Created in COMSOL Multiphysics 6.0

Ultrasonic Flowmeter with Piezoelectric Transducers

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 shows how to simulate an ultrasonic flowmeter with piezoelectric transducers in the simplified no-flow case. The simulatoin approach is based on the discontinuous Galerkin (dG) method which is well suited for acoustically large transient problems. The model is a true multiphyiscs problem that involves acoustic-structure interaction and piezoelectric effect. The former is modeled with the Elastic Waves, Time Explicit and the Pressure Acoustics, Time Explicit physics interfaces coupled through the Pair Acoustic-Structure Boundary, Time Explicit multiphysics feature. The latter is handled 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 flowmeter 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 $\alpha = 45°$. The dimensions of the pipes used in this tutorial are the same as the ones given in the model Ultrasound Flowmeter with Generic Time-of-Flight Configuration. The pipe walls are considered rigid. There are two transducers placed at either end of the signal pipe. They operate as a transmitter and a receiver. Both transducers are identical and consist of a piezoelectric unit, a matching layer, and a damping block. 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*. This model solves the full three-dimensional problem with the symmetry condition imposed on the sagittal plane of the structure.

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 (here, PZT-5H) that is used for the conversion between the electric and the mechanical waves. Its thickness is taken to be 1/2 of the wavelength.

A direct propagation of the acoustic wave from the piezoelectric material to 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 $Z_{\mathrm{water}} \approx 1.5$ MRayl, and for PZT-5H, $Z_{\mathrm{PZT}} \approx 34.5$ MRayl. Therefore, a 1/4 wavelength thick matching layer is required to minimize the losses. Its impedance should be close to the geometric mean of those of the piezoelectric and the fluid materials, i.e.

$$Z_{\mathrm{match}} = \sqrt{Z_{\mathrm{water}} Z_{\mathrm{PZT}}} \approx 7.2 \text{ MRayl.}$$

The piezoelectric element is surrounded by a backing layer block (also called damping block) at the back. The damping block absorbs the waves radiated from the back face of the piezoelectric element. The properties of the used matching and damping materials are shown in Table 1 .

TABLE 1: MATCHING AND DAMPING MATERIAL PROPERTIES

Part	**Material**	**Density, kg/m3**	**Longitudinal wave speed, m/s**	**Shear wave speed, m/s**
Matching layer	Alumina/Epoxy	2280	3400	1920
Damping block	Tungsten/Epoxy	6580	1500	775

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.

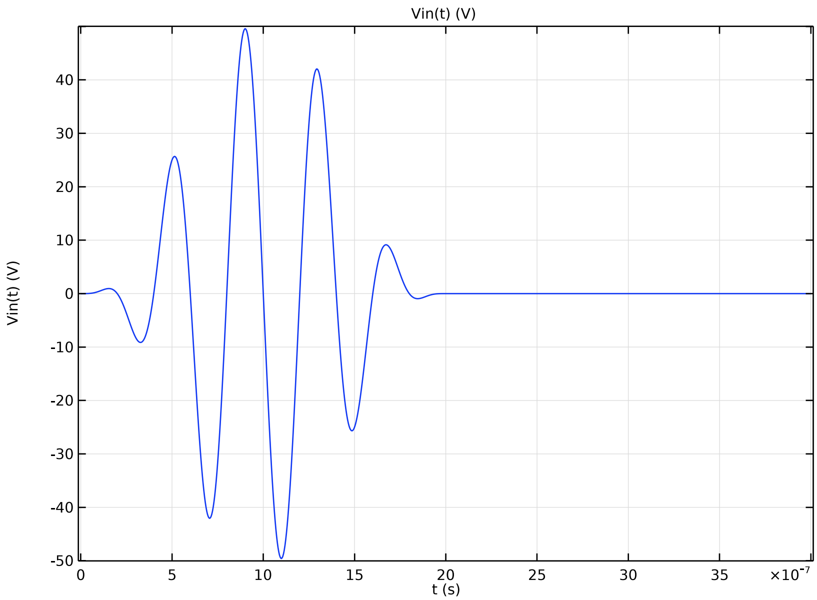


Figure 1: Input voltage applied to the transmitter.

