Created in COMSOL Multiphysics 5.5

Ultrasonic Flowmeter with Piezoelectric Transducers: Coupling Between FEM and DG

Introduction

Ultrasonic flowmeters are used to determine the velocity of the fluid flowing through a pipe. The principle is to send an ultrasonic signal across the flow at a skew angle. In case of no flow, the transmitting time between the transmitter and the receiver is the same for the signals sent in the upstream and the downstream directions. Otherwise, the downstream traveling wave moves faster than the one traveling upstream. In many cases piezoelectric transducers are used to send and receive the ultrasonic wave.

This tutorial example shows how to simulate an ultrasonic flowmeter with piezoelectric transducers in the simplified no-flow case. The finite element method (FEM) is used to model the piezoelectric transducers, whereas the modeling of the ultrasonic wave propagation is based on the discontinuous Galerkin (DG) method. The whole model is split into three sub-models. A FEM to DG one-way coupling is used to send the wave from the transmitter, and a DG to FEM one-way coupling is used for the receiver.

Model Definition

The flowmeter geometry used in this model is the same as given in the model Ultrasound Flowmeter with Generic Time-of-Flight Configuration. It consists of a main pipe and a signal pipe of a smaller diameter. The signal tube is tilted to the main pipe at the angle $\alpha = 45^o$. Two piezoelectric transducers placed at either end of the signal pipe operate as the transmitter and the receiver. An input voltage signal applied to the transmitter results in the mechanical deformation of the piezoelectric transducer, which is due to the inverse piezoelectric effect. The mechanical deformation in its turn generates an acoustic wave in the fluid. When the acoustic wave reaches the receiver, the inverse process takes place: the mechanical load is being converted into an electric signal because of the direct piezoelectric effect.

As previously mentioned, this model studies the propagation of the acoustic wave is the simplified no-flow condition. Thus the main focus is the interaction between the fluid and the solid and the conversion *input electric signal - acoustic wave - output electric signal*. The background flow can be added if necessary, as is done in the model Ultrasound Flowmeter with Generic Time-of-Flight Configuration.

This model solves the full three-dimensional problem with the symmetry condition imposed on the sagittal plane of the structure. The model involves the following physical phenomena.

PIEZOELECTRICITY: SOLID MECHANICS AND ELECTROSTATICS

The transmitter and the receiver can be used interchangeably; therefore, the principles of their construction are the same. The main part is a disk made of a piezoelectric material (for example, PZT) that is used for conversion of between the electric and the mechanical waves.

A direct propagation of the mechanical wave in the piezoelectric transducer in form of the acoustic wave in the fluid will result in significant reflections from the solid/fluid interface and, consequently, losses. This is due to the impedance mismatch: for water, the characteristic acoustic impedance $Z_{\text{water}} \approx 1.5 \times 10^6\ \text{kg/m}^2\cdot\text{s}$, and for PZT-5H, $Z_{\text{PZT}} \approx 30 \times 10^6\ \text{kg/m}^2\cdot\text{s}$. Therefore, a matching layer is required to minimize the losses. Its size is taken to be 1/4 of the wavelength, and its impedance is calculated as follows Ref. 1:

$$Z_{\text{match}} = \sqrt{Z_1 Z_2}\,. \tag{1}$$

The impedance matching itself is a problem that requires a though investigation and is out of scope of this model. Here, the acrylic plastic is used as the matching layer material.

The input signal applied to the transmitter is a harmonic voltage pulse of the amplitude $V_0 = 50$ V, the frequency $f_0 = 2.5$ MHz, and duration of 2 μs. The voltage profile is depicted in Figure 1.

ACOUSTICS: CONVECTED WAVE EQUATION, TIME EXPLICIT

The vibrations of the transmitter generate an ultrasonic acoustic wave traveling from the transmitter to the receiver. The propagation of the acoustic wave is defined by the normal velocity boundary condition prescribed on the interface between the matching layer of the transmitter and the fluid domain. The velocity components are derived from the submodel that simulates the transmitter.

When the acoustic signal reaches the other end of the signal tube, it will interact with the receiver. The acoustic pressure imposed on the interface between the fluid domain and the matching layer of the receiver will be used as the boundary condition in the receiver submodel.

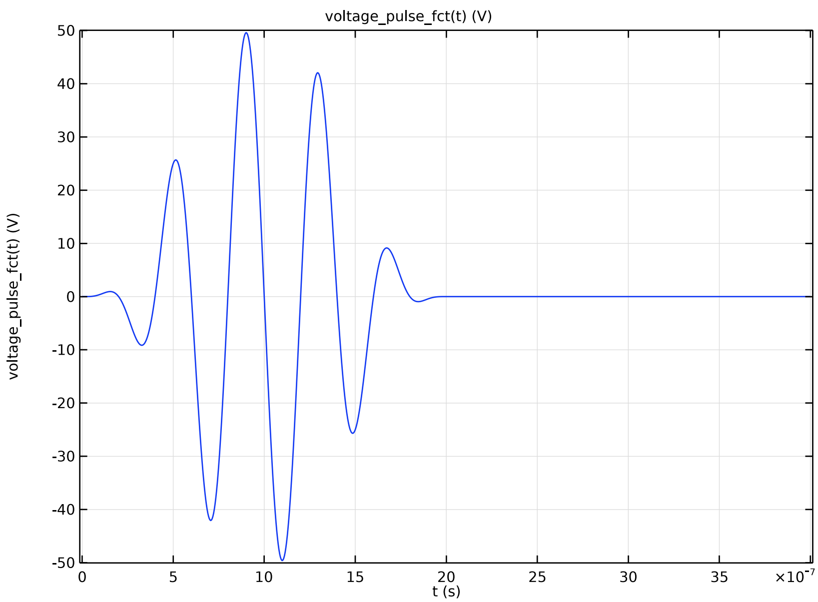


Figure 1: Input voltage applied to the transmitter.

