Created in COMSOL Multiphysics 6.1

Transient Gaussian Explosion

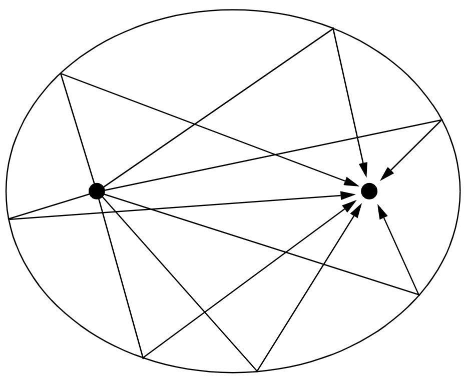
This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example introduces some important concepts to have in mind when solving transient problems. In particular, it examines the relationship between the frequency content in the sources driving the model, the mesh resolution, and the time step.

Model Definition

An ellipse with sound-hard walls has the interesting property that an acoustic signal emanating from one of the foci refocuses at the other focal point b/c seconds later, where b (SI unit: m) is the major axis length and c (SI unit: m/s) is the speed of sound.



Inspired by Ref. 1 and Ref. 2, this model involves a Gaussian explosion at one focus of an ellipse to illustrate some properties of time-dependent acoustic problems. The major and minor axis lengths are 10 m and 8 m, respectively. The major axis coincides with the x-axis and the foci are located at $x = -3$ m and $x = 3$ m. Because of symmetry, the model can be limited to the upper half plane.

Denoting the fluid density by ρ and the speed of sound by c, the acoustic pressure field $p(\mathbf{x}, t)$, inside the elliptical chamber is governed by the scalar wave equation

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}\nabla p\right) = S(\mathbf{x}, t)$$

where the point-source term on the right-hand side is given by

$$S(\mathbf{x}, t) = \frac{dg}{dt}(t)\,\delta^{(2)}(\mathbf{x} - \mathbf{x}_0)$$

where $\delta^{(2)}$ is the two-dimensional Dirac delta function. The time dependence of the explosion is determined by the cutoff Gaussian pulse

$$g(t) = \begin{cases} Ae^{-\pi^2 f_0^2 (t-\tau)^2} & 0 < t < 2\tau \\ 0 & \text{otherwise} \end{cases}$$

describing the rate of airflow (SI unit: m^2/s) away from the source, located at $\mathbf{x} = \mathbf{x}_0$. The parameter f_0, which is proportional to the pulse bandwidth, is chosen as $f_0 = 380$ Hz. As the following plots show, by taking $\tau = 1/f_0$ the pulse very closely approximates a full Gaussian, the effect of the cutoff tails being numerically insignificant.

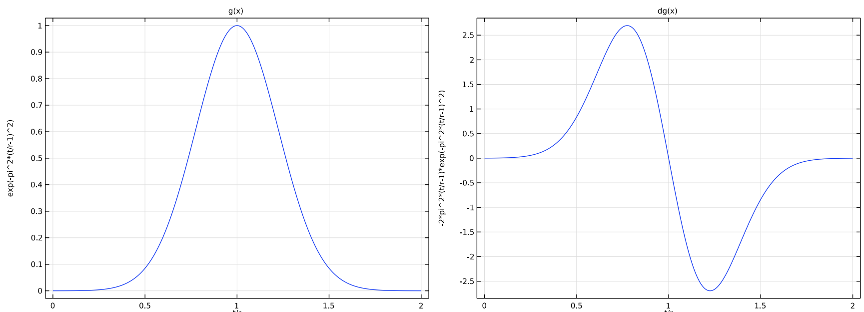


Figure 1: Normalized Gaussian pulse (left) and its derivative (right).

A particularly interesting property of the Gaussian function is that its Fourier transform, depicted in Figure 3, is equally simple (neglect cutoff effects):

$$G(\omega) \equiv \int_{-\infty}^{\infty} g(t) e^{-i\omega t} dt = \frac{2A\sqrt{\pi}}{\omega_0} e^{-\frac{\omega^2}{\omega_0^2} - i\omega t_0}$$

where $\omega_0 = 2\pi f_0$. The magnitude of the Fourier transform falls off quickly for increasing angular frequencies ω. Practically all the energy in the signal is contained in the frequency band $-2\omega_0 < \omega < 2\omega_0$ with most of it concentrated between $-\omega_0$ and ω_0.

