Created in COMSOL Multiphysics 6.2

Nonlinear Slit Resonator

Introduction

In many applications, acoustic waves interact with surfaces that have small perforations or slits. These can exist in muffler systems; in soundproofing structures; in liners for noise suppression in jet engines; or in grilles and meshes in front of, for example, miniature speakers in mobile devices.

At medium to high sound pressure levels, the local particle velocity in the narrow region of the perforate or slit can be so large that the linear assumptions of acoustics break down. Typically, vortex shedding takes place in the vicinity of that region. This leads to nonlinear losses and in audio applications also nonlinear distortion of the sound signal. The nonlinear effects are sometimes included through semiempirical parameters in analytical transfer impedance models for perforates.

In this tutorial, a narrow slit is located in front of a resonator volume. The model couples *Pressure Acoustics, Transient* and *Thermoviscous Acoustics, Transient* to model the nonlinear transient problem. The complex nonlinear losses associated with the vortex shedding and the viscous dissipation are captured using the *Nonlinear Thermoviscous Acoustics Contributions* feature. The incident acoustic field has an amplitude corresponding to 155 dB SPL.

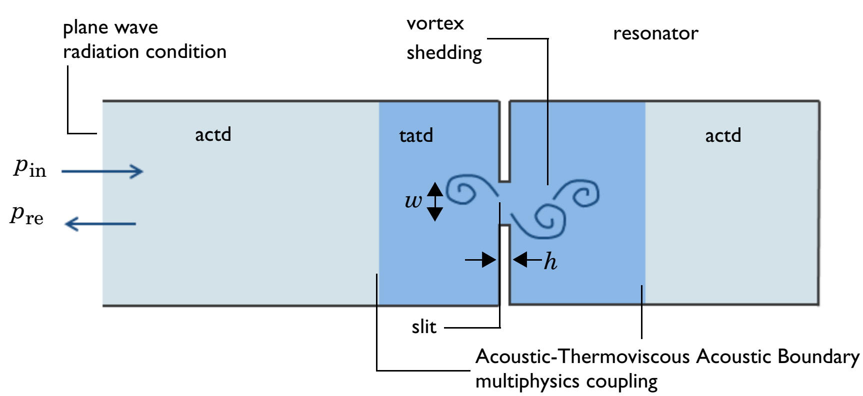


Figure 1: Sketch of the resonator system with annotations.

Model Definition

A sketch of the resonator system is depicted in Figure 1. It consists of a main duct leading to a narrow slit backed by a resonator. The slit has a height h of 1.02 mm (0.04 in) and a width w of 1.27 mm (0.05 in). The geometry and model parameters are taken from Ref. 1

(parallel slit P1, in the reference). The reference also contains results of the experimentally measured reflection coefficients (in figure 8 in the reference).

A harmonic signal $p_{\mathrm{in}}(t) = p_0 \cdot \sin(2\pi f_0 t)$ is sent in at the left (note that the sketch is rotated compared to the model, where the signal enters the top). The signal interacts with the slit and the resonator, and is reflected. The reflected signal is called $p_{\mathrm{re}}(t)$. The amplitude of the incident signal corresponds to 155 dB SPL. This leads to high local velocities in the slit region which lead to vortex shedding and distortion of the reflected harmonic signal (generation of higher harmonics) as the acoustics are nonlinear. The model is solved for f_0 set to 500 Hz, 1 kHz, 1.5 kHz, and 2 kHz.

To model this in COMSOL, use the **Thermoviscous Acoustics, Transient** interface with the **Nonlinear Thermoviscous Acoustic Contributions** feature in a domain around the slit. The rest of the domains are modeled with **Pressure Acoustics, Transient**. The two physics are coupled using the **Acoustic-Thermoviscous Acoustics Boundary** multiphysics coupling. The nonlinear effects, captured by the nonlinear thermoviscous feature, are large and require that stabilization is enabled. To speed up the model, the discretization in the thermoviscous domain is switched to all linear.

The **Nonlinear Thermoviscous Acoustic Contributions** feature also allow using a **Second order** density expansion. The nonlinearities in this model are dominated by the high particle velocity and vortex shedding (detachment). The default **First order** representation is adequate. The validity of the assumption is checked in Figure 6.

