Created in COMSOL Multiphysics 5.5

Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers

Introduction

This small tutorial is a standard test and benchmark model for nonreflecting conditions and sponge layers for linearized Euler like systems, see Ref. 1 and Ref. 2. The Convected Wave Equation, Time Explicit interface solves the linearized Euler equations with an adiabatic equation of state and the interface has the Absorbing Layers to model infinite domains.

An acoustic pulse is generated by an initial Gaussian distribution at the center of the computational domain. The pulse propagates in a high Mach number uniform flow. An analytical solution exists to the problem and is used to validate the solution. The model shows how to set up and use the absorbing layers.

Model Definition

The computational domain with absorbing layers is depicted in Figure 1. The model is set up in a dimensionless system where the speed of sound $c_0 = 1$ and the density $\rho_0 = 1$. The Gaussian pulse is emitted at the origin $\mathbf{x} = \mathbf{0}$ with initial values

$$\begin{bmatrix} p \\ u \\ v \end{bmatrix} = \begin{bmatrix} 1 \\ \beta x \\ \beta y \end{bmatrix} e^{-\alpha(x^2 + y^2)} \qquad \text{for} \qquad t = 0 \tag{1}$$

where $\alpha = \ln(2)/9$ and $\beta = 0.04$. The parameters and the expressions for the initial values are defined as parameters and variables and loaded from files during the model setup. The pulse propagates in a uniform background flow $\mathbf{u}_0 = (\mathrm{Ma},0)$, where $\mathrm{Ma} = 0.5$. The analytical solution to Equation 1 is given by (see Ref. 2)

$$\begin{aligned} p(\mathbf{x}, t) &= \frac{1}{2\alpha}\int_0^{\infty}\left[\cos(\lambda t) - \frac{\beta}{2\alpha}\lambda\sin(\lambda t)\right]\lambda J_0(\lambda r)e^{-\frac{\lambda^2}{4\alpha}}d\lambda \\ r &= \sqrt{(x - \mathrm{Ma}t)^2 + y^2} \end{aligned} \tag{2}$$

In the model, the `integrate()` operator is used to express the analytical solution. The integration is performed on a finite interval.

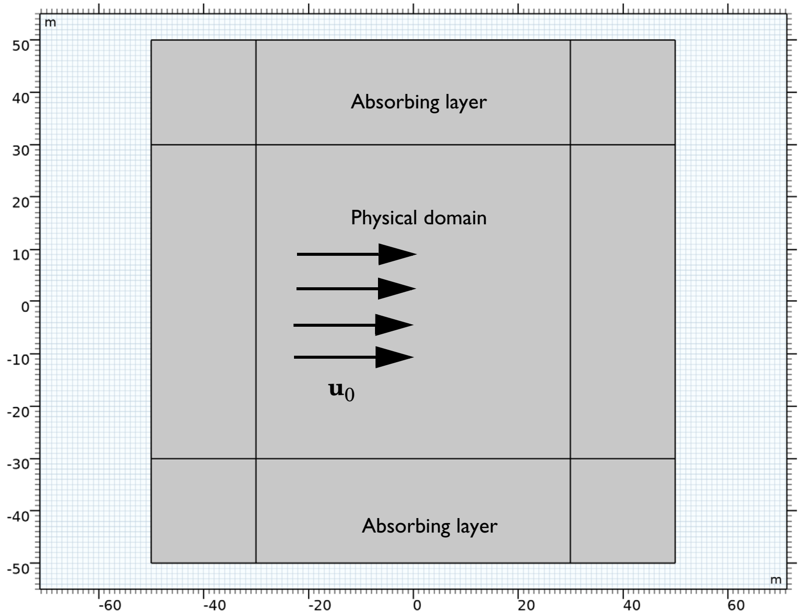

Figure 1: Geometry, physical domain and absorbing layers.

Results and Discussion

The propagation of the acoustic pulse is depicted in Figure 2 and Figure 3. The two figures show the acoustic pressure and the acoustic particle velocity, respectively, at four consecutive time instances. In Figure 4, the pressure and velocity are depicted at the final simulated time $t = 120$. Here, the pulse is inside the absorbing layer. It is evident how the scaling in the layer slows the pulse (it moves slower and slower towards the outer edges) and makes it propagate more normal to the outer boundary. This shows one of the principles of the absorbing layer. The other two mechanisms at work is filtering and a simple impedance condition at the outer boundary.