Therefore, when using a forcing function of this type, it is enough to resolve wavelengths corresponding to the angular frequency ω_0, which in turn corresponds to the frequency f_0. In this model, the parameter that defined the maximal frequency to resolve is f_{max}, it is set equal to 600 Hz which is slightly higher than f_0 (and will resolve the pulse fully).

In order to well resolve the propagating pulse in space the rule of thumb is to use a minimum of 5 to 6 elements per wavelength (at the maximal frequency), when using quadratic Lagrange elements. The typical mesh size is given by

$$h_{\max} = \frac{\lambda_{\min}}{N} = \frac{c}{f_{\max}N}$$

Where $h_{\max}$ is a typical maximal mesh-element size, and N is the number of elements per wavelength required to resolve a harmonic wave with some accuracy. The following discussion uses $N = 5$. The important point is that there is little to gain in prescribing a forcing function that contains frequencies that the mesh cannot resolve.

In addition to controlling the pulse shape, the mesh resolution imposes a restriction on the internal time steps size taken by the solver. In the transient interfaces of the Acoustics Module, the solver is controlled from the **Transient Solver Settings** section found at the top physics level. The user should enter the **Maximum frequency to resolve** in the model. It is also possible to choose a **Time stepping** (method). Here it is recommended to use the default **Fixed (preferred)** option.

TIME STEPPING EXPLANATION

The logic for the automatic choice made is as follows. The relationship between mesh size and time-step size is closely related to the CFL number (Ref. 3), which is defined as

$$\text{CFL} = \frac{c\Delta t}{h}$$

This nondimensional number can be interpreted as the fraction of an element the wave travels in a single time step. A CFL number around 1 would correspond to the same resolution in space and time if the discretization errors were of the same size; however, that is normally not the case.

By default, COMSOL Multiphysics uses the implicit second-order accurate method generalized-α to solve transient acoustics problems. In space, the default is second-order Lagrange elements. Generalized-α introduces some numerical damping of high frequencies but much less than the BDF method.

The temporal discretization errors for generalized-α are larger than the spatial discretization errors when second-order elements are used in space. The limiting step size, where the errors are of roughly the same size, can be found somewhere at $\text{CFL} < 0.2$. You can get away with a longer time step if the forcing does not make full use of the mesh resolution; that is, if high frequencies are absent from the outset.

When the excitation contains all the frequencies the mesh can resolve, there is no point in using an automatic time-step control which can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune when there is weak but important high-frequency content. Instead, you can use your knowledge of the typical mesh size, speed of sound, and CFL number to calculate and prescribe a fixed time step. This is exactly the default behavior when the **Fixed (preferred)** method is chosen in the **Transient Solver Settings** section. The **Free** option corresponds to the automatic time-step control but with some tighter controls of the allowed time-steps. This latter option is still not recommended as the manual option typically yields much better results (and is faster).

The internal time step generated by the **Fixed (preferred)** option and the entered **Maximal frequency to resolve** is set by assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength). The following step is generated

$$\Delta t = \frac{h_{\max}\mathrm{CFL}}{c} = \frac{\mathrm{CFL}}{f_{\max}N} \approx \frac{1}{60f_{\max}}$$

Assuming that N is between 5 and 6 and the CFL number is roughly 0.1. These values give a good margin of safety.

Results and Discussion

Using the properties of air for the medium and selecting a mesh density based on the parameters. The model runs for 0.035 s so that you can study the refocusing at the right-hand focus point at roughly 0.0315 s.

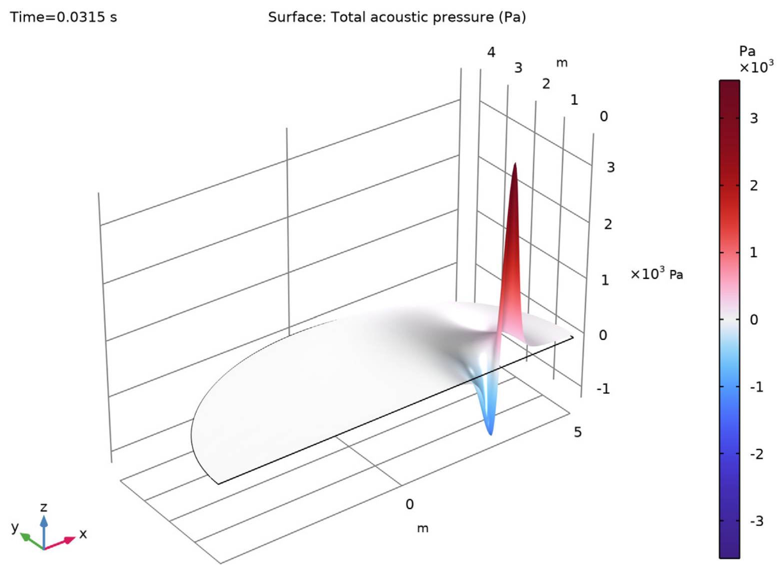


Figure 2: The refocusing occurs at roughly 0.0315 s. An animation gives a better feeling for the process.

