Created in COMSOL Multiphysics 6.0

High-Intensity Focused Ultrasound (HIFU) Propagation Through a Tissue Phantom

Introduction

The high-intensity focused ultrasound is used in many different biomedical applications, such as thermal ablation of tumors, transcranial HIFU surgery, shock wave lithotripsy, etc. A HIFU signal is focused on a small focal zone, where its intensity reaches higher levels. In this case, nonlinear effects may become significant and one speaks of the propagation of finite amplitude sound waves. The nonlinear nature of the phenomenon results in the energy transfer from lower to higher harmonics. The contribution of the higher-order harmonics grows with the propagation distance.

This example shows how to model nonlinear propagation of HIFU through a dissipative media using the *Nonlinear Pressure Acoustics, Time Explicit* physics interface. The interfaces solves the system of nonlinear acoustic equations in the form of a hyperbolic conservation law, see Ref. 1, using the discontinuous Galerkin finite element method (dG-FEM) and an explicit time integration scheme. The used approach is suitable when the cumulative nonlinear effects, that is, those cumulated through propagation, dominate the local nonlinear effects.

In this model, the emitted signal is a tone burst pulse that occupies only a limited part of the computational domain on its way. Adaptive mesh refinement is used for automatic remeshing following the signal propagation. This ensures that the mesh is fine enough to resolve the higher-order harmonics where it is required.

Model Definition

The focusing of an ultrasonic signal is typically achieved either by using a phase delay or a focusing lens on the transducer side. In this model, a spherically focused ultrasound transducer with a concave lens is used to emit the signal. The transducer housing and the lens are assumed to be rigid. The model setup is shown in Figure 1. The model geometry is axially symmetric.

The spherical lens of radius r and aperture a emits a signal that is focused at the focal point $\mathbf{F}$ located in the tissue phantom. The signal is a five-cycle tone burst with the amplitude P_0 = 0.1 MPa and the center frequency f_0 = 1 MHz as shown in Figure 2. The signal amplitude at the source location is enough th generate of higher-order harmonics, but not high enough for the formation of shocks, and therefore no special shock-capturing techniques are required in this model.

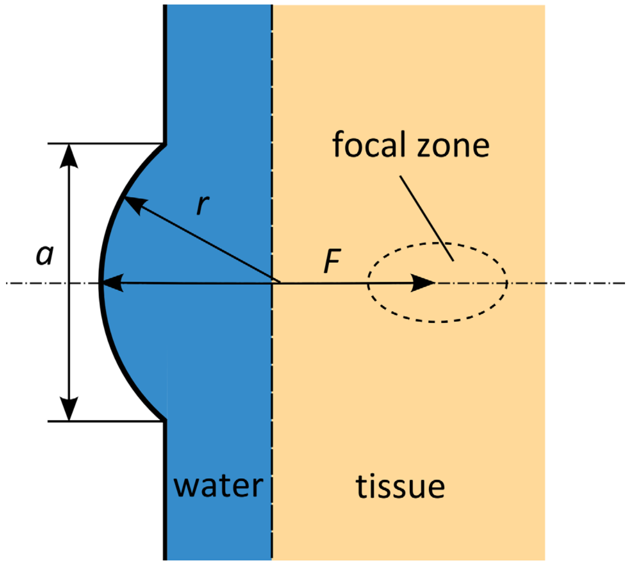


Figure 1: Model geometry.

The traveling time of the signal to the focal point can be computed as

$$t_{\mathrm{F}} = \frac{d_{\mathrm{water}}}{c_{\mathrm{water}}} + \frac{d_{\mathrm{tissue}}}{c_{\mathrm{tissue}}},$$

where c and d are the speed of sound and the traveling distance in the corresponding materials, respectively. At the frequency of 1 MHz, the time to focus is approximately equal to $40/f_0$.

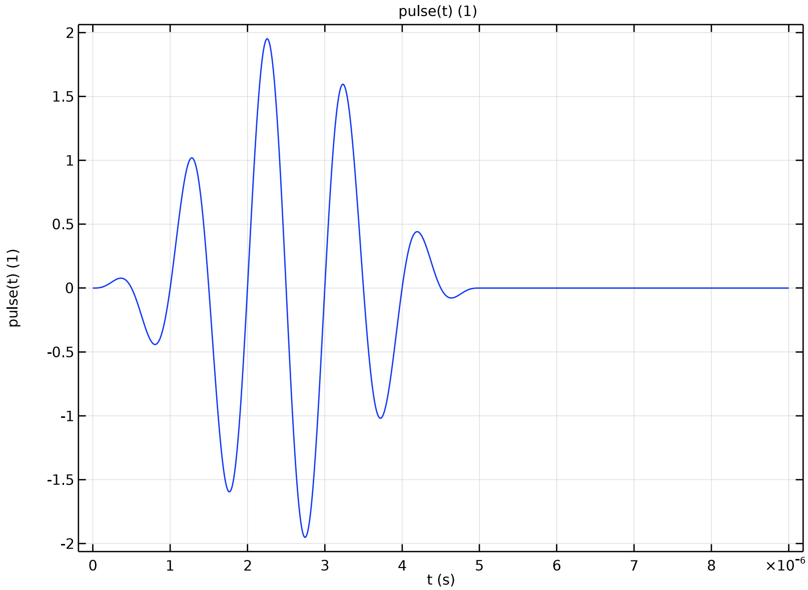


Figure 2: Source signal.

