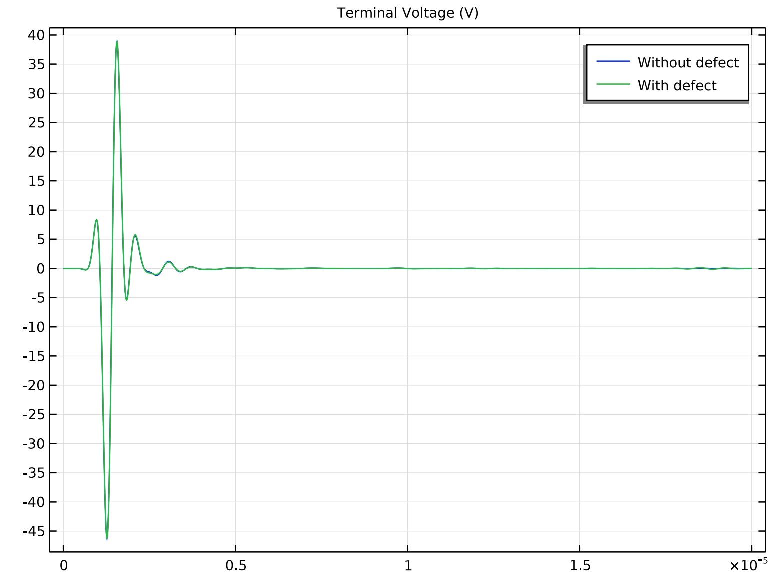


Figure 4: Voltage signal on the transducer terminal for the test samples with (green) and without (blue) the defect.

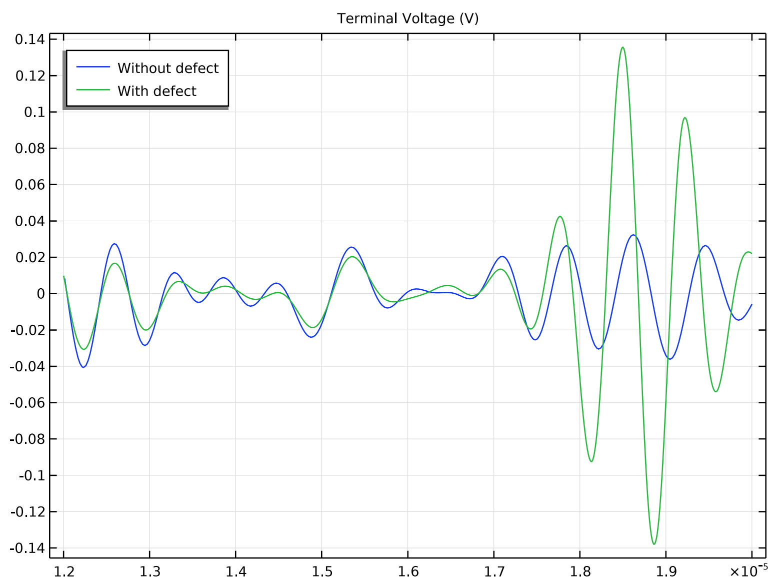


Figure 5: Voltage signal zoomed around the arrival time of the reflected wave.

Notes About the COMSOL Implementation

DEFECT MODELING

The defect considered in this model is a zero-thickness fracture which is modeled with the *Fracture* boundary condition available in the *Elastic Waves, Time Explicit* physics interface. This boundary condition implements the concept of an imperfectly bonded interface between two elastic domains (see Ref. 2 for the theoretical details). The properties of the fracture are given through the boundary stiffness which has the unit of stress per length. The case of zero boundary stiffness used in this model corresponds to an interior free surface.

GEOMETRY AND MESH

The model geometry is an assembly, which makes the parts of the geometry separated from one another and connected via *Identity Boundary Pairs*. The nodes of the generated mesh elements do not have to match on either side for the pairs thus making the mesh nonconformal. For wave propagation problems, feasible results are achieved when the mesh resolves the wavelengths of the propagating waves. The wavelength, in its turn, depends on the speed of sound in the material. Thus materials with lower speed of sound require finer mesh than those with higher speed of sound. The use of a nonconformal mesh in this tutorial makes it possible to reduce the number of DOFs solved for in the model. In this model, the mesh resolves the wavelength of the shear waves in each material, and it follows from Table 1 that one mesh element in the transducer domain corresponds to at least two mesh elements in the damping block domain on the other side of the pair as seen in Figure 6. The same is valid for the wedge/test sample pair.