In the final two figures, the simulated results are compared with the analytical solution. In Figure 5, the pressure at point (20, 30) is depicted as function of time. In Figure 6, the pressure is depicted along the x-axis at $t = 50$. Both show very good agreement with the analytical solution. The absorbing layers can under optimal condition reduce the spuriously reflected waves to 1/1000 of the incident field amplitude.

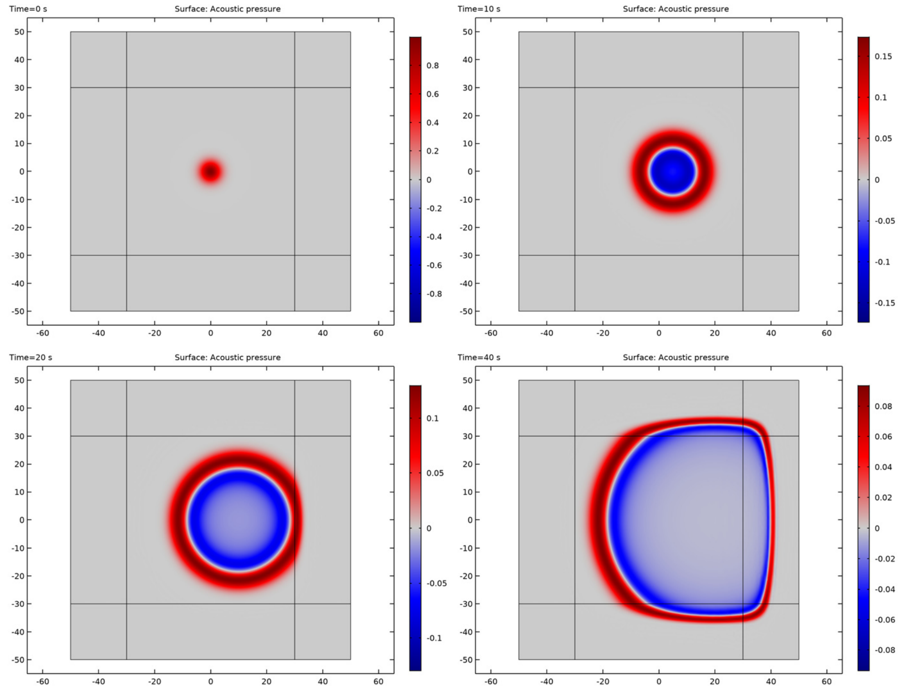

Figure 2: The pressure profile at different time steps.

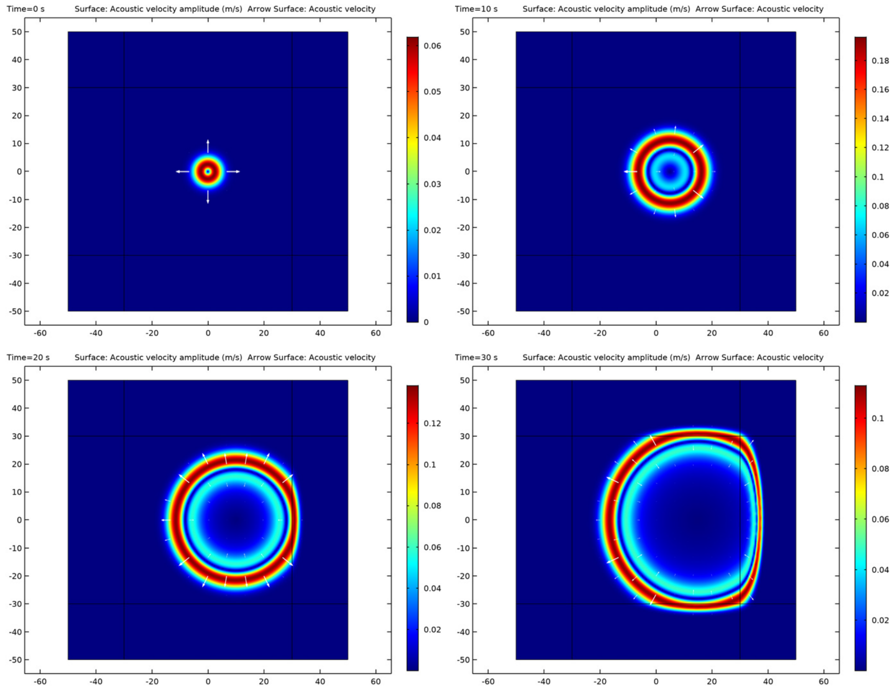

Figure 3: The acoustic velocity profiles at different time steps.