Results and Discussion

The evolution of the acoustic velocity for the 2 kHz excitation is shown at 6 time instances, for the last period simulated, in Figure 2. The model is run until the reflected signal hits the inlet/outlet (parameter `Tstart`) plus an additional 5 periods (of the incident harmonic signal). The pressure in the slit, as well as the incident and reflected pressure for the 500 Hz excitation, is depicted in Figure 3. An FFT of the pressure in the slit for all four excitation frequencies is depicted in Figure 4. The plot shows a large peak for the excitation frequencies and then the additional generated harmonics.

The absolute value of the reflection coefficient is computed using the `timeint()` operator in order to implement the expression for the reflection coefficient. The results are shown in Figure 5 and show good agreement with the experimental results reported in Ref. 1.

$$|R|^2 = \left(\int_{T_s}^{T_s + 4T_0} p_{\mathrm{re}}(t)^2 dt \right) \Big/ \left(\int_{T_s}^{T_s + 4T_0} p_{\mathrm{in}}(t)^2 dt \right)$$

where T_s is the start time for the integration given as the moment where the reflected signal arrives back to the inlet/outlet. The time averaging is done over 4 periods.

Finally, to check the validity of the first order density expansion, the value

$$\frac{\max(\rho_t)}{\rho_0}$$

is plotted as function of time. ρ_t is the acoustic density fluctuation, evaluated with the variable `tatd.rho_t`. The maximum is taken over the thermoviscous domain using the `maxop1()` operator set up in the **Definitions**. The plot shows one peak up to 0.07 while most fluctuations are below 0.04, meaning that it is still adequate to assume that linearity applies as $|\rho_t| << \rho_0$.

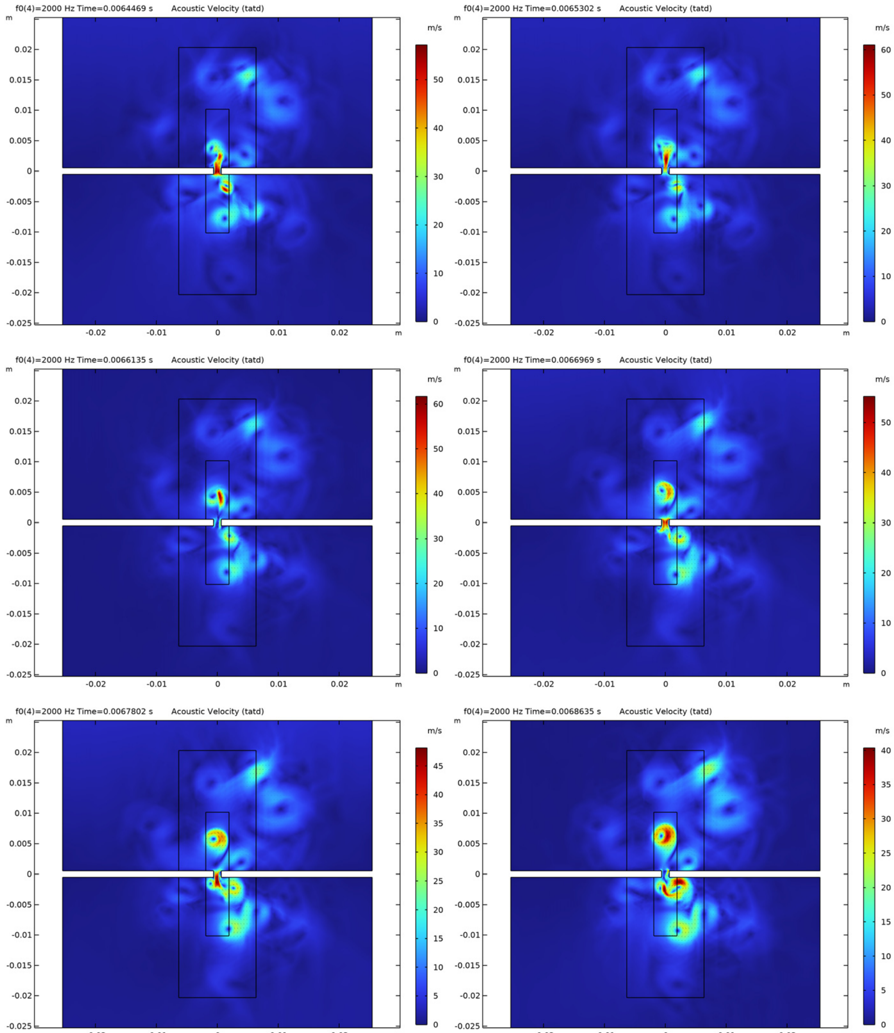


Figure 2: Evolution of the acoustic velocity fluctuations showing vortex shedding with six images over the last simulated period $T_0 = 1/f_0$, for $f_0 = 2$ kHz.