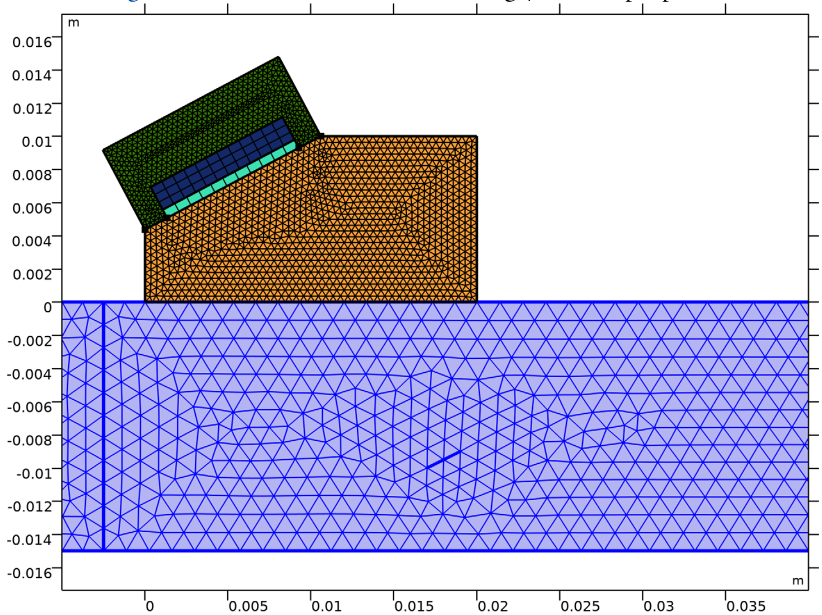


Figure 6: Model mesh.

References

1. R.A. Webster, "Passive materials for high frequency piezocomposite ultrasonic transducers," PhD thesis, University of Birmingham, 2010.

2. M. Schoenberg, "Elastic wave behavior across linear slip interfaces," *J. Acoust. Soc. Am.*, vol. 65, issue 5, 1980.

Application Library path: `Acoustics_Module/Ultrasound/angle_beam_ndt`

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 Click **Done**.

GEOMETRY 1

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 `angle_beam_ndt_geom_sequence.mph`.

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

GLOBAL DEFINITIONS

Geometry

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

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

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 `angle_beam_ndt_parameters.txt`.

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

Create a voltage source given by a modulated Gaussian pulse with the center frequency f_0.

Voltage Source

1 In the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

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

3 In the **Function name** text field, type `V0`.

4 Locate the **Definition** section. In the **Expression** text field, type `100*exp(-((t - 2*T0)/(T0/2))^2)*sin(2*pi*f0*t)`.

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

6 Locate the **Units** section. In the **Function** text field, type `V`.

7 In the table, enter the following settings:

Argument	Unit
t	s

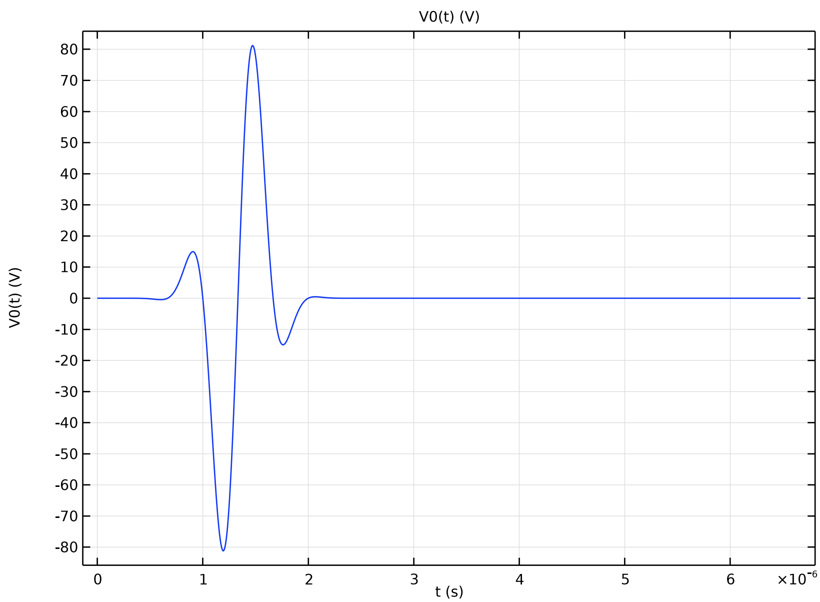
8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	t	0	10*T0	0	s