Results and Discussion

The propagation of the ultrasonic acoustic wave near the transmitter at time t = 2.4 μs is shown in Figure 2. The driving voltage applied to the transmitter is that depicted in Figure 1.

The propagation of the ultrasonic pressure wave over the symmetry plane of the flowmeter is depicted in Figure 3. Note that the upper left plot presents the full profile of the signal shown in Figure 2.

By the time t = 10 μs the pressure signal has reached the receiver. Figure 4 shows the mechanical deformation profile of the receiver. In Figure 5 you can see the profiles of the input voltage applied to the transducer and the output electric signal read on the receiver. As previously mentioned, the time of flight will be different for the signal propagating downstream and the one propagating upstream in the presence of a background flow.

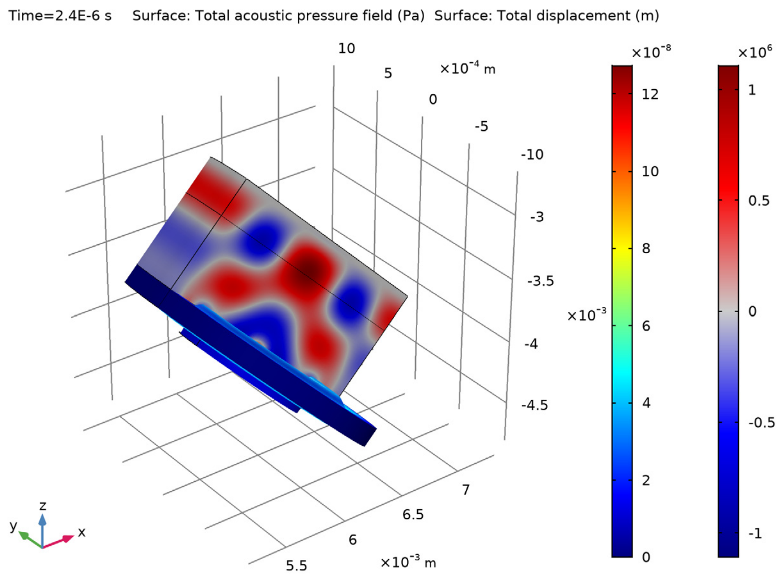


Figure 2: Propagation of the ultrasonic acoustic wave from the transmitter.

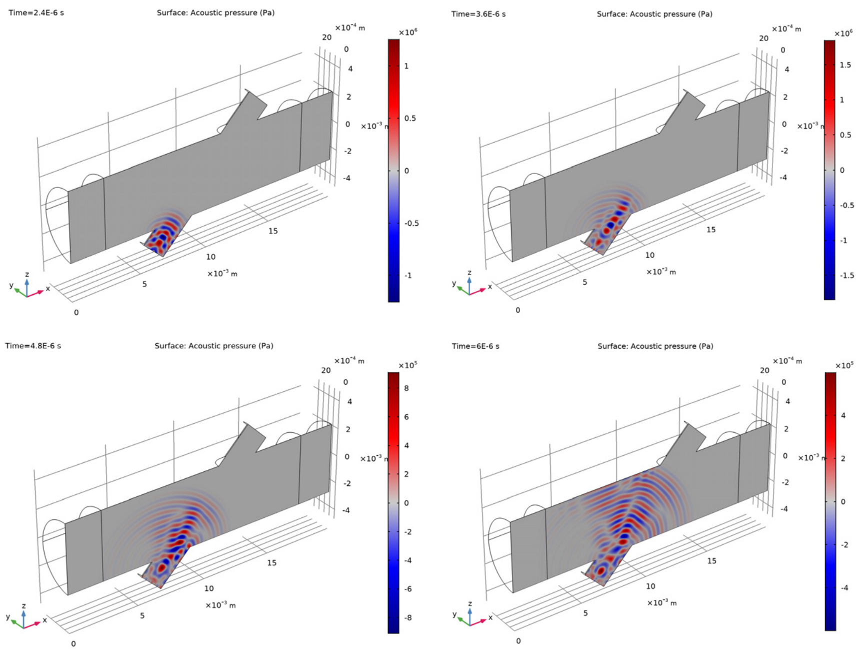


Figure 3: Propagation of the acoustic pressure signal at 4 time steps.

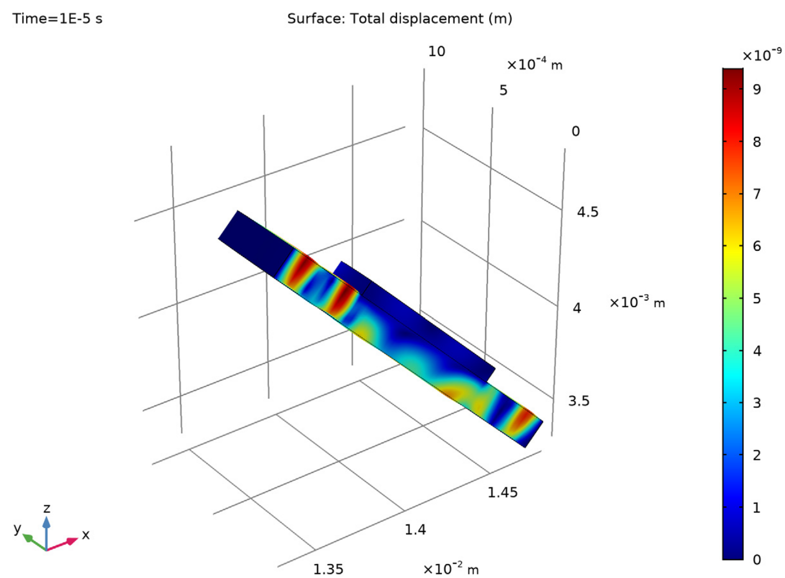


Figure 4: Mechanical deformation of the receiver.