Results and Discussion

The propagation of the pressure wave in solid and fluid domains is shown in Figure 2. The signal emitted by the transmitter propagates into the fluid at t = 4 µs (upper-left corner). The signal reaches the upper wall of the main pipe at t = 6 µs (upper-right corner) and propagates further in the signal pipe to the receiver at t = 8 µs (lower-left corner). The signal reaches the end of the signal tube and generates an elastic wave in the receiver at t = 10 µs (lower-right corner).

The elastic wave in the receiver piezoelectric element is converted into an electric signal. In Figure 3 you can see the profiles of the input voltage applied to the transducer and the output electric signal read on the receiver.

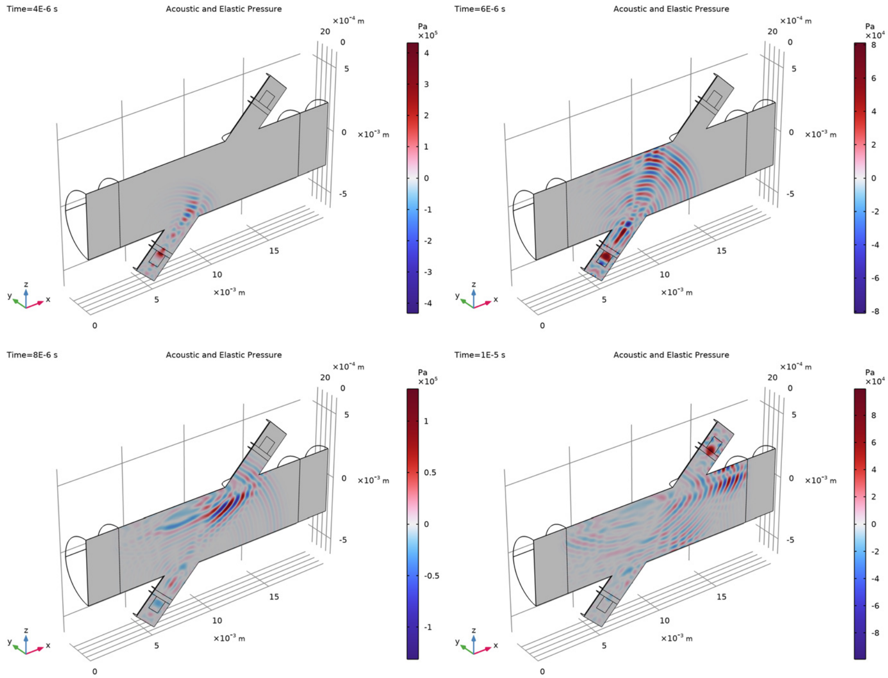


Figure 2: Propagation of the acoustic pressure signal at 4 time steps.

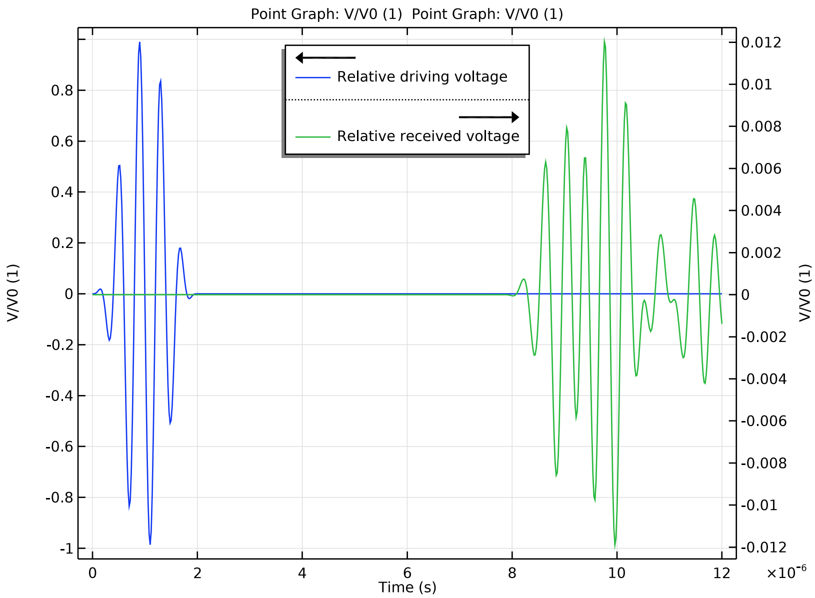


Figure 3: Input and output electric signals as functions of time.