Results and Discussion

The evolution of the signal traveling from the source is illustrated in Figure 3. The tone burst pulse has left the source at t = 10 µs and travels towards the boundary between the water and the tissue phantom domains, which it passes at t = 20 µs with some part being reflected back to the source. The focusing of the signal becomes visible at t = 30 µs and reaches its maximum at t = 40 µs.

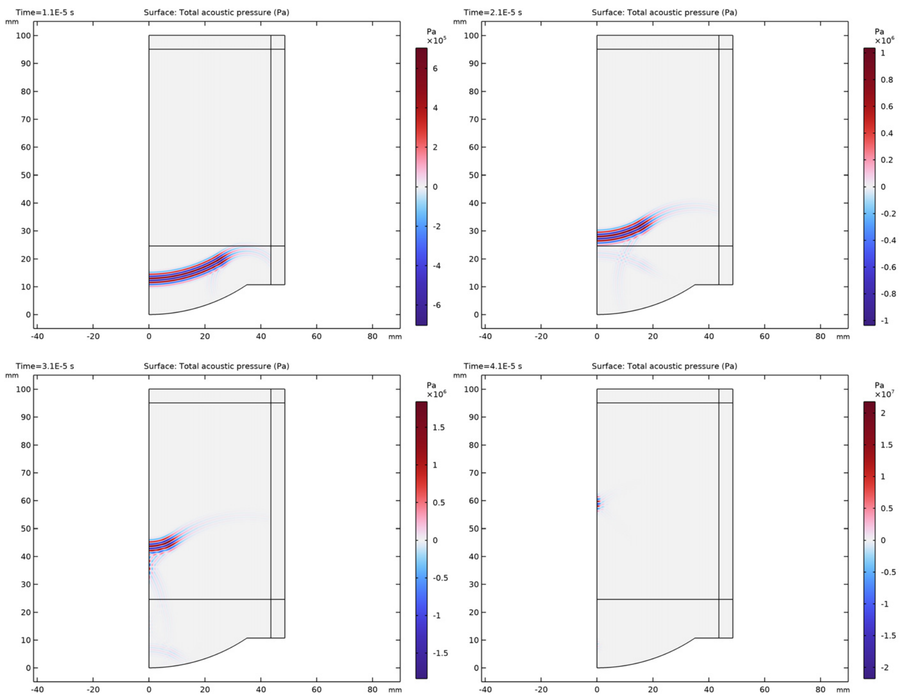


Figure 3: Propagation of the ultrasonic signal at times t = 10, 20, 30, and 40 µs.

Figure 3 also shows that the signal strength increases as it comes closer to the focal zone. This is confirmed by the plots of the signal at the water/tissue interface and at the focal point depicted in Figure 4. The acoustic pressure amplitude at the focal point is about 10 times greater than that at the water/tissue interface. Another observation is that the positive pressure peaks are almost twice as high as the negative ones, at the focal point. This indicated that the signal becomes highly nonlinear as it reaches the focal zone.

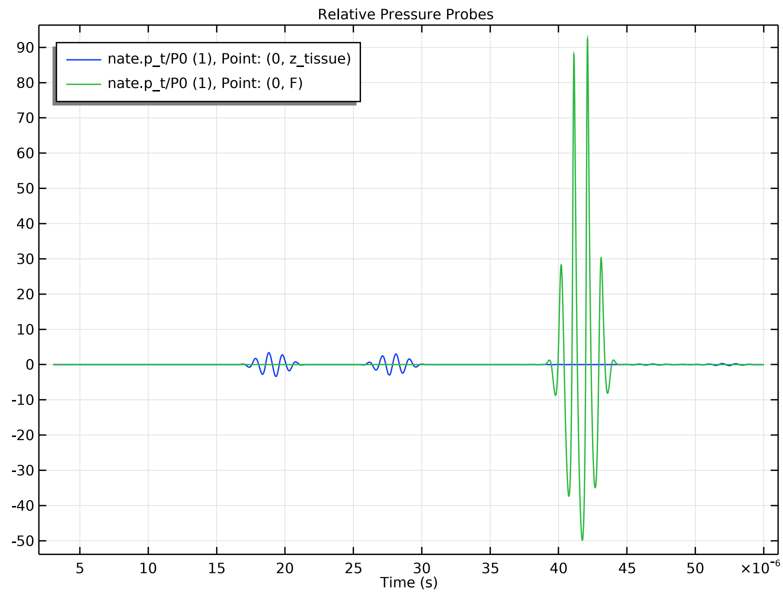


Figure 4: Acoustic pressure at the water/tissue interface and at the focal point.