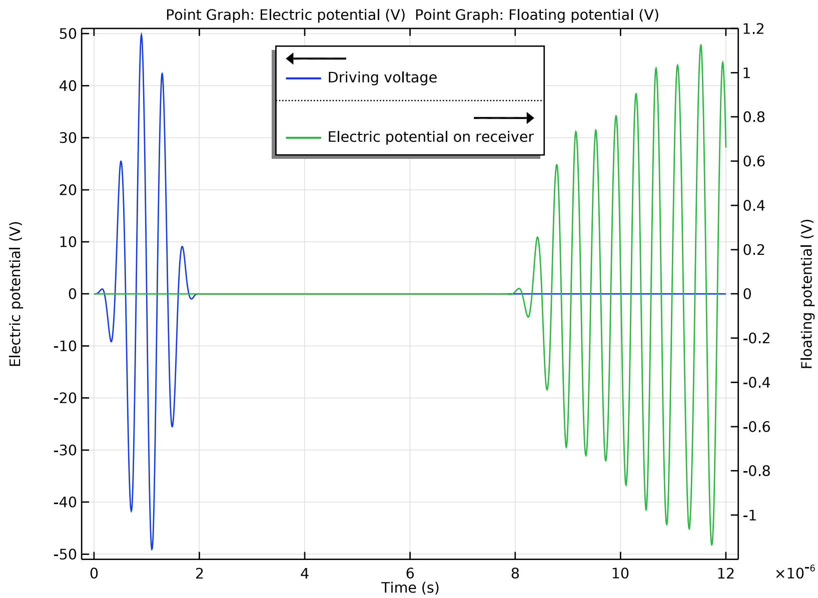


Figure 5: Input and output electric signals as functions of time.

Notes About the COMSOL Implementation

The described split of the entire model into three submodels has a certain reason. The parts that model the piezoelectric transducers use the conventional FEM, whereas the ultrasonic wave propagation is modeled by the DG method. The time-dependent solvers for both approaches are different, which makes the direct FEM to DG or DG to FEM couplings impossible.

The workaround discussed in this tutorial is based on the indirect one-way coupling between the models that use different numerical schemes. This is done through the identity mapping coupling operators `idmap1` and `idmap2`. These are used on the transmitter-fluid and the fluid-receiver interfaces, respectively. Thus the operators map the results between the submodels and provide the required couplings.

The following points of the submodels' setups must be taken into account:

- The transmitter submodel includes the domains for the piezoelectric transducer, the matching layer, and the fluid layer. The last is located on top of the matching layer and is used to account the attached fluid mass providing the fluid-structure interaction. The

Pressure Acoustics, Transient interface solves for the pressure wave in the fluid. The fluid domain is truncated by a perfectly matched layer. Note that this approach remains appropriate even in the case of a background flow, because the considered fluid domain is occupies only a small part of the sensing tube adjacent to the transmitter. Therefore, the interaction between the fluid attached to the matching layer and the background flow in the main pipe is negligible.

- In order to account for the interaction between the ultrasonic pressure wave and the receiver, an impedance boundary condition is imposed on the fluid-receiver interface in a Convected Wave Equation, Time Explicit interface.

Reference

1. V. A. Gavrilova, M. G. Fazlyyyakhmatov, and N. F. Kashapov, "Protective matching polymer powder coating of piezoelectric element," *J. Phys.: Conf. Ser.*, vol. 479, issue 1, 2013.

Application Library path: `Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers`

Modeling Instructions

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

NEW

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

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 `flow_meter_piezoelectric_transducers_parameters.txt`.

   Specify the driving voltage applied to the transmitter.

Rectangle 1 (rect1)

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

2 In the **Settings** window for **Rectangle**, type `rect_fct` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Lower limit** text field, type `0.5e-6`.

4 In the **Upper limit** text field, type `1.5e-6`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1e-6`.

Analytic 1 (an1)

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

2 In the **Settings** window for **Analytic**, type `voltage_pulse_fct` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `V0*sin(omega0*t)*rect_fct(t)`.

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

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

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

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

Argument	Lower limit	Upper limit
t	0	10*T0

8 Click **Plot**.

The input electric signal should look like the one in Figure 1.

The entire model is split into submodels. First, create a model that simulates the propagation of the acoustic signal induced by the input signal applied to the transmitter.

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 1

The geometry will contain the domains corresponding to the piezoelectric transducer, the matching layer, and a layer of the fluid in the signal tube. The last is used to account for the attached fluid mass and model the fluid-structure interaction.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type D_transducer/2.