Notes About the COMSOL Implementation

GEOMETRY AND MESH

The model geometry is an assembly and therefore parts of the geometry are 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 non-conformal. For wave propagation problems, feasible results are achieved when the mesh resolves the wavelengths of the propagating waves. In the solid domains, the minimal wavelength is given by the shear wave speed. Thus materials with lower speed of sound require finer mesh than those with higher speed of sound (compare the shear wave speeds for the matching and the damping material given in Table 1). The use of a non-conformal mesh in this tutorial makes it possible to reduce the number of DOFs solved for in the model.

Application Library path: `Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 Click **Done**.

Load the parameters that define the geometry and the physical properties of the system.

GLOBAL DEFINITIONS

Geometry Parameters

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

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

3 Click **Load from File**.

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

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

Model Parameters

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

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

3 Click **Load from File**.

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

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

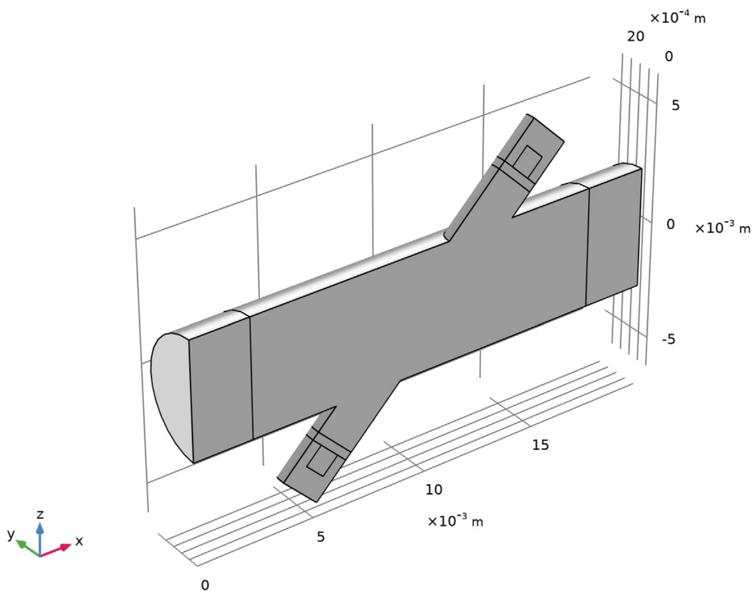
Import the model geometry sequence from the geometry file. The instructions to the geometry are found in the appendix at the end of this document.

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 `flow_meter_piezoelectric_transducers_geom_sequence.mph`.

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



Specify the driving voltage signal applied to the transmitter.

GLOBAL DEFINITIONS

Rectangle 1 (rect1)

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

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

3 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`.

6 Click **Plot**.

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `V0*sin(2*pi*f0*t)*rect1(t)`.

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

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

6 In the table, enter the following settings:

Argument	Unit
t	s

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

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

8 Click **Plot**.

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

Create selections to simplify the model setup.

DEFINITIONS

Water

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

2 Select Domains 1–3 only.

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

PZT

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

2 Select Domains 5 and 9 only.

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

Matching

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

2 Select Domains 6 and 7 only.

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

Backing

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

2 Select Domains 4 and 8 only.

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

Symmetry

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, 16, 22, 30, 33, 40, 46, and 55 only.

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

Solid

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

2 In the **Settings** window for **Union**, type `Solid` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **PZT**, **Matching**, and **Backing**.

5 Click **OK**.

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

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 In the table, enter the following settings:

	x	**y**	**z**
x1	`cos(alpha)`	`0`	`-sin(alpha)`
x3	`sin(alpha)`	`0`	`cos(alpha)`

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

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

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.

COMPONENT 1 (COMP1)

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

ADD PHYSICS

1. Go to the **Add Physics** window.
2. In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Time Explicit (pate)**.
3. Click **Add to Selection** in the window toolbar.

PRESSURE ACOUSTICS, TIME EXPLICIT (PATE)

1. In the **Settings** window for **Pressure Acoustics, Time Explicit**, locate the **Domain Selection** section.
2. From the **Selection** list, choose **Water**.

ADD PHYSICS

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

ELASTIC WAVES, TIME EXPLICIT (ELTE)

1. In the **Settings** window for **Elastic Waves, Time Explicit**, locate the **Domain Selection** section.
2. From the **Selection** list, choose **Solid**.

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 **PZT**.