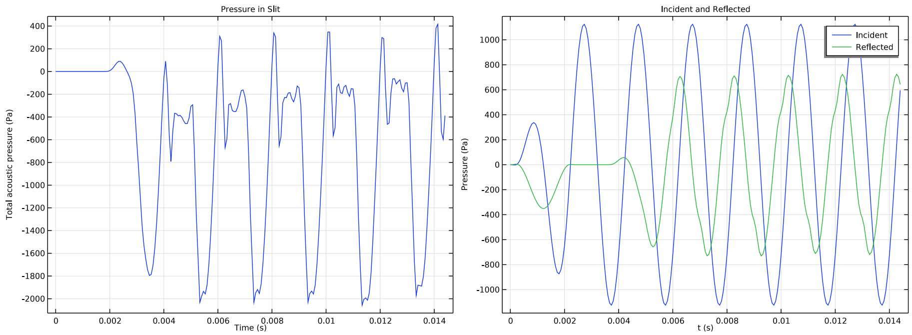


Figure 3: (left) Pressure in the slit for the 500 Hz excitation and (right) the incident and reflected signal at the inlet/outlet.

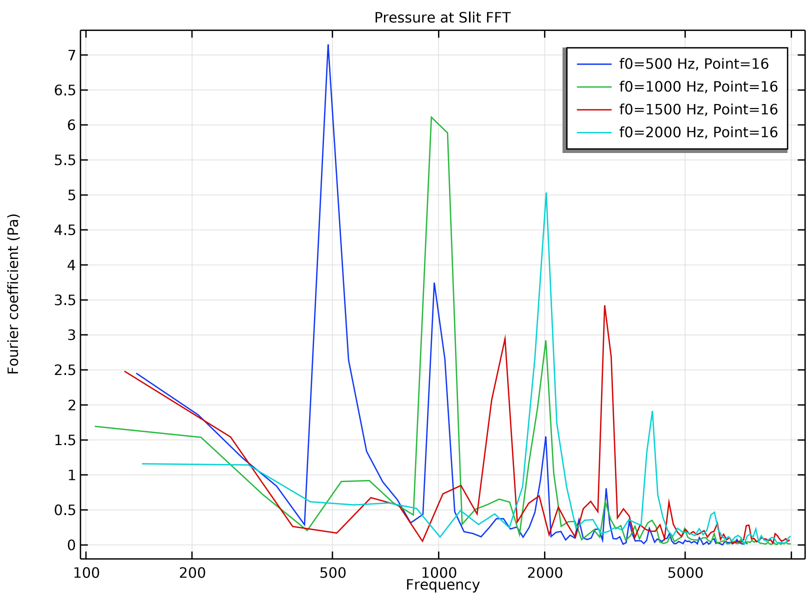


Figure 4: FFT of the pressure signal measured in the slit for the four excitation frequencies.