4 In the **Height** text field, type lam0+L_pml.

5 Locate the **Position** section. In the **x** text field, type L/2.

6 In the **z** text field, type -L_transducer/2.

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

Layer name	Thickness (m)
Layer 1	L_pml

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

9 Select the **Layers on top** check box.

Cylinder 2 (cyl2)

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

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

3 In the **Radius** text field, type D_transducer/2.

4 In the **Height** text field, type L_matching.

5 Locate the **Position** section. In the **x** text field, type L/2.

6 In the **z** text field, type -L_transducer/2-L_matching.

Cylinder 3 (cyl3)

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

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

3 In the **Radius** text field, type D_transducer/4.

4 In the **Height** text field, type L_piezo.

5 Locate the **Position** section. In the **x** text field, type L/2.

6 In the **z** text field, type -L_transducer/2-L_matching-L_piezo.

Rotate 1 (rot1)

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

2 In the **Settings** window for **Rotate**, locate the **Rotation** section.

3 In the **Angle** text field, type alpha.

4 Locate the **Point on Axis of Rotation** section. In the **x** text field, type L/2.

5 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.

6 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Work Plane 1 (wp1)

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** list, choose **zx-plane**.

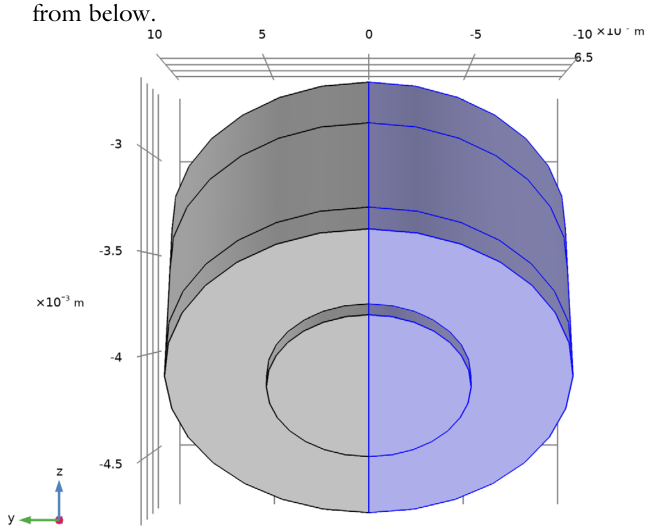
Partition Objects 1 (par1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
2. Click in the **Graphics** window and then press Ctrl+A to select all objects.
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. In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
2. In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. On the object **par1(1)**, select Domains 1 and 3 only.
5. On the object **par1(2)**, select Domain 1 only.
6. On the object **par1(3)**, select Domain 1 only.

7 Click the **Go to YZ View** button in the **Graphics** toolbar three times to view the geometry from below.



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

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

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

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>Water, liquid**.

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

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

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

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

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

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

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Lead Zirconate Titanate (PZT-5H) (mat2)**.

2 Select Domain 3 only.

   It might be easier to select the correct domain by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

Acrylic plastic (mat3)

1 In the **Model Builder** window, click **Acrylic plastic (mat3)**.

2 Select Domain 1 only.

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>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

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

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

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

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

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

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

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

2 Select Domains 2 and 4 only.

SOLID MECHANICS (SOLID)

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

2 Select Domains 1 and 3 only.

ELECTROSTATICS (ES)

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

2 Select Domain 3 only.

ADD STUDY

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

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

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

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

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

DEFINITIONS

Explicit 1

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 **Boundary**.

4 Select Boundary 6 only.

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

Explicit 2

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

2 Select Domains 1 and 3 only.

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

The fluid domain is truncated by introducing a perfectly matched layer (PML) which absorbs the propagating acoustic waves in the fluid.

Perfectly Matched Layer 1 (pml1)

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

2 Select Domain 4 only.

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

4 From the **Coordinate stretching type** list, choose **Rational**.

Define a coordinate system that corresponds to the piezoelectric material orientation: the z-axis of the piezoelectric crystal points along the signal tube axis.

Base Vector System 2 (sys2)

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

2 In the **Settings** window for **Base Vector System**, type `sys_PZT` in the **Label** text field.

3 Locate the **Settings** section. Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

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

	x	y	z
x1	cos(pi/4)	0	-sin(pi/4)
x3	sin(pi/4)	0	cos(pi/4)

Create a mapping operator that will be used to map the results between the submodels.

Identity Mapping 1 (idmap1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Identity Mapping**.

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

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

4 Select Boundary 6 only.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

Modify the **Typical Wave Speed for Perfectly Matched Layers** and **Transient Solver Settings** according to the fluid material and the frequency of the input signal. These settings are used to adjust the PML and the time-dependent solver settings.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

2 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Typical Wave Speed for Perfectly Matched Layers** section.

3 In the c_{ref} text field, type `c0`.

4 Locate the **Transient Solver Settings** section. In the **Maximum frequency to resolve** field enter `f0`.

Symmetry 1

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

2 Select Boundaries 5 and 12 only.

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>Piezoelectric Material**.

2 Select Domain 3 only.

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

4 From the **Coordinate system** list, choose **sys_PZT (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 **Damping type** list, choose **Rayleigh damping**.

4 In the α_{dM} text field, type `alpha_dmp`.

5 In the β_{dK} text field, type `beta_dmp`.

Fixed Constraint 1

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

2 Select Boundaries 2 and 15 only.

Symmetry 1

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

2 Select Boundaries 1 and 8 only.

Roller 1

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

2 Select Boundary 9 only.

ELECTROSTATICS (ES)

Charge Conservation, Piezoelectric 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose **Charge Conservation, Piezoelectric**.

2 Select Domain 3 only.

Ground 1

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

2 Select Boundary 9 only.

Electric Potential 1

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

2 Select Boundary 10 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `voltage_pulse_fct(t)`.

COMPONENT 1 (COMP1)

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

ADD MULTIPHYSICS

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

2 In the tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Transient**.

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

4 In the **Home** toolbar, click **Add Multiphysics**.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 3 and 10 only.