Figure 5 shows the frequency content of the recorded signals. The frequency content of the signal entering the tissue phantom is not much different from the source signal. That is, the propagation up to this point is linear. The signal at the focal point is however highly nonlinear, which is seen from the number of generated higher-order harmonics.

The plots of the minimum and maximum on-axis pressure calculated are depicted in Figure 6. The values are computed according to the following formulas

$$p^{-} = \min_{t \in [0, T_{\text{end}}]}(p) \qquad p^{+} = \max_{t \in [0, T_{\text{end}}]}(p) \tag{1}$$

and are normalized with respect to the maximum pressure at the focal point. Figure 6 shows that p^+ is almost twice as high as p^-, which is a distinctive feature of nonlinear propagation of a HIFU signal. The bounds of the focal zone are also seen in Figure 6.

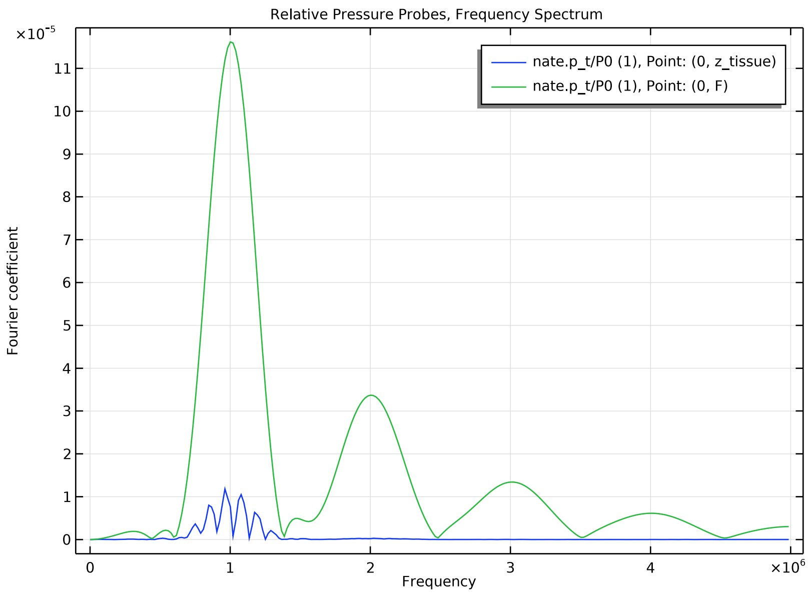


Figure 5: Frequency spectrum of the acoustic pressure at the water/tissue interface and at the focal point.

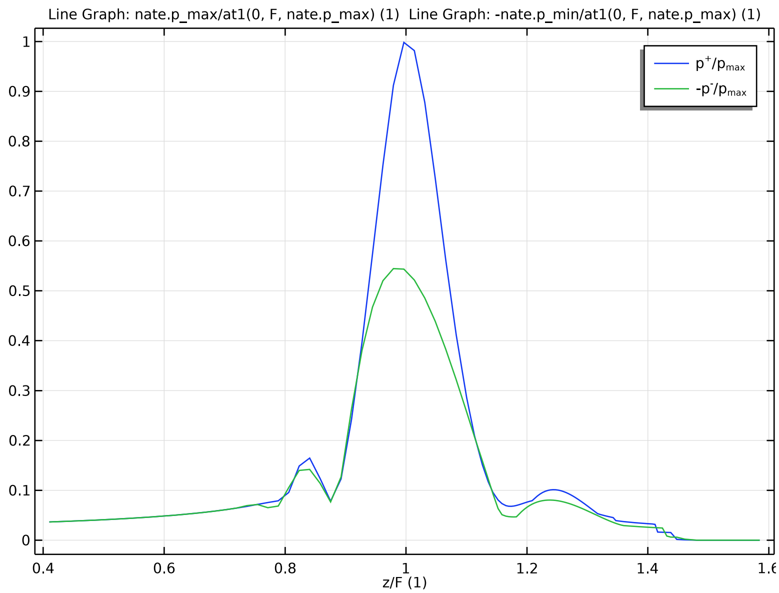


Figure 6: Frequency spectrum of the acoustic pressure at the water/tissue interface and at the focal point.

Notes About the COMSOL Implementation

MESH AND TIME EXPLICIT SOLVER

While solving a wave propagation problem, the used mesh typically has to be fine enough to resolve the frequency content of the signal, that is, the specified number of the higher-order harmonics, N, as discussed in the tutorial *Nonlinear Propagation of a Cylindrical Wave — Verification Model*. However, a distinctive feature of the model is that the source signal is a pulse and therefore the propagating signal is finite in space. This suggests that a fine mesh is only required in a finite part of the computational domain where the signal is located at a certain time, thus saving a lot of DOFs. This is achievable by enabling the **Adaptive mesh refinement** option which can be found in the **Adaptation** section of the Time Dependent study step.