For the selected combination of mesh size, pulse shape, and time step, the solution can be shown to be both smooth and accurate. Selecting a smaller value for N leads to oscillations if the CFL number is small enough (choosing a too low frequency to resolve), while selecting a higher CFL number (and consequentially a larger time step) leads to an inaccurate solution.

Figure 3 shows the Fourier transform of the source signal. Plotting the frequency content of the source signal is a good way for selecting the maximal frequency to resolve in the model.

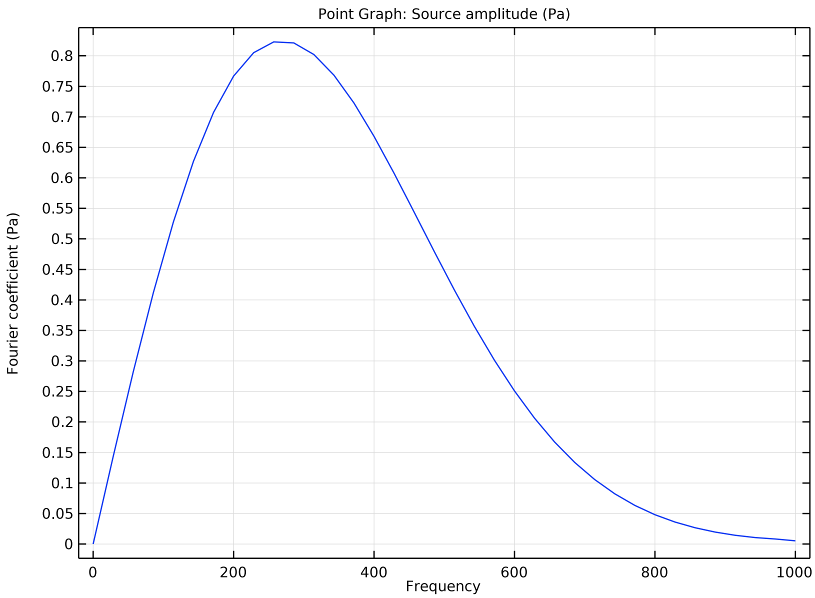


Figure 3: Fourier transform of the source signal.

References

1. B. Yue and M.N. Guddati, "Dispersion-reducing Finite Elements for Transient Acoustics", *J. Acoust. Soc. Am.*, vol. 118, no. 4, pp. 2132–2141, 2005.

2. H.-O. Kreiss, N.A. Peterson, and J. Yström, "Difference Approximations for the for the second order wave equation", *SIAM J. of Num. Analys.*, vol. 40, 1940–1967, 2002.

3. R. Courant, K.O. Friedrichs, and H. Lewy, "On the Partial Difference Equations of Mathematical Physics", *IBM Journal*, vol. 11, pp. 215–234, 1956.

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

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

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Ellipse 1 (e1)

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

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

3 In the **a-semiaxis** text field, type 5.

4 In the **b-semiaxis** text field, type 4.

5 In the **Sector angle** text field, type 180.

Point 1 (pt1)

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

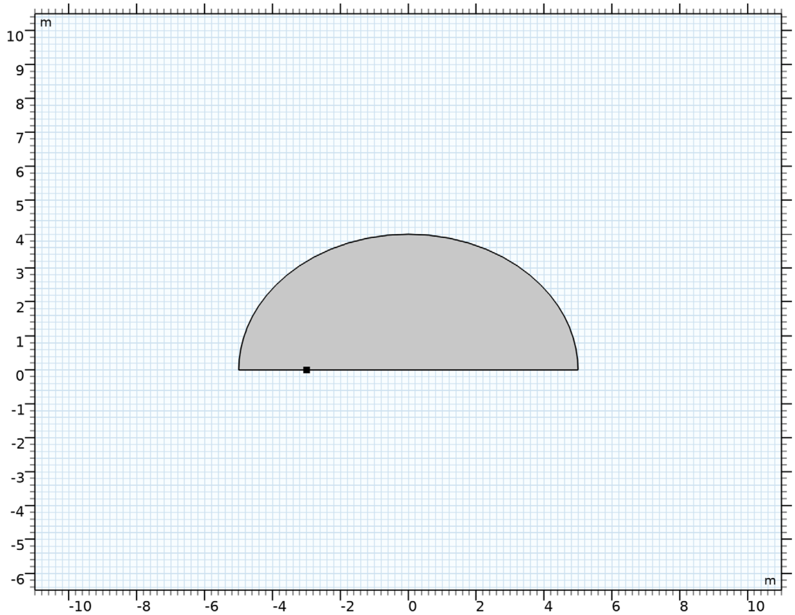
2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type -3.