9 Click **Create Plot**.

RESULTS

Voltage Source



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

Load the voltage signal received by the transducer when testing a sample without any defects.

GLOBAL DEFINITIONS

Interpolation 1 (int1)

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

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

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

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

Note that you can click **Import** to embed the imported file in the model.

6 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
V_no_defect	1

7 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **None**.

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

Function	Unit
V_no_defect	V

9 In the **Argument** table, enter the following settings:

Argument	Unit
Column 1	s

10 Click **Create Plot**.

RESULTS

Terminal Voltage

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

Function 1

1 In the **Model Builder** window, expand the **Terminal Voltage** node, then click **Function 1**.

2 In the **Settings** window for **Function**, click to expand the **Legends** section.

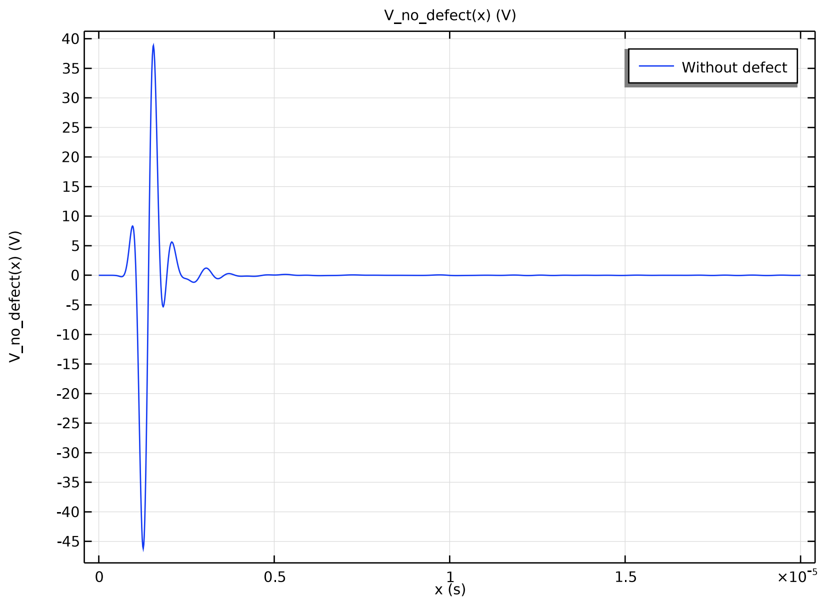
3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
`Without defect`

6 In the **Terminal Voltage** toolbar, click **Plot**.



Create selections to simplify the model setup.

DEFINITIONS

Transducer

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

2 Select Domain 6 only.

3 In the **Settings** window for **Explicit**, type `Transducer` in the **Label** text field.

4 Locate the **Color** section. From the **Color** list, choose **Color 5**.

Matching Layer

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

2 Select Domain 7 only.

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

4 Locate the **Color** section. From the **Color** list, choose **Color 10**.

Damping Block

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

2 Select Domain 4 only.

3 In the **Settings** window for **Explicit**, type `Damping Block` in the **Label** text field.

4 Locate the **Color** section. From the **Color** list, choose **Color 9**.

Wedge

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

2 Select Domain 5 only.

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

4 Locate the **Color** section. From the **Color** list, choose **Color 18**.

Test Sample

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

2 Select Domains 1–3 only.

3 In the **Settings** window for **Explicit**, type `Test Sample` in the **Label** text field.

4 Locate the **Color** section. From the **Color** list, choose **Color 4**.

Add a coordinate system to align the piezoelectric crystal in such a way that its Z-axis is perpendicular to the transducer surface.