PRESSURE ACOUSTICS, TIME EXPLICIT (PATE)

Impedance 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Pressure Acoustics, Time Explicit (pate)** and choose **Impedance**.

2 Select Boundaries 1 and 19 only.

Symmetry 1

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

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

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

ELASTIC WAVES, TIME EXPLICIT (ELTE)

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

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 **PZT**.

Low-Reflecting Boundary 1

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

2 Select Boundaries 21 and 52 only.

Symmetry 1

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

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

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

Elastic Waves, Time Explicit Model 1

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

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**.

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 `0.01`.

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

8 In the ζ_2 text field, type `0.01`.

Elastic Waves, Time Explicit Model 1

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

Damping 2

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 **Backing**.

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 `0.025`.

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

8 In the ζ_2 text field, type `0.025`.

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 `0.005`.

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

7 In the ζ_2 text field, type `0.005`.

ELECTROSTATICS (ES)

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

Charge Conservation, Piezoelectric 1

1 In the **Physics** toolbar, click **Domains** 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 **PZT**.

Ground 1

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

2 Select Boundaries 29 and 57 only.

Electric Potential 1

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

2 Select Boundary 32 only.

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

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

Floating Potential 1

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

2 Select Boundary 54 only.

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

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

Symmetry Plane 1

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

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

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

Add multiphysics coupling features.

MULTIPHYSICS

Piezoelectric Effect, Time Explicit 1 (pzete1)

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

Pair Acoustic-Structure Boundary, Time Explicit 1 (aspte1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Pair Acoustic-Structure Boundary, Time Explicit**.

2 In the **Settings** window for **Pair Acoustic-Structure Boundary, Time Explicit**, locate the **Pair Selection** section.

3 Under **Pairs**, click + **Add**.

4 In the **Add** dialog box, in the **Pairs** list, choose **Identity Boundary Pair 1 (ap1)** and **Identity Boundary Pair 2 (ap2)**.

5 Click **OK**.

Now, set up the materials.

MATERIALS

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in>Water, liquid**.

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

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

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

MATERIALS

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

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

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

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**.

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

5 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

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

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 **Backing**.

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

5 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

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

DEFINITIONS

Absorbing Layer 1 (ab1)

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

2 Select Domains 1 and 3 only.

Create a mesh. The mesh should be fine enough to resolve the shortest wavelength in each material. Note that the mesh element nodes on one side of the boundary pairs do not match with those on the other side. As a result, the mesh elements adjacent to the pairs have different size, which helps to reduce the number of DOFs in the model.

MESH 1

Free Triangular 1

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

2 Select Boundaries 32, 35, 37, 42, 43, and 54 only.

Size 1

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

2 Select Boundaries 32 and 54 only.

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

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `cs_pzt/f0/2`.

Size 2

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

2 Select Boundaries 35, 37, 42, and 43 only.

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

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `cs_match/f0/2`.

Swept 1

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

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

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

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

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 `4`.

Swept 2

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

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

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

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

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 `2`.

Free Tetrahedral 1

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

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. Use the **Element Quality Optimization** functionality available for the tetrahedral mesh to avoid this. This step is very important.

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

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

4 Clear the **Avoid too large elements** check box.

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

Size 1

1 Right-click **Free Tetrahedral 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 **Water**.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `lam0/1.5`.

8 Select the **Minimum element size** check box.

9 In the associated text field, type `lam0/3`.

Size 2

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

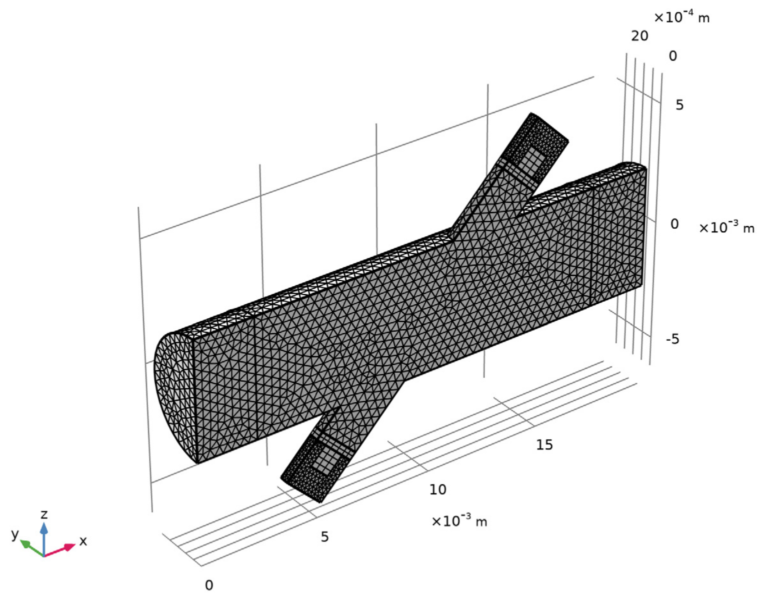
4 From the **Selection** list, choose **Backing**.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `cs_damp/f0/1.5`.