Size 1

1 Right-click **Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lam0/6`.

6 Click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 Select Domain 3 only.

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 `2`.

Swept 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 Select Domain 1 only.

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.

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

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

Swept 3

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 Select Domain 2 only.

Size 1

1 Right-click **Swept 3** 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lam0/6`.

Swept 4

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 4** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `6`.

4 Click **Build All**.

COMPONENT 1 (COMP1)

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

ADD COMPONENT

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

GEOMETRY 2

Create the geometry that contains the main pipe, the signal tube, and the receiver. As the transmitter, the receiver contains a piezoelectric transducer and a matched layer.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `D/2`.

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

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

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

Layer name	Thickness (m)
Layer 1	0.5*D

7 Clear the **Layers on side** check box.

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

9 Select the **Layers on top** check box.

Cylinder 2 (cyl2)

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

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

3 In the **Radius** text field, type `D_transducer/2`.

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `-L_transducer/2`.

Cylinder 3 (cyl3)

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

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

3 In the **Radius** text field, type `D_transducer/2`.

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `L_transducer/2`.

Cylinder 4 (cyl4)

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

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

3 In the **Radius** text field, type `D_transducer/4`.

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `L_transducer/2+L_matching`.

Rotate 1 (rot1)

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

2 Select the objects **cyl2**, **cyl3**, and **cyl4** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `alpha`.

5 From the **Axis type** list, choose **y-axis**.

6 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `L/2`.

Work Plane 1 (wp1)

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** list, choose **xz-plane**.

Partition Objects 1 (par1)

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

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

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 In the **Model Builder** window, right-click **Geometry 2** and choose **Delete Entities**.

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

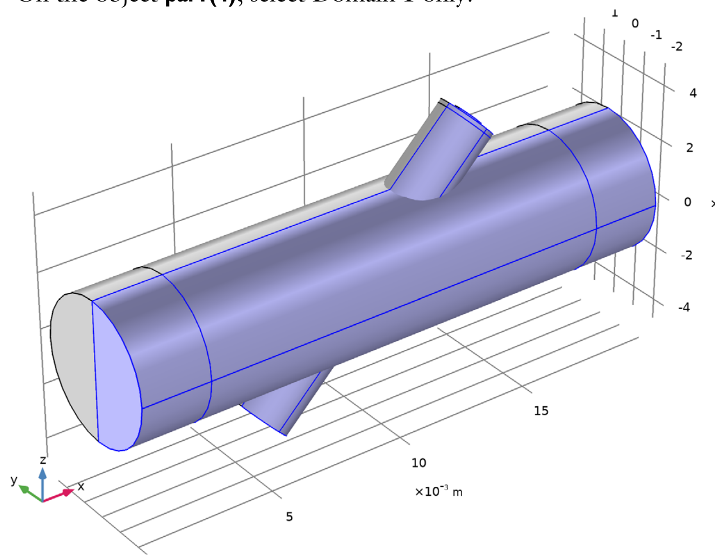
3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **par1(1)**, select Domains 1, 3, and 5 only.

5 On the object **par1(2)**, select Domain 1 only.

6 On the object **par1(3)**, select Domain 1 only.

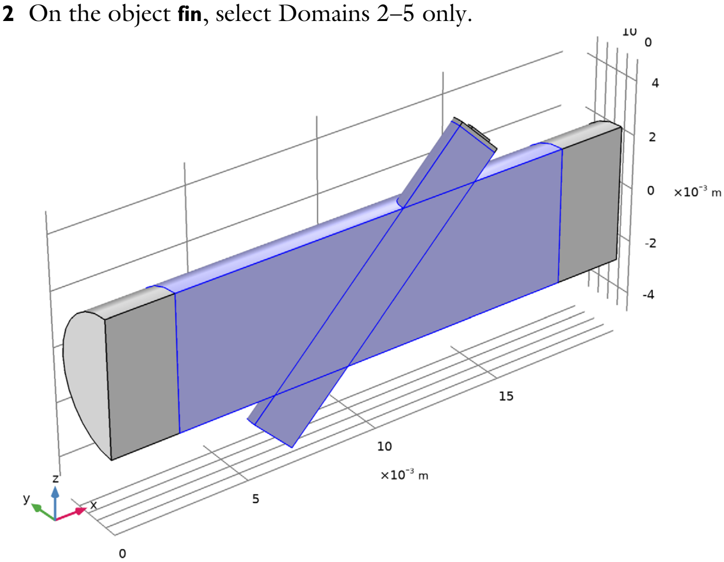
7 On the object **par1(4)**, select Domain 1 only.



Form Composite Domains 1 (cmd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 On the object **fin**, select Domains 2–5 only.



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

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

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>Water, liquid**.

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

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

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

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

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

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

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat5)

1 In the **Model Builder** window, under **Component 2 (comp2)>Materials** click **Lead Zirconate Titanate (PZT-5H) (mat5)**.