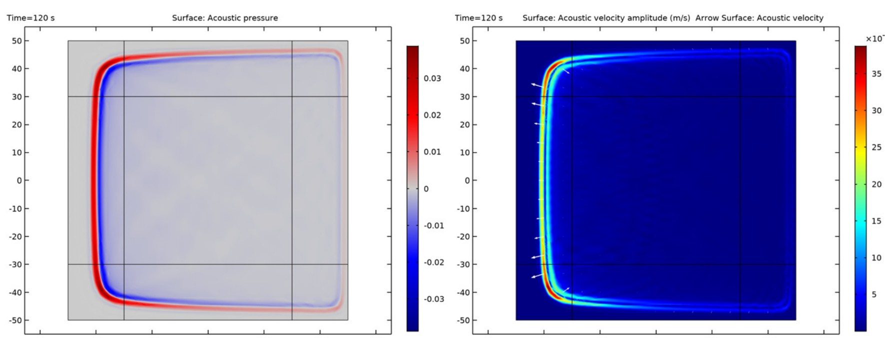

Figure 4: The pressure profile (left) and acoustic velocity profile (right) at the final simulated time t = 120.

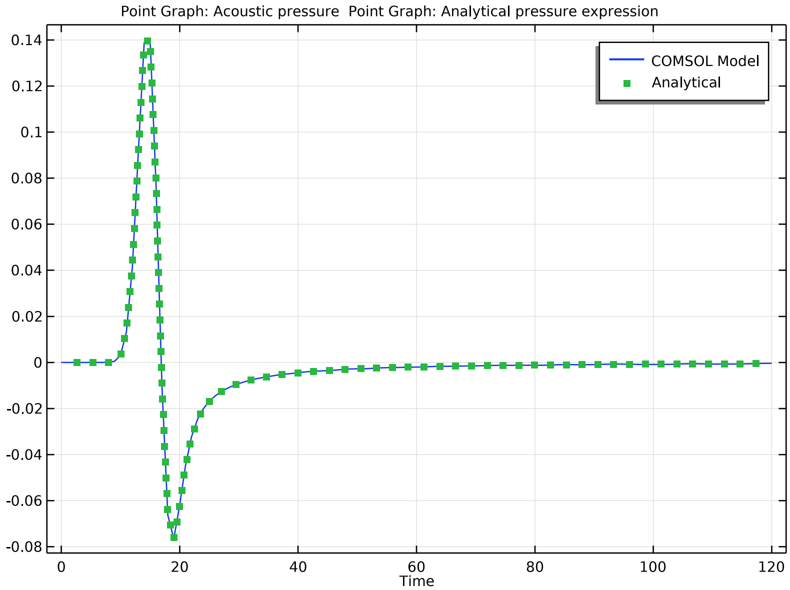

Figure 5: The pressure as function of time in point (x,y) = (20,30). The model solution compared with the analytical solution.

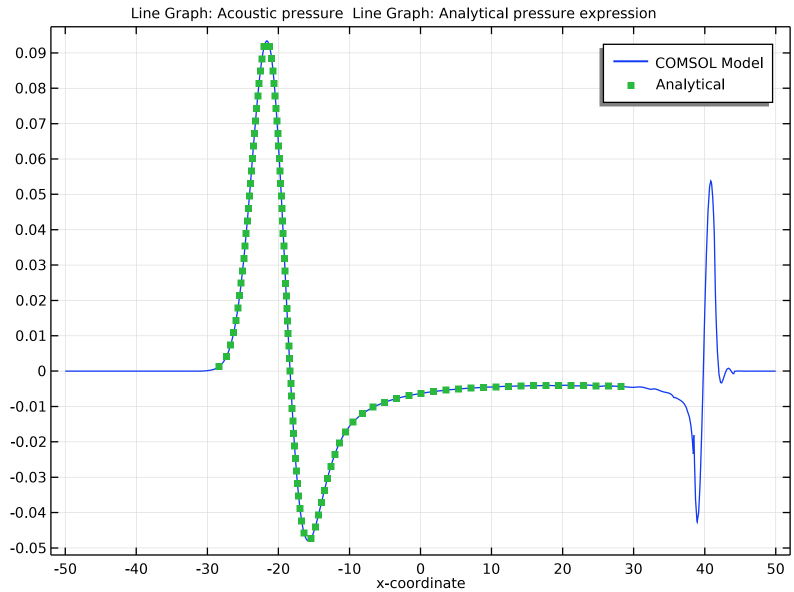

Figure 6: Pressure profile along the x-axis at t = 50 comparing the analytical solution and the COMSOL solution.