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



ROOT

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

ADD STUDY

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

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

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

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)`.

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

4 Under **Selections**, click **Add**.

Store the results on the symmetry plane only.

5 In the **Add** dialog box, select **Symmetry** in the **Selections** list.

6 Click **OK**.

Step 1: Time Dependent

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

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

Plot the pressure in the fluid and solid domains and inspect the propagating acoustic signal at different times to get the results like the ones in Figure 2.

RESULTS

Acoustic and Elastic Pressure

1 In the **Settings** window for **3D Plot Group**, type `Acoustic and Elastic Pressure` in the **Label** text field.

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

Selection 1

1 In the **Model Builder** window, expand the **Acoustic and Elastic Pressure** node.

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

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

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

Surface 2

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Elastic Waves, Time Explicit>Stress>elte.p - Pressure - Pa**.

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

4 In the **Acoustic and Elastic Pressure** toolbar, click **Plot**.

Plot the driving voltage applied to the transmitter and the voltage signal read on the receiver. The result should look like the one in Figure 3.

Sent and Received Signals

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 **Time selection** list, choose **Interpolated**.

4 In the **Times (s)** text field, type `range(0, T0/20, 30*T0)`.

Point Graph 1

1 Right-click **Sent and Received Signals** and choose **Point Graph**.

2 Select Point 39 only.

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

4 In the **Expression** text field, type `V/V0`.

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

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

7 In the table, enter the following settings:

Legends
Relative driving voltage

Point Graph 2

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

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

3 Click **Clear Selection**.

4 Select Point 73 only.

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

Legends
Relative received voltage

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, select the **Plot on secondary y-axis** check box for **Point Graph 2**.

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

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

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 Click **Done**.

GLOBAL DEFINITIONS

Geometry Parameters

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

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

3 Click **Load from File**.

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

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

Model Parameters

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

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

3 Click **Load from File**.

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

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

GEOMETRY 1

Cylinder 1 (cyl1)

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

2 In the **Settings** window for **Cylinder**, locate the **Axis** section.

3 From the **Axis type** list, choose **x-axis**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `D/2`.

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

6 Click to expand 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`.

Cylinder 5 (cyl5)

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 `2*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`.

Difference 1 (dif1)

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

2 Select the object **cyl5** only.

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

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

5 Select the object **cyl4** only.

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

Copy 1 (copy1)

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

2 Select the objects **cyl3**, **cyl4**, and **dif1** only.

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

4 In the **z** text field, type `-L_transducer`.

Mirror 1 (mir1)

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

2 Select the objects **copy1(1)**, **copy1(2)**, and **copy1(3)** only.

3 In the **Settings** window for **Mirror**, locate the **Point on Plane of Reflection** section.

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

Rotate 1 (rot1)

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

2 Select the objects **cyl2**, **cyl3**, **cyl4**, **dif1**, **mir1(1)**, **mir1(2)**, and **mir1(3)** only.

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

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

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

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 the **Select Box** button in the **Graphics** toolbar.

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

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

5 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, 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.

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

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

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

11 On the object **par1(8)**, select Domain 1 only.

Union 1 (uni1)

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

2 Select the objects **del1(1)** and **del1(2)** only.

Delete Entities 2 (del2)

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

2 On the object **uni1**, select Boundaries 13, 14, 16, and 19 only.

Form Union (fin)

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

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

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

Make sure to select the **Create imprints** check box to create an assembly with still matching surface pairs. This will eliminate the use of fallback and thus increase the performance.

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

5 Click **Build Selected**.

Ignore Edges 1 (ige1)

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

2 On the object **fin**, select Edges 19, 20, 23, and 27 only.

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