The initial mesh built in the model resolves the fundamental harmonic. With the default quartic discretization order, the mesh element size should not exceed 1.5 of the fundamental wavelength. The adaptive mesh refinement ensures that a finer mesh is created to resolve the propagating signal. On the **Adaptive Mesh Refinement** node, select

General modification for the Adaptation method and clear the **Allow coarsening** check box. Thus the resulting mesh will always resolve the fundamental harmonic. Then make the **Error indicator** capture sharp gradients of the acoustic pressure by typing `sqrt(comp1.pr^2+comp1.pz^2)`.

The *Nonlinear Pressure Acoustic, Time Explicit* physics is based on dG-FEM and uses an explicit time integration schemes. The solver time step obeys the CFL condition to ensure the stability of the time integration method. That is, $\Delta t \leq h_{min}/c_{max}(p)$, where h_{min} is the minimum mesh element size and $c_{max}(p)$ is the maximum wave propagation speed. In the nonlinear case the speed of sound depends on the acoustic pressure. In this model, the mesh follows the nonlinear propagation of the signal and therefore will have smaller elements around the its sharp fronts and larger elements away from it. It is worth selecting the time-stepping method to **Adam-Bashforth 3 (local)** and enabling the **Update time levels** option available in the **General** section of the time-explicit solver. Figure 7 shows the values of the cell wave time scale `elte.wtc` variable, that the time step is proportional to. The smaller values are found for the smaller mesh elements in the vicinity of the signal, while the larger values are used in the rest of the computational domain.

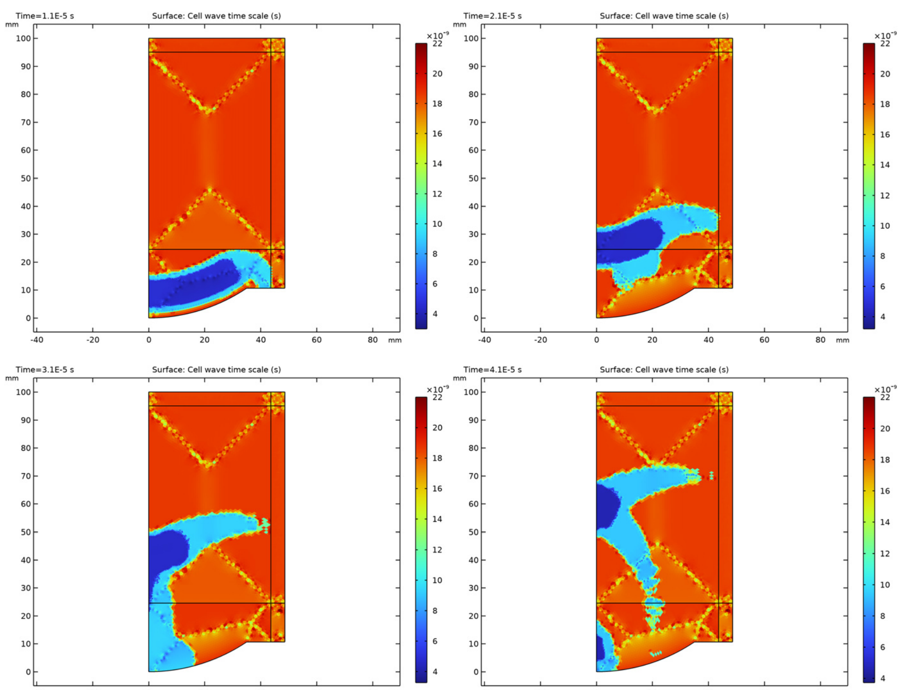


Figure 7: Cell wave time scale at times t = 10, 20, 30, and 40 µ*s.*

References

1. M.A. Diaz, M.A. Solovchuk, and T.W.H. Sheu, “A conservative numerical scheme for modeling nonlinear acoustic propagation in thermoviscous homogeneous media”, *J. Comp. Phys.*, vol. 363, 2018.

Application Library path: `Acoustics_Module/Ultrasound/hifu_tissue_sample`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Acoustics>Ultrasound>Nonlinear Pressure Acoustics, Time Explicit (nate)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 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 `hifu_tissue_sample_parameters.txt`.

Rectangle 1 (rect1)

1 In the **Home** toolbar, click f(x) **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`.

4 In the **Upper limit** text field, type `5*T0`.

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

Create a five-cycle tone burst source signal and inspect its frequency content.

Pulse

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

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

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

4 Locate the **Definition** section. In the **Expression** text field, type `sin(omega0*t)*(1 - cos(omega0*t/5))*rect1(t)`.