2 Select Domain 4 only.

Acrylic plastic (mat6)

1 In the **Model Builder** window, click **Acrylic plastic (mat6)**.

2 Select Domain 3 only.

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>Ultrasound>Convected Wave Equation, Time Explicit (cwe)**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	

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

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

7 In the table, enter the following settings:

Studies	Solve
Study 1	

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

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

10 In the table, enter the following settings:

Studies	Solve
Study 1	

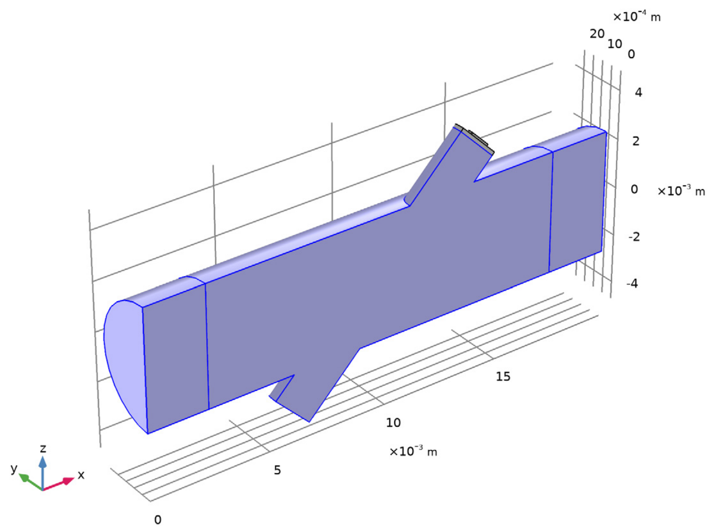
11 Click **Add to Component 2** in the window toolbar.

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

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Convected Wave Equation, Time Explicit (cwe)**.

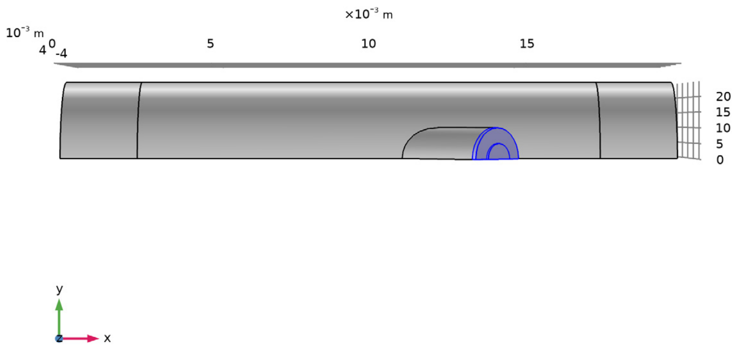
2 Select Domains 1, 2, and 5 only.



SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Solid Mechanics 2 (solid2)**.

2 Select Domains 3 and 4 only.



ELECTROSTATICS 2 (ES2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Electrostatics 2 (es2)**.

2 Select Domain 4 only.

ADD STUDY

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

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

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

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Pressure Acoustics, Transient (actd)	
Solid Mechanics (solid)	
Electrostatics (es)	
Solid Mechanics 2 (solid2)	
Electrostatics 2 (es2)	

5 Find the **Multiphysics couplings in study** subsection. In the table, enter the following settings:

Multiphysics couplings	Solve
Acoustic-Structure Boundary 1 (asb1)	
Piezoelectric Effect 1 (pze1)	

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

7 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Pressure Acoustics, Transient (actd)	
Solid Mechanics (solid)	
Electrostatics (es)	
Convected Wave Equation, Time Explicit (cwe)	

8 Find the **Multiphysics couplings in study** subsection. In the table, enter the following settings:

Multiphysics couplings	Solve
Acoustic-Structure Boundary 1 (asb1)	
Piezoelectric Effect 1 (pze1)	

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

10 From the **Home** menu, choose **Add Study**.

DEFINITIONS (COMP2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Definitions**.

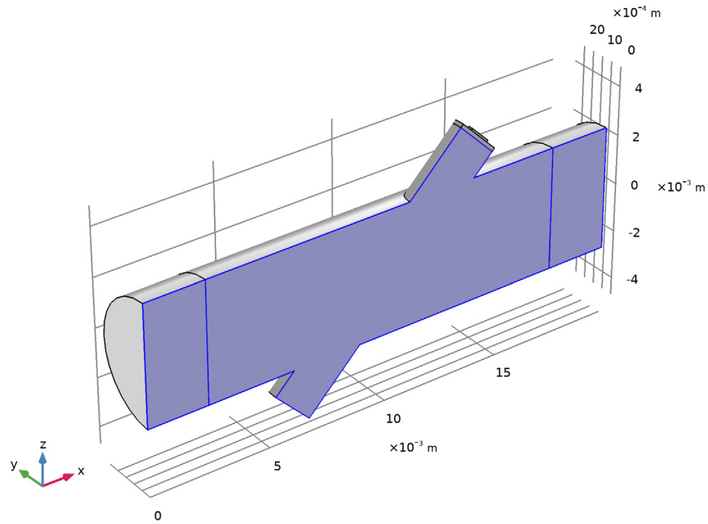
Explicit 3

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 **Boundary**.

4 Select Boundaries 2, 6, and 25 only.



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

6 In the **Model Builder** window, under **Component 2 (comp2)** click **Definitions**.

Explicit 4

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 **Boundary**.

4 Select Boundary 10 only.

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

6 In the **Model Builder** window, click **Definitions**.

Explicit 5

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 **Boundary**.

4 Select Boundary 16 only.

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

6 In the **Model Builder** window, click **Definitions**.

Explicit 6

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 **Boundary**.