Transducer Coordinate System

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

2 In the **Settings** window for **Base Vector System**, locate the **Base Vectors** section.

3 From the **Out-of-plane index** list, choose **2**.

4 In the table, enter the following settings:

	x	**y**
x1	`cos(alpha)`	`sin(alpha)`
x3	`-sin(alpha)`	`cos(alpha)`

5 Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

6 In the **Label** text field, type `Transducer Coordinate System`.

Create a view to zoom in on the area of the transducer and the defect.

View 2

1. In the **Model Builder** window, right-click **Definitions** and choose **View**.
2. In the **Settings** window for **View**, locate the **View** section.
3. Select the **Lock axis** check box.

Axis

1. In the **Model Builder** window, expand the **View 2** node, then click **Axis**.
2. In the **Settings** window for **Axis**, locate the **Axis** section.
3. In the **x minimum** text field, type `-0.005`.
4. In the **x maximum** text field, type `0.04`.
5. In the **y minimum** text field, type `-0.012`.
6. In the **y maximum** text field, type `0.012`.
7. Click **Update**.

   The result should look like the one shown in Figure 2.

Now, proceed to setting up the physics. Note that the model geometry is an assembly and therefore each physics interface automatically imposes the **Continuity** boundary condition on all boundary pairs.

ADD PHYSICS

1. In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Acoustics>Elastic Waves>Elastic Waves, Time Explicit (elte)**.
4. Click **Add to Component 1** in the window toolbar.

ELASTIC WAVES, TIME EXPLICIT (ELTE)

Elastic Waves, Time Explicit Model 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Elastic Waves, Time Explicit (elte)** click **Elastic Waves, Time Explicit Model 1**.
2. In the **Settings** window for **Elastic Waves, Time Explicit Model**, locate the **Linear Elastic Material** section.
3. From the **Specify** list, choose **Pressure-wave and shear-wave speeds**.

Piezoelectric Material 1

1. In the **Physics** toolbar, click **Domains** and choose **Piezoelectric Material**.
2. In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

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

Low-Reflecting Boundary 1

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

2 Select Boundaries 1, 13, and 15 only.

Fracture 1

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

2 Select Boundary 8 only.

Elastic Waves, Time Explicit Model 1

In the **Model Builder** window, click **Elastic Waves, Time Explicit Model 1**.

Damping 1

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

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

3 From the **Selection** list, choose **Matching Layer**.

4 Locate the **Damping Settings** section. From the **Input parameters** list, choose **Damping ratios**.

5 In the f_1 text field, type `0.99*f0`.

6 In the ζ_1 text field, type `5e-2`.

7 In the f_2 text field, type `1.01*f0`.

8 In the ζ_2 text field, type `5e-2`.

9 Right-click **Damping 1** and choose **Duplicate**.

Damping 2

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

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

3 From the **Selection** list, choose **Damping Block**.

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

Damping 3

1 In the **Model Builder** window, click **Damping 3**.

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

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

4 Locate the **Damping Settings** section. In the ζ_1 text field, type `1e-2`.

5 In the ζ_2 text field, type `1e-2`.

6 Right-click **Damping 3** and choose **Duplicate**.

Damping 4

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

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

3 From the **Selection** list, choose **Test Sample**.

4 Locate the **Damping Settings** section. In the ζ_1 text field, type `0.5e-2`.

5 In the ζ_2 text field, type `0.5e-2`.

Piezoelectric Material 1

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

2 In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Transducer Coordinate System (sys2)**.

Mechanical Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.

2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.