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

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

Argument	Unit
t	s

7 In the **Function** text field, type `1`.

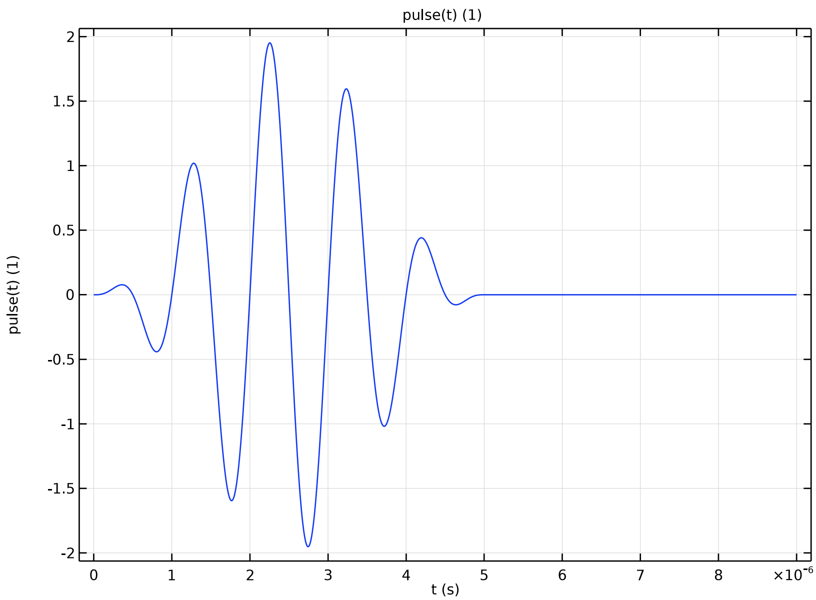
8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
t	0	9*T0	s

9 Click **Create Plot**.

RESULTS

Tone Burst Pulse



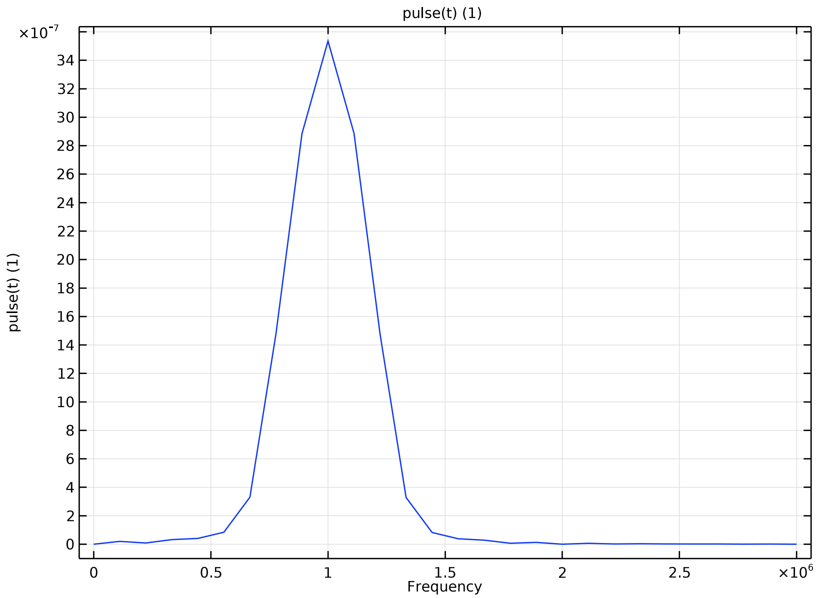
10 In the **Settings** window for **1D Plot Group**, type `Tone Burst Pulse` in the **Label** text field.

Function 1

1. In the **Model Builder** window, expand the **Tone Burst Pulse** node, then click **Function 1**.
2. In the **Settings** window for **Function**, locate the **Output** section.
3. From the **Display** list, choose **Discrete Fourier transform**.
4. From the **Show** list, choose **Frequency spectrum**.
5. From the **Scale** list, choose **Multiply by sampling period**.
6. Select the **Frequency range** check box.
7. In the **Maximum** text field, type `3*f0`.

8 In the **Tone Burst Pulse** toolbar, click **Plot**.

The plot shows that the frequency content of the signal lies close to the center frequency f_0.



GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Circle 1 (c1)

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

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

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

4 Locate the **Position** section. In the **z** text field, type `r_source`.

5 Click **Build Selected**.

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_source`.

4 In the **Height** text field, type `r_source-sqrt(r_source^2-w_source^2)`.

5 Click **Build Selected**.

Intersection 1 (int1)

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

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

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

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 `r_model+th_abs`.

4 In the **Height** text field, type `h_model+th_abs-z_tissue`.

5 Locate the **Position** section. In the **z** text field, type `z_tissue`.

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

Layer name	Thickness (mm)
Layer 1	th_abs

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

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

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

10 Click **Build Selected**.

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

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 `r_model+th_abs`.

4 In the **Height** text field, type `z_tissue-(r_source-sqrt(r_source^2-w_source^2))`.

5 Locate the **Position** section. In the **z** text field, type `r_source-sqrt(r_source^2-w_source^2)`.

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

Layer name	Thickness (mm)
`Layer 1`	`th_abs`

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

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

9 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 Boundary 3 only.

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

Create two domain probe points on the symmetry axis—one at the water/tissue interface and the other at the focal point—which will record the signal during its propagation from the source.