References

1. H.L. Atkins, "Application of Essentially Nonoscillatory Methods to Aeroacoustic Flow Problems," *Proceedings of ICASE/LaRCWorkshop on Benchmark Problems in Computational Aeroacoustics*, edited by J.C. Hardin, J.R. Ristorcelli, and C.K.W. Tam, NASA CP-3300, pp. 15–26, 1995.

2. H.L. Atkins and C.W. Shu, "Quadrature-Free Implementation of Discontinuous Galerkin Method for Hyperbolic Equations," *AIAA Journal*, vol. 36, pp. 775–782, 1998.

Application Library path: `Acoustics_Module/Tutorials,_Pressure_Acoustics/gaussian_pulse_absorbing_layers`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Acoustics>Ultrasound>Convected Wave Equation, Time Explicit (cwe)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters used to define the geometry, the Gaussian pulse, and the background flow properties.

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 `gaussian_pulse_absorbing_layers_parameters.txt`.

GEOMETRY 1

Square 1 (sq1)

1 In the **Geometry** toolbar, click **Square**.
2 In the **Settings** window for **Square**, locate the **Size** section.
3 In the **Side length** text field, type `W`.
4 Locate the **Position** section. From the **Base** list, choose **Center**.
5 Click to expand the **Layers** section. Select the **Layers to the left** check box.
6 Select the **Layers to the right** check box.
7 Select the **Layers on top** check box.

8 In the table, enter the following settings:

Layer name	**Thickness (m)**
`Layer 1`	`dW`

9 Click **Build All Objects**.

The geometry consists of a physical domain surrounded by the absorbing layers, see Figure 1.

DEFINITIONS

Next, load the variables that define the initial Gaussian shape (acoustic pressure and velocity components) and then set up the absorbing layers.

Variables 1

1 In the **Model Builder** window, expand the **Definitions** node.

2 Right-click **Component 1 (comp1)>Definitions** and choose **Variables**.

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

4 Click **Load from File**.

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

Absorbing Layer 1 (ab1)

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

2 Select Domains 1–4 and 6–9 only.

Before setting up the physics, change the unit system to be dimensionless.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

At the physics interface level, there are a number of interesting settings. Some are hidden per default. Start by enabling the **Advanced Physics** options to be able to see the **Filter Parameters for Absorbing Layers** section.

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

3 Click **OK**.

In the same way, you can also access the **Discretization** section by enabling it. Notice that the default discretization for both the pressure and the velocity is **Quartic**, that is, fourth order. Further details about this choice and the time explicit method used in the Convected Wave Equation interface can be found in the Acoustics Module User's Guide.

Convected Wave Equation Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Convected Wave Equation, Time Explicit (cwe)** click **Convected Wave Equation Model 1**.

2 In the **Settings** window for **Convected Wave Equation Model**, locate the **Model Input** section.

3 In the p_0 text field, type `p0`.

4 Specify the $\mathbf{u}_0$ vector as

Ma*c0	x
0	y

5 Locate the **Fluid Properties** section. From the ρ_0 list, choose **User defined**. In the associated text field, type `rho0`.

6 From the c_0 list, choose **User defined**. In the associated text field, type `c0`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `p_i`.

4 Specify the $\mathbf{u}$ vector as

u_i	x
v_i	y
0	z

Finally, set up the impedance condition and apply it on all the outer boundaries. This simple nonreflecting condition is the final piece (combined with filtering and coordinate stretching) that makes the absorbing layers work.

Acoustic Impedance 1

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

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

3 From the **Selection** list, choose **All boundaries**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

For the mesh, we have selected the default **Finer** setting. A more thorough approach would be to define the mesh size according to the highest frequency component in the model (the smallest wavelength to resolve). For the time explicit discontinuous Galerkin method (with default fourth order discretization), use two mesh elements per wavelength. Guidelines for meshing can be found in the documentation for the interface.