3 From the **Input parameters** list, choose **Damping ratios**.

4 In the f_1 text field, type `0.99*f0`.

5 In the ζ_1 text field, type `0.5e-2`.

6 In the f_2 text field, type `1.01*f0`.

7 In the ζ_2 text field, type `0.5e-2`.

ADD PHYSICS

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

2 In the tree, select **AC/DC>Electric Fields and Currents>Electrostatics (es)**.

3 Click **Add to Selection** in the window toolbar.

ELECTROSTATICS (ES)

1 In the **Settings** window for **Electrostatics**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Transducer**.

Charge Conservation, Piezoelectric 1

1 Right-click **Component 1 (comp1)>Electrostatics (es)** and choose **Charge Conservation, Piezoelectric**.

2 In the **Settings** window for **Charge Conservation, Piezoelectric**, locate the **Domain Selection** section.

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

Ground 1

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

2 Select Boundary 33 only.

Terminal 1

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

2 Select Boundary 32 only.

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

4 From the **Terminal type** list, choose **Circuit**.

ADD PHYSICS

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

2 In the tree, select **AC/DC>Electrical Circuit (cir)**.

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

4 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ELECTRICAL CIRCUIT (CIR)

Voltage Source 1 (V1)

1 Right-click **Component 1 (comp1)>Electrical Circuit (cir)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
n	0

4 Locate the **Device Parameters** section. In the v_{src} text field, type `V0(t)`.

Resistor 1 (R1)

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `2[ohm]`.

External I-Terminal 1 (termI1)

1 In the **Electrical Circuit** toolbar, click **External I-Terminal**.

2 In the **Settings** window for **External I-Terminal**, locate the **Node Connections** section.

3 In the **Node name** text field, type `2`.

4 Locate the **External Terminal** section. From the V list, choose **Terminal voltage (es/term1)**.

MULTIPHYSICS

Piezoelectricity, Time Explicit 1 (pzete1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multiphysics** and choose **Piezoelectricity, Time Explicit**.

Define a **Global Variable Probe** that will record the voltage signal at the terminal.

DEFINITIONS

Global Variable Probe 1 (var1)

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

2 In the **Settings** window for **Global Variable Probe**, type `V_with_defect` in the **Variable name** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electrostatics>Terminals>es.V0_1 - Terminal voltage - V**.

4 Locate the **Expression** section. Select the **Description** check box.

5 Click to expand the **Table and Window Settings** section.

Add the absorbing layers (sponge layers) used to truncate the computational domain.

Absorbing Layer 1 (ab1)

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

2 Select Domains 1 and 3 only.

Now, set up the materials.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Aluminum**.

4 Click **Add to Component** in the window toolbar.

5 In the tree, select **Built-in>Acrylic plastic**.

6 Click **Add to Component** in the window toolbar.

MATERIALS

Acrylic plastic (mat2)

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

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

ADD MATERIAL

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

2 In the tree, select **Built-in>Lead Zirconate Titanate (PZT-5H)**.

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

4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat3)

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

2 From the **Selection** list, choose **Transducer**.

Matching Material

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

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

3 From the **Selection** list, choose **Matching Layer**.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	cp_match	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	cs_match	m/s	Pressure-wave and shear-wave speeds
Density	rho	rho_match	kg/m³	Basic

5 In the **Label** text field, type `Matching Material`.

Damping Material

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

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

3 From the **Selection** list, choose **Damping Block**.

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

Property	Variable	Value	Unit	Property group
Pressure-wave speed	cp	cp_damp	m/s	Pressure-wave and shear-wave speeds
Shear-wave speed	cs	cs_damp	m/s	Pressure-wave and shear-wave speeds
Density	rho	rho_damp	kg/m³	Basic

5 In the **Label** text field, type `Damping Material`.

Create a mesh. The mesh should be fine enough to resolve the shortest wavelength in each material. Note that the mesh element nodes lie misaligned on either side of the boundary pairs. As a result, the mesh elements adjacent to the pairs have different size as shown in Figure 6, which helps to reduce the number of DOFs in the model.

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 6 and 7 only.

Distribution 1

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

2 Select Boundary 34 only.

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

4 In the **Number of elements** text field, type `3`.

Size 1

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

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

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

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

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

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

Size 2

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

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

3 From the **Selection** list, choose **Matching Layer**.

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

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

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

Free Triangular 1

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

Size 1

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

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

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

4 From the **Selection** list, choose **Damping Block**.

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

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

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

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Wedge**.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Maximum element size** check box. In the associated text field, type `cs_plast/f0/1.5`.

Size 3

1 Right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Test Sample**.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section.
7 Select the **Maximum element size** check box. In the associated text field, type `cs_al/f0/1.5`.
8 Click **Build All**.