DEFINITIONS

Domain Point Probe 1

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

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

3 In row **Coordinates**, set **z** to `z_tissue`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

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

3 In the **Expression** text field, type `nate.p_t/P0`.

Domain Point Probe 2

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

2 In the **Model Builder** window, click **Domain Point Probe 2**.

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

4 In row **Coordinates**, set **z** to `F`.

Point Probe Expression 2 (ppb2)

1 In the **Model Builder** window, click **Point Probe Expression 2 (ppb2)**.

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

3 In the **Expression** text field, type `nate.p_t/P0`.

Now, set up 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 3–6 only.

3 In the **Settings** window for **Absorbing Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

NONLINEAR PRESSURE ACOUSTICS, TIME EXPLICIT (NATE)

Nonlinear Pressure Acoustics, Time Explicit Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Nonlinear Pressure Acoustics, Time Explicit (nate)** click **Nonlinear Pressure Acoustics, Time Explicit Model 1**.

2 In the **Settings** window for **Nonlinear Pressure Acoustics, Time Explicit Model**, locate the **Pressure Acoustics Model** section.

3 From the **Fluid model** list, choose **General dissipation**.

Impose the **Material Discontinuity** boundary condition upon the interface between the water and the tissue parts of the computational domain.

Material Discontinuity 1

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

2 Select Boundaries 3 and 11 only.

Pressure 1

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

2 Select Boundary 18 only.

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

4 In the $p_0(t)$ text field, type `P0*pulse(t)`.

Impedance 1

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

2 Select Boundaries 6 and 14–17 only.

It is also of interest to compute the minimum and maximum acoustic pressure on the symmetry axis over the simulated time interval.

Nonlinear Pressure Acoustics, Time Explicit Model 1

In the **Model Builder** window, click **Nonlinear Pressure Acoustics, Time Explicit Model 1**.

Compute Minimum and Maximum Pressure 1

1 In the **Physics** toolbar, click **Attributes** and choose **Compute Minimum and Maximum Pressure**.

2 Select Boundary 2 only.

Set up the materials. The physics interface and the chosen fluid model will suggest which material properties should be defined.

MATERIALS

Water

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

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

3 Select Domains 1 and 4 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho_water`	kg/m^3	Basic
Speed of sound	c	`c_water`	m/s	Basic
Sound diffusivity	delta_diff	`delta_water`	m^2/s	Attenuation and dissipation model
Parameter of nonlinearity	BA	`BA_water`	1	Nonlinear model

Tissue

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

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

3 Select Domains 2, 3, 5, and 6 only.

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

Property	Variable	Value	Unit	Property group
Density	rho	`rho_tissue`	kg/m^3	Basic
Speed of sound	c	`c_tissue`	m/s	Basic

Property	Variable	Value	Unit	Property group
Sound diffusivity	delta_diff	delta_tissue	m²/s	Attenuation and dissipation model
Parameter of nonlinearity	BA	BA_tissue	1	Nonlinear model

Now, proceed to the mesh. Even though the problem at hand is nonlinear, make the mesh resolve only the fundamental harmonic. To do it, limit the maximum element size by 1.5 of the wavelength in the corresponding materials.

MESH 1

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 Select Domains 1 and 4 only.

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 `c_water/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 Select Domains 2, 3, 5, and 6 only.

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 `c_tissue/f0/1.5`.

8 Click **Build All**.

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type `range(0, T0, 55*T0)`.

This setting saves the solution at times multiple to T_0 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 signals at the probes will be computed for each time step taken by the solver thus providing a much higher temporal resolution of the results.

4 Click to expand the **Adaptation** section. Select the **Adaptive mesh refinement** check box.

This setting enables the adaptation of the mesh with respect to the propagating signal by creating intermediate meshes. Those will be fine enough in the parts of the computational domain where it is required to resolve higher-order harmonics generated because of the nonlinear nature of the problem.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Explicit Solver 1**.

3 In the **Settings** window for **Time-Explicit Solver**, locate the **General** section.

4 From the **Method** list, choose **Adams-Bashforth 3 (local)**.

5 From the **Update time levels** list, choose **Factor**.

The **Adams-Bashforth 3 (local)** time-stepping method uses local time steps chosen in accordance with the size of mesh elements.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Explicit Solver 1** node, then click **Adaptive Mesh Refinement**.

7 In the **Settings** window for **Adaptive Mesh Refinement**, locate the **General** section.

8 Find the **Mesh element control** subsection. From the **Adaptation method** list, choose **General modification**.

9 Clear the **Allow coarsening** check box.

This setting makes sure that the intermediate meshes will still be fine enough to resolve the fundamental harmonic.