4 Select Boundaries 3, 4, 7–9, 11–13, 26, and 27 only.

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

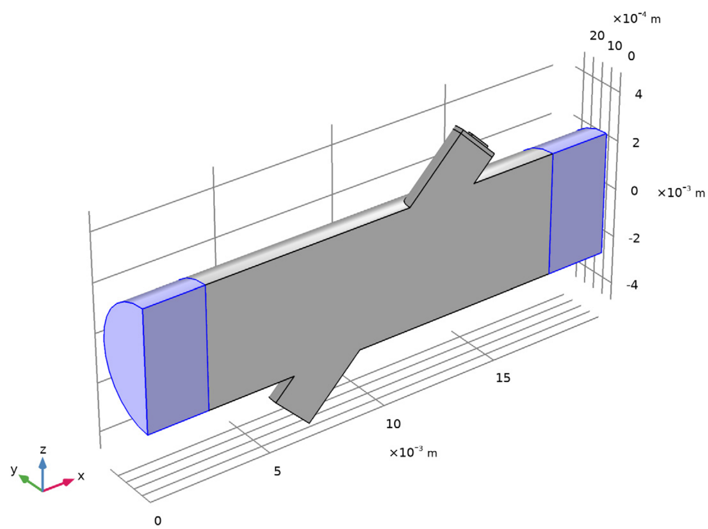
6 In the **Model Builder** window, click **Definitions**.

Absorbing Layer 1 (ab1)

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

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

2 Select Domains 1 and 5 only.



3 In the **Model Builder** window, click **Definitions**.

Identity Mapping 2 (idmap2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Identity Mapping**.

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

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

4 Select Boundary 16 only.

Again, create a coordinate system specifying the piezoelectric material orientation.

5 In the **Model Builder** window, click **Definitions**.

Base Vector System 4 (sys4)

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

2 In the **Settings** window for **Base Vector System**, type `sys_PZT` in the **Label** text field.

3 Locate the **Settings** section. Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

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

	x	y	z
x1	cos(pi/4)	0	-sin(pi/4)
x3	sin(pi/4)	0	cos(pi/4)

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

Symmetry 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Convected Wave Equation, Time Explicit (cwe)** and choose **Symmetry**.

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

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

Acoustic Impedance 1

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

2 Select Boundaries 1 and 28 only.

Acoustic Impedance 2

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

On the receiver-fluid interface, specify the characteristic acoustic impedance of the matching layer.

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

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

4 Locate the **Acoustic Impedance** section. In the Z text field, type `Z_matching`.

Normal Velocity 1

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

On the transmitter-fluid interface, specify the velocity obtained from the transmitter submodel.

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

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

4 Locate the **Normal Velocity** section. From the **Type** list, choose **Velocity**.

5 Specify the $\mathbf{v}_b(t)$ vector as

`comp1.idmap1(withsol('sol1', ut, setval(t, t)))`	x
`comp1.idmap1(withsol('sol1', vt, setval(t, t)))`	y
`comp1.idmap1(withsol('sol1', wt, setval(t, t)))`	z

SOLID MECHANICS 2 (SOLID2)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Solid Mechanics 2 (solid2)** and choose **Fixed Constraint**.

2 Select Boundaries 14 and 22 only.

Symmetry 1

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

2 Select Boundaries 15 and 19 only.

Roller 1

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

2 Select Boundary 21 only.

Boundary Load 1

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

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

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

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the p text field, type `idmap2(withsol('sol2', p2, setval(t, t)))`.

Piezoelectric Material 1

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

2 Select Domain 4 only.

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

4 From the **Coordinate system** list, choose **sys_PZT (sys4)**.

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 **Damping type** list, choose **Rayleigh damping**.

4 In the α_{dM} text field, type `alpha_dmp`.

5 In the β_{dK} text field, type `beta_dmp`.

ELECTROSTATICS 2 (ES2)

Ground 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Electrostatics 2 (es2)** and choose **Ground**.

2 Select Boundary 21 only.

Floating Potential 1

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

2 Select Boundary 20 only.

3 In the **Settings** window for **Floating Potential**, locate the **Floating Potential** section.

4 Select the **Floating potential group** check box.

Charge Conservation, Piezoelectric 1

1 In the **Physics** toolbar, click **Domains** and choose **Charge Conservation, Piezoelectric**.

2 Select Domain 4 only.

MULTIPHYSICS

Piezoelectric Effect 2 (pze2)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Piezoelectric Effect**.

MESH 2

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **Free Tetrahedral**.

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

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

4 Select Domains 1, 2, and 5 only.

When modeling using physics interfaces that are based on the DG method, it is important to avoid small mesh elements, since they control the time steps taken by the solver. In order to avoid this use the **Element Quality Optimization** functionality available for the tetrahedral mesh. This step is very important.

5 Click to expand the **Element Quality Optimization** section. Select the **Avoid too small elements** check box.

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

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

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

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lam0/1.5`.

5 In the **Minimum element size** text field, type `lam0/3`.

6 In the **Model Builder** window, click **Mesh 2**.

7 Click **Build All**.

MESH 3

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

Free Triangular 1

1 Right-click **Mesh 3** and choose **More Operations>Free Triangular**.

2 Select Boundaries 17 and 20 only.

Size 1

1 Right-click **Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lam0/6`.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 3** and choose **Swept**.

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

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

4 Select Domain 4 only.

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 `2`.

Swept 2

1 In the **Model Builder** window, right-click **Mesh 3** and choose **Swept**.
2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 3 only.

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `4`.
4 In the **Model Builder** window, click **Mesh 3**.
5 Click **Build All**.

COMPONENT 2 (COMP2)

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

STUDY 1

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, locate the **Study Settings** section.
3 Clear the **Generate default plots** check box.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, type `Transmitter` in the **Label** text field.
3 Locate the **Study Settings** section. In the **Times** text field, type `range(0,T0/5,30*T0)`.
4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Multiphysics couplings	Solve for
Piezoelectric Effect 2 (pze2)	

5 Click to expand the **Mesh Selection** section.

STUDY 2

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

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

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

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, type `Propagation of Signal` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Times** text field, type `range(0,T0/5,30*T0)`.