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

STUDY 1

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, T0/5, 30*T0)`.

This setting saves the solution at times multiple to $T_0/5$ in the whole computational domain. It only influences the stored solution (and thus the file size). The internal time steps taken by the solver are automatically controlled by COMSOL to fulfill the appropriate CFL condition.

On the other hand, the terminal voltage at the probe will be computed for each time step taken by the solver thus providing a much higher temporal resolution of the result.

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

All the plots are depicted in the previous sections of the documentation.

RESULTS

Terminal Voltage

1 In the **Model Builder** window, under **Results** click **Terminal Voltage**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 In the **Title** text area, type `Terminal Voltage (V)`.

Table Graph 1

1 Right-click **Terminal Voltage** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
`With defect`

6 In the **Terminal Voltage** toolbar, click **Plot**.

Pressure and Shear Waves

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

2 In the **Settings** window for **2D Plot Group**, type `Pressure and Shear Waves` in the **Label** text field.

3 Locate the **Data** section. From the **Time (s)** list, choose **1.2E-5**.

Surface 1

1 Right-click **Pressure and Shear Waves** and choose **Surface**.

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

3 In the **Expression** text field, type `sqrt(abs(elte.I2s))*sign(elte.I2s)`.

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

5 In the **Color Table** dialog box, select **Aurora>Twilight** in the tree.

6 Click **OK**.

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

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

9 In the **Pressure and Shear Waves** 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 **2D**.

2 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

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

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`.

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

Point 1 (pt1)

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

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

3 In the **y** text field, type `H-L*sin(alpha)`.

Point 2 (pt2)

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 `L*cos(alpha)`.

4 In the **y** text field, type `H`.

Line Segment 1 (ls1)

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

2 On the object **pt1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **pt2**, select Point 1 only.

Partition Objects 1 (par1)

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

2 Select the object **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Tool objects**.

5 Select the object **ls1** only.

6 Click **Build Selected**.

Delete Entities 1 (del1)

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

2 Click in the **Graphics** window and then press Ctrl+D to clear all objects.

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

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

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

6 Click **Build Selected**.

Point 3 (pt3)

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 `(L-D)/2*cos(alpha)`.

4 In the **y** text field, type `H-(L+D)/2*sin(alpha)`.

Point 4 (pt4)

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 `(L+D)/2*cos(alpha)`.

4 In the **y** text field, type `H-(L-D)/2*sin(alpha)`.

Rectangle 2 (r2)

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

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

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

4 In the **Height** text field, type `H_pzt+H_match`.

5 Locate the **Position** section. In the **x** text field, type `(L-D)/2*cos(alpha)`.

6 In the **y** text field, type `H-(L+D)/2*sin(alpha)`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `alpha`.

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

Layer name	Thickness (m)
Layer 1	H_match

Rectangle 3 (r3)

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 `L`.

4 In the **Height** text field, type `3.5*H_pzt`.

5 Locate the **Position** section. In the **y** text field, type `H-L*sin(alpha)`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `alpha`.

Difference 1 (dif1)

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

2 Select the object **r3** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **r2** only.

6 Select the **Keep objects to subtract** check box.

7 Click **Build Selected**.

Split 1 (spl1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Split**.

2 Select the object **r2** only.

3 In the **Settings** window for **Split**, click **Build Selected**.

Partition Edges 1 (pare1)

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

2 On the object **del1**, select Boundary 3 only.

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

4 From the **Type of specification** list, choose **Vertex projection**.

5 On the object **pt3**, select Point 1 only.

6 On the object **pt4**, select Point 1 only.

7 Click **Build Selected**.

Rectangle 4 (r4)

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_ts`.

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

5 Locate the **Position** section. In the **x** text field, type `-W_ts/10`.

6 In the **y** text field, type `-H_ts`.

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

Layer name	Thickness (m)
Layer 1	H_ts/2

8 Clear the **Layers on bottom** check box.

9 Select the **Layers to the left** check box.

10 Select the **Layers to the right** check box.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)
0.017	-0.01
0.019	-0.009

Union 1 (uni1)

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

2 Select the objects **pol1** and **r4** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

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 Select the **Create imprints** check box.

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