10 Locate the **Error Estimation** section. In the **Error indicator** text field, type `sqrt(comp1.pr^2+comp1.pz^2)`.

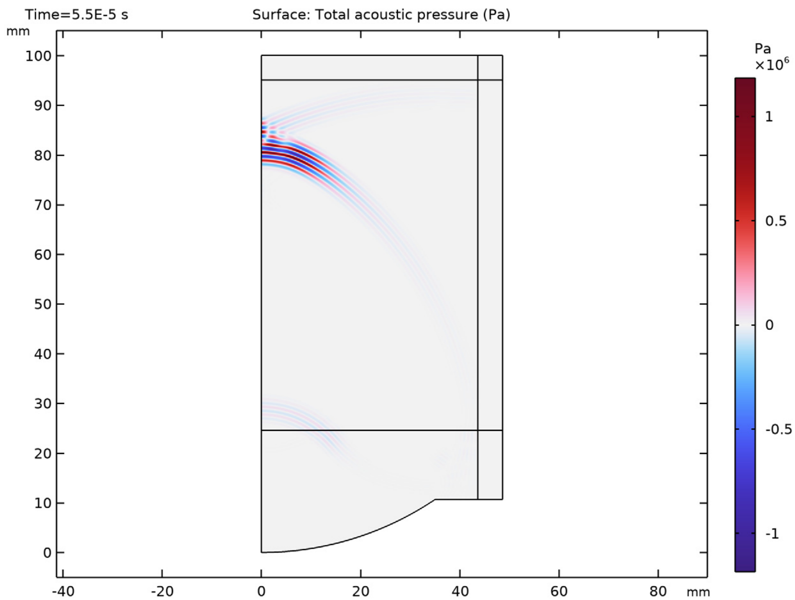
This makes the **Error indicator** trace the sharp gradients of the acoustic pressure.

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

RESULTS

Acoustic Pressure (nate)

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



Relative Pressure Probes

1 In the **Model Builder** window, expand the **Results>Probe Plot Group 4** node, then click **Probe Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Relative Pressure Probes` in the **Label** text field.

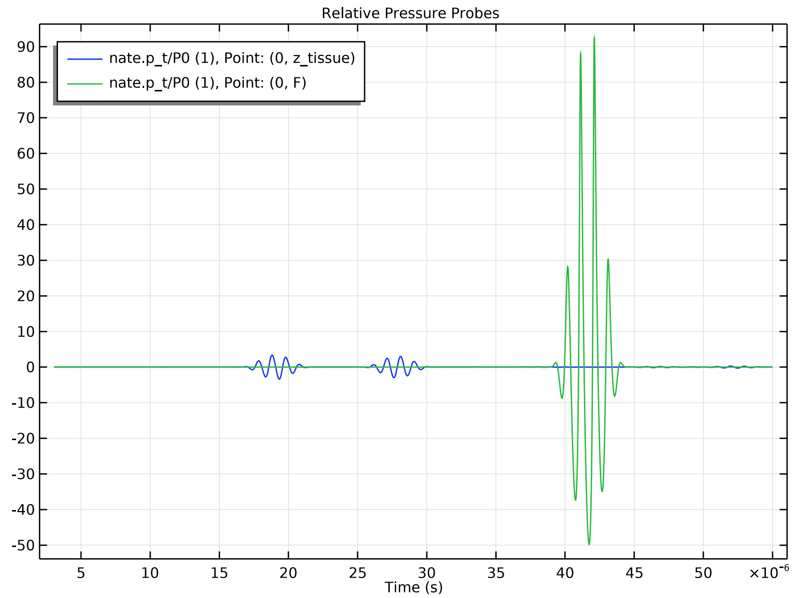
3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Probe Table Graph 1

1 In the **Model Builder** window, click **Probe Table Graph 1**.

2 In the **Relative Pressure Probes** toolbar, click **Plot**.



Relative Pressure Probes, Frequency Spectrum

1 In the **Model Builder** window, right-click **Relative Pressure Probes** and choose **Duplicate**.

2 In the **Model Builder** window, click **Relative Pressure Probes 1**.

3 In the **Settings** window for **1D Plot Group**, type `Relative Pressure Probes, Frequency Spectrum` in the **Label** text field.

4 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Probe Table Graph 1

1 In the **Model Builder** window, click **Probe Table Graph 1**.

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

3 From the **Transformation** list, choose **Discrete Fourier transform**.

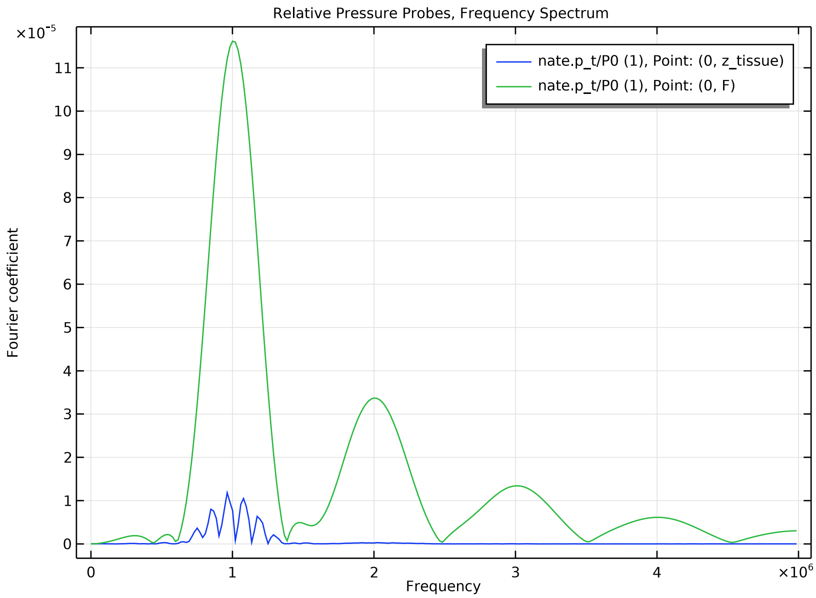
4 From the **Show** list, choose **Frequency spectrum**.