STUDY 1

Step 1: Time Dependent

Solve the model from time t = 0 to 120 in steps of 1 (dimensionless time units). Simply modify the default expression.

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 **Times** text field, type `range(0,1,120)`.

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

RESULTS

Acoustic Pressure (cwe)

The first default plot shows the acoustic pressure in the computational domain. You can change the times to plot the distribution at various times, see Figure 2 and Figure 4.

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

2 From the **Time (s)** list, choose **0**.

3 In the **Acoustic Pressure (cwe)** toolbar, click **Plot**.

The second default plot shows the acoustic velocity. The plots, for selected times, can be seen in Figure 3 and Figure 4.

Acoustic Velocity (cwe)

1 In the **Model Builder** window, click **Acoustic Velocity (cwe)**.

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

3 From the **Time (s)** list, choose **0**.

4 In the **Acoustic Velocity (cwe)** toolbar, click **Plot**.

Create some extra data sets to use in the following plots. Create a cut point to plot the pressure as function of time in a desired coordinate set. Create a cut line to plot the pressure along a line for a given time. Finally, create a data set with a selection restricting the solution to the physical domain (not the absorbing layers).

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

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

3 In the **x** text field, type `20`.

4 In the **y** text field, type `10`.

Cut Line 2D 1

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **x** to `-50`.

4 In row **Point 2**, set **x** to `50`.

Cut Line 2D 2

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **x** to `-30`.

4 In row **Point 2**, set **x** to `30`.

The pressure as function of time is depicted in Figure 5 where it is compared with the analytical solution.

1D Plot Group 4

1 In the **Results** toolbar, click **1D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 2D 1**.

Point Graph 1

1 Right-click **Pressure in Point** and choose **Point Graph**.

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

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

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

5 In the table, enter the following settings:

Legends
`COMSOL Model`

Point Graph 2

1 In the **Model Builder** window, right-click **Pressure in Point** and choose **Point Graph**.

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

3 In the **Expression** text field, type `p_a`.

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

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

6 In the table, enter the following settings:

Legends
`Analytical`

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

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

9 In the **Number** text field, type `100`.

10 In the **Pressure in Point** toolbar, click **Plot**.

The pressure on the x-axis for a given time is depicted in Figure 6 where it is compared with the analytical solution.

1D Plot Group 5

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

Line Graph 1

1 Right-click **Pressure over Cut Line** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Cut Line 2D 1**.

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

5 In the **Times (s)** list, select **40**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `p`.

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

8 In the **Expression** text field, type `x`.

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

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

11 In the table, enter the following settings:

Legends
`COMSOL Model`

12 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

13 In the **Pressure over Cut Line** toolbar, click **Plot**.

14 Click **Plot**.

Line Graph 2

1 In the **Model Builder** window, right-click **Pressure over Cut Line** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Cut Line 2D 2**.

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

5 In the **Times (s)** list, select **40**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `p_a`.

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

8 In the **Expression** text field, type `x`.

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

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

11 In the **Number** text field, type `100`.

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

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

14 In the table, enter the following settings:

Legends
`Analytical`

15 Locate the **Quality** section. From the **Resolution** list, choose **Extra fine**.

Note that when plotting the solution of the simulation, as function of spatial variables (line plot or a surface), you need to increase the resolution. This is because of the default quartic elements used. The default plots generated already have a higher default resolution.

16 In the **Pressure over Cut Line** toolbar, click **Plot**.

Finally, plot the pressure only in the physical domain. This plot can, for example, be used to create a nice animation. Under the **Export >** node select **Animation** and then select the last plot.

Another interesting observation is that the remaining (spuriously) reflected waves have an amplitude which is a factor 1000 smaller than the original signal (from the solution at time t = 0).

Acoustic Pressure (cwe) 1

1. In the **Model Builder** window, right-click **Acoustic Pressure (cwe)** and choose **Duplicate**.
2. In the **Settings** window for **2D Plot Group**, type `Acoustic Pressure (cwe) Selection` in the **Label** text field.

Selection 1

1. In the **Model Builder** window, expand the **Results>Acoustic Pressure (cwe) Selection** node.
2. Right-click **Acoustic Pressure** and choose **Selection**.
3. Select Domain 5 only.
4. Click the **Zoom Extents** button in the **Graphics** toolbar.

5 In the **Acoustic Pressure (cwe) Selection** toolbar, click **Plot**.