4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Multiphysics couplings	Solve for
Piezoelectric Effect 2 (pze2)	

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

6 Under **Selections**, click **Add**.

Store the results on the structure-fluid interfaces and the symmetry plane only.

7 In the **Add** dialog box, in the **Selections** list, choose **Symmetry**, **Source**, and **Receiver**.

8 Click **OK**.

STUDY 3

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

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

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

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, type `Receiver` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Times** text field, type `range(0,T0/5,30*T0)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-4`.

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

Geometry	Mesh
Geometry 1	No mesh
Geometry 2	Mesh 3

Since this multiphysics problem involves a transient acoustics interface, the **Transient Solver Settings** defined for the acoustics interface will be automatically used when solving the coupled problem. This ensures optimal solver configuration for this piezoacoustic problem solved in the time domain.

STUDY 1

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

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

STUDY 2

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

2 Click **Compute**.

The default solver settings generated for Study 3 will be different from those in Study 1. This is because the Pressure Acoustics, Transient interface is absent here. It is required to change the default solver settings to **Manual** and specify the time step explicitly.

STUDY 3

In the **Model Builder** window, click **Study 3**.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `1/(60*f0)`.

This expression is the same as the one generated in Study 1.

6 In the **Model Builder** window, click **Study 3**.

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

RESULTS

Datasets

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

Mirror 3D 1

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

Create a plot that shows the propagating acoustic signal induced by the driving voltage applied to the transmitter. The result should look as in Figure 2.

3D Plot Group 1

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Transmitter` in the **Label** text field.
3 Locate the **Data** section. From the **Time (s)** list, choose **2.4E-6**.

Surface 1

1 Right-click **Transmitter** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 From the **Color table** list, choose **Wave**.
4 Select the **Symmetrize color range** check box.

Surface 2

1 In the **Model Builder** window, right-click **Transmitter** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 1**.
4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>solid.disp - Total displacement - m**.
5 Locate the **Data** section. From the **Time (s)** list, choose **2.4E-6**.

Deformation 1

1 Right-click **Surface 2** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** check box.
4 In the associated text field, type `1e3`.

Transmitter

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

2 In the **Model Builder** window, click **Transmitter**.

3 In the **Transmitter** toolbar, click **Plot**.

Next, create a plot showing the propagation of the acoustic signal through the main pipe. For selected times, the acoustic signal profile can be seen in Figure 3.

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

3D Plot Group 2

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (4) (sol2)**.

4 From the **Time (s)** list, choose **2.4E-6**.

5 In the **Label** text field, type `Signal Propagation`.

Surface 1

1 Right-click **Signal Propagation** and choose **Surface**.

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

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

4 Select the **Symmetrize color range** check box.

5 Click to expand the **Quality** section. Modify the **Resolution** and increase the **Element refinement** to represent the solution properly. This is because the shape functions used for the convected wave equation interface are 4th order per default.

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

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

Selection 1

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

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

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

4 In the **Signal Propagation** toolbar, click **Plot**.

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

Plot the mechanical deformation of the receiver caused by the incoming acoustic signal. The deformation is depicted in Figure 4.

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

3D Plot Group 6

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Receiver` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 3/Solution 3 (6) (sol3)**.
4 From the **Time (s)** list, choose **1E-5**.

Surface 1

1 Right-click **Receiver** and choose **Surface**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2>Solid Mechanics 2> Displacement>solid2.disp - Total displacement - m**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** check box.
4 In the associated text field, type `1e3`.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.
6 In the **Receiver** toolbar, click **Plot**.

   The last plot will show the input voltage applied to the transmitter and the output voltage signal read on the receiver (see Figure 5).

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

1D Plot Group 7

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Sent and Received Signals` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **None**.

Point Graph 1

1 Right-click **Sent and Received Signals** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 1/Solution 1 (1) (sol1)**.
4 Select Point 4 only.
5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electrostatics>Electric>V - Electric potential - V**.

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

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

8 In the table, enter the following settings:

Legends
Driving voltage

9 Locate the **Data** section. From the **Time selection** list, choose **Interpolated**.

10 Click **Range**.

11 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

12 In the **Start** text field, type 0.

13 In the **Stop** text field, type 30*T0.

14 In the **Number of values** text field, type 2001.

15 Click **Replace**.

Point Graph 2

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

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

3 From the **Dataset** list, choose **Study 3/Solution 3 (6) (sol3)**.

4 Select Point 17 only.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 2>Electrostatics 2>Floating potentials>es2.fp1.V - Floating potential - V**.

6 Locate the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
Electric potential on receiver

9 Locate the **Data** section. From the **Time selection** list, choose **Interpolated**.

10 Click **Range**.

11 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

12 In the **Start** text field, type 0.

13 In the **Stop** text field, type 30*T0.

14 In the **Number of values** text field, type 2001.

15 Click **Replace**.

Sent and Received Signals

1 In the **Model Builder** window, click **Sent and Received Signals**.

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

3 Select the **Two y-axes** check box.

4 In the table, enter the following settings:

Plot	Plot on secondary y-axis
Point Graph 2	√

5 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

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

7 In the **y minimum** text field, type -51.

8 In the **y maximum** text field, type 51.

9 In the **Secondary y minimum** text field, type -1.2.

10 In the **Secondary y maximum** text field, type 1.2.

11 In the **Sent and Received Signals** toolbar, click **Plot**.