5 From the **Scale** list, choose **Multiply by sampling period**.

6 Select the **Frequency range** check box.

7 In the **Maximum** text field, type `5*f0`.

8 In the **Relative Pressure Probes, Frequency Spectrum** toolbar, click **Plot**.



On-Axis Relative Pressure

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

2 In the **Settings** window for **1D Plot Group**, type `On-Axis Relative Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Refined Mesh Solution 1 (sol2)**.

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

5 In the **Times (s)** list, select **4.1E-5**.

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

Line Graph 1

1 Right-click **On-Axis Relative Pressure** and choose **Line Graph**.

2 Select Boundaries 1 and 2 only.

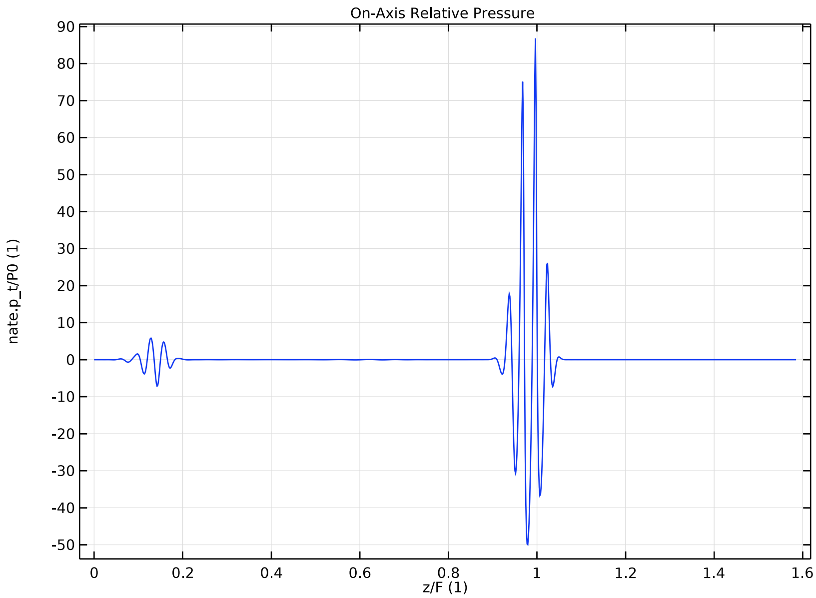
3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `nate.p_t/P0`.

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

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

7 In the **On-Axis Relative Pressure** toolbar, click **Plot**.



Normalized Positive and Negative Pressure

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

2 In the **Settings** window for **1D Plot Group**, type `Normalized Positive and Negative Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Refined Mesh Solution 1 (sol2)**.

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

Line Graph 1

1 Right-click **Normalized Positive and Negative Pressure** and choose **Line Graph**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Nonlinear Pressure Acoustics, Time Explicit>Pressure and sound pressure level> nate.p_max - Maximum acoustic pressure - Pa**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `nate.p_max/at1(0, F, nate.p_max)`.

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

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

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

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

9 In the table, enter the following settings:

Legends
p⁺/p_{max}

10 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

Line Graph 2

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

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Nonlinear Pressure Acoustics, Time Explicit>Pressure and sound pressure level> nate.p_min - Minimum acoustic pressure - Pa**.

3 Locate the **y-Axis Data** section. In the **Expression** text field, type `-nate.p_min/at1(0, F, nate.p_max)`.

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

Legends
-p⁻/p_{max}

5 In the **Normalized Positive and Negative Pressure** toolbar, click **Plot**.

Cell Wave Time Scale

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

Inspect the local cell wave time scale, which the time step is proportional to.

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

2 From the **Dataset** list, choose **Study 1/Refined Mesh Solution 1 (sol2)**.

3 Click **Plot Last**.

4 In the **Label** text field, type `Cell Wave Time Scale`.

Surface 1

1 Right-click **Cell Wave Time Scale** 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 1 (comp1)>**

Nonlinear Pressure Acoustics, Time Explicit>Cell time scale>nate.wtc - Cell wave time scale - s.

3 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

4 From the **Smoothing** list, choose **None**.

5 In the **Cell Wave Time Scale** toolbar, click **Plot**.