4 Click **Build All Objects**.

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

The completed geometry should look like that in the figure below.



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 In the table, enter the following settings:

Name	Expression	Value	Description
c_air	343[m/s]	343 m/s	Speed of sound in air
f_max	600[Hz]	600 Hz	Maximum frequency to resolve
N	5	5	Elements per wavelength
h_max	c_air/f_max/N	0.11433 m	Typical element size
A	4[m^2/s]	4 m²/s	Point-source amplitude
f0	380[Hz]	380 Hz	Source frequency bandwidth
t0	1/f0	0.0026316 s	Source pulse half width

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>Air**.

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

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

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

Enter the maximal frequency to be resolved in the model. This value can be assessed from a Fourier analysis of the sources as depicted in Figure 3.

Under the **Transient Solver Settings** section enter the **Maximum frequency to resolve** and set it to `f_max`.

Point Source 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Pressure Acoustics, Transient (actd)** and choose **Points>Point Source**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Source**, locate the **Point Source** section.

4 From the **Type** list, choose **Gaussian pulse**.

5 In the A text field, type `A`.

6 In the f_0 text field, type `f0`.

7 In the t_p text field, type `t0`.

MESH

Proceed and generate the mesh using the **Physics-controlled mesh** functionality. The frequency controlling the maximum element size is per default taken **From study**. Set the desired **Frequencies** in the study step. In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. In this model we use the default **Automatic** option, which gives 5 elements per wavelength.

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(0,0.5e-3,35e-3)`.

This setting gives you a solution output at every 0.5 ms from t = 0 to t = 35 ms. This should not be confused with the time steps actually taken by the solver. These are set automatically based on the settings entered above. To see where they end up, do the following:

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-Dependent Solver 1**.

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

The solver automatically picks up the settings entered at the physics level (in the Transient Solver Settings section) when the default solver is generated or when solve is clicked the first time. It is in general not necessary to edit these settings.

Experienced users maybe find it useful to edit these settings.

MESH 1

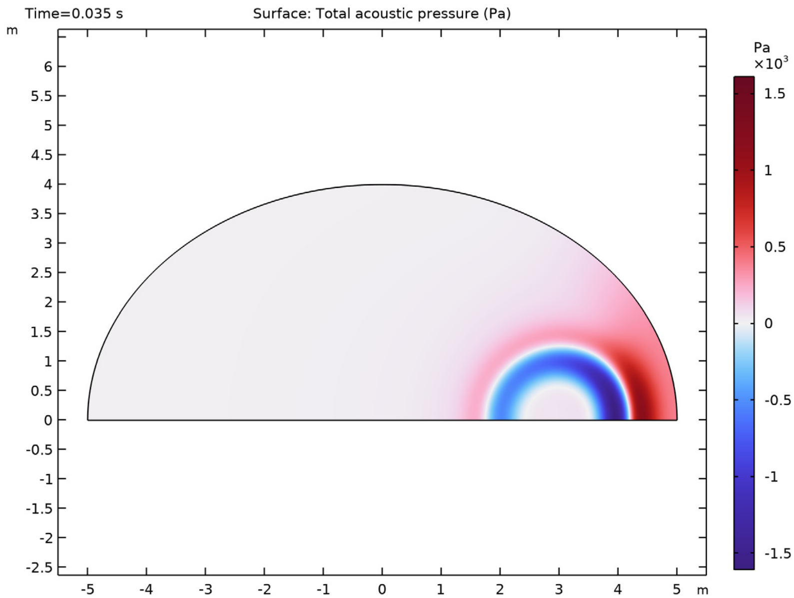
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

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

RESULTS

Acoustic Pressure (actd)



The default plot shows the pressure at the final time. To get a more attractive plot, you can add a height.