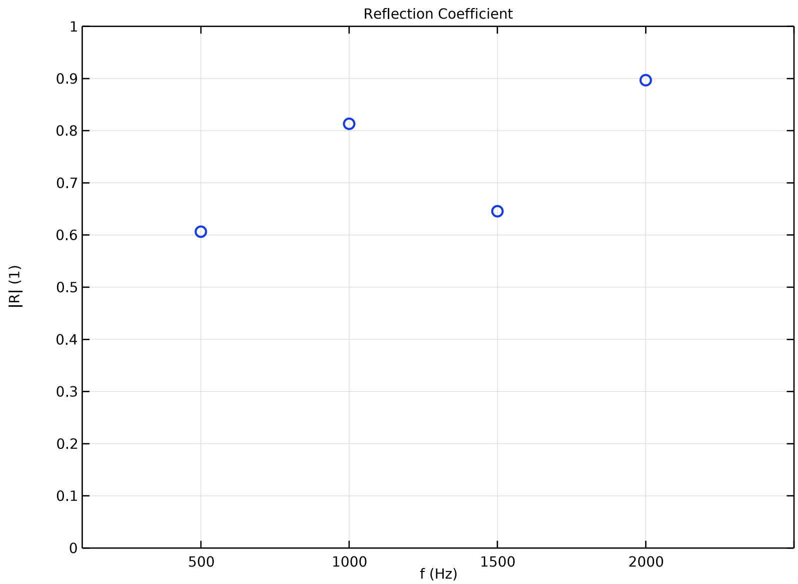


Figure 5: Absolute value of the reflection coefficient for the four excitation frequencies.

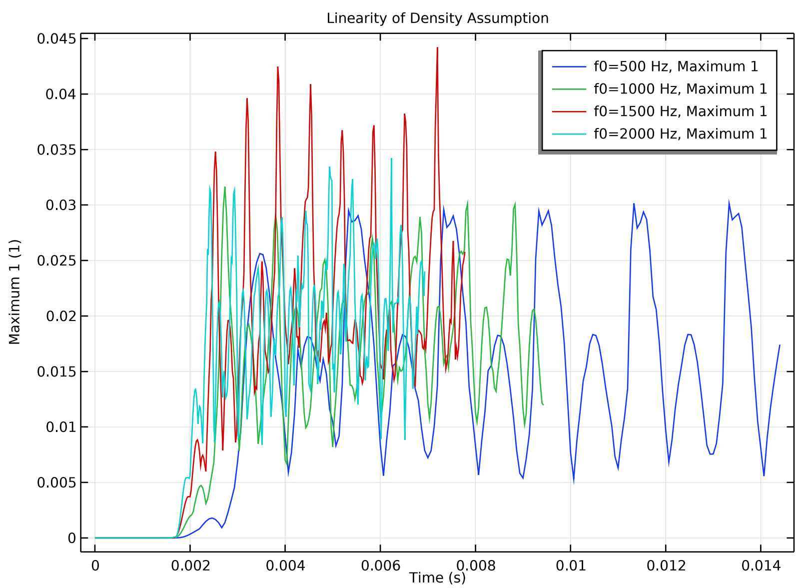


Figure 6: Maximum absolute density fluctuation relative to the equilibrium density.

Reference

1. C. K. W. Tam, H. Ju, M G. Jones, W. R. Watson, and T. L. Parrot, “A computational and experimental study of slit resonators,” *J. Sound. Vib.*, vol. 284, pp. 947-984, 2005.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/nonlinear_slit_resonator`

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>Thermoviscous Acoustics>Thermoviscous Acoustics, Transient (tatd)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Time Dependent**.
8 Click **Done**.

Load the model parameters (source, frequency, harmonics to resolve, and so on) and the geometry parameters.

GLOBAL DEFINITIONS

Parameters 1 - Model

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Parameters 1 - Model` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Parameters 2 - Geometry

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

2 In the **Settings** window for **Parameters**, type `Parameters 2 - Geometry` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Build the geometry. It consists of several **Rectangle** features that are all parameterized. In the geometry several domains are created that will help meshing near the slit.

GEOMETRY 1

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_tube`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **y** text field, type `h_tube/2-(h_r+h_slit/2)`.

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 `w_tube`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

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

Layer name	**Thickness (m)**
`Layer 1`	`(w_tube-w_slit)/2`

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

8 Select the **Layers to the left** check box.

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

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 `10*w_slit`.

4 In the **Height** text field, type `40*h_slit`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Rectangle 4 (r4)

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 `3*w_slit`.

4 In the **Height** text field, type `20*h_slit`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Rectangle 5 (r5)

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_tube`.

4 In the **Height** text field, type `0.14[m]`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Union 1 (uni1)

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

2 Click the **Select All** button in the **Graphics** toolbar.

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

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 **uni1**, select Domains 3, 7, 10, and 13–15 only.

Delete Entities 2 (del2)

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

2 On the object **del1**, select Boundaries 24 and