Height Expression 1

1. In the **Model Builder** window, expand the **Acoustic Pressure (actd)** node.
2. Right-click **Surface 1** and choose **Height Expression**.

   You can select different times to look at the wave using the parent Acoustic Pressure (`actd`) node. At t = 0.0315 s, you are close to the moment when the waves refocus.

Acoustic Pressure (actd)

1. In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Time (s)** list, choose **0.0315**.
4. In the **Acoustic Pressure (actd)** toolbar, click **Plot**.

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

The resulting plot is found in Figure 2. It is illustrative to animate transient problems in general and wave propagation in particular; you can do this by right-clicking the Export node and adding an Animation feature.

It is also possible to plot the field on all sides of the point source by defining a mirror dataset.

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

The symmetry line is the x-axis.

2 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

3 From the **Axis entry method** list, choose **Point and direction**.

4 Find the **Direction** subsection. In the **X** text field, type `1`.

5 In the **Y** text field, type `0`.

Acoustic Pressure (actd)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.

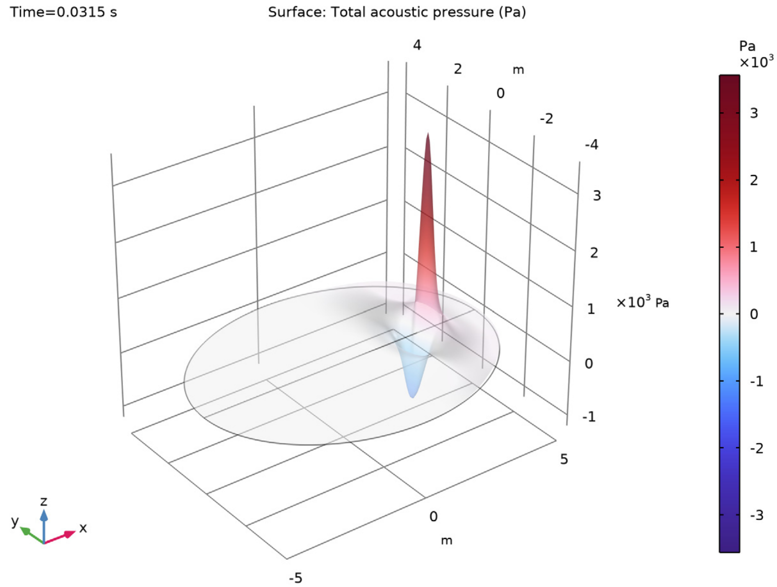
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

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

4 In the **Acoustic Pressure (actd)** toolbar, click **Plot**.

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

6 Click the **Transparency** button in the **Graphics** toolbar.



7 Click the **Transparency** button in the **Graphics** toolbar to restore the default transparency setting.

The following instructions show how to create Figure 1. These instructions are optional.

Plot the normalized Gaussian and its derivative by plotting two analytic functions.

GLOBAL DEFINITIONS

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `exp(-pi^2*(x-1)^2)`.

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

Argument	Lower limit	Upper limit	Unit
x	0	2	

5 Click **Create Plot**.

RESULTS

Normalized Gaussian

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

2 Locate the **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `t/r`.

4 In the **y-axis label** text field, type `exp(-pi^2*(t/r-1)^2)`.

5 In the **Normalized Gaussian** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Analytic 2 (an2)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `-2*pi^2*(x-1)*exp(-pi^2*(x-1)^2)`.

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

Argument	Lower limit	Upper limit	Unit
x	0	2	

5 Click **Create Plot**.

RESULTS

Derivative of norm. Gaussian

1 In the **Settings** window for **1D Plot Group**, type `Derivative of norm. Gaussian` in the **Label** text field.

2 Locate the **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `t/r`.

4 In the **y-axis label** text field, type `-2*pi^2*(t/r-1)*exp(-pi^2*(t/r-1)^2)`.

5 In the **Derivative of norm. Gaussian** toolbar, click **Plot**.

Finally, create the plot shown in Figure 3 where the frequency content of the source is analyzed.

Fourier Transform of Source

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

2 In the **Settings** window for **1D Plot Group**, type `Fourier Transform of Source` in the **Label** text field.

Point Graph 1

1 Right-click **Fourier Transform of Source** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Pressure Acoustics, Transient>Sources>actd.mls1.S - Source amplitude - N/m^2**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Discrete Fourier transform**.

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

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

7 In the **Fourier Transform of Source** toolbar, click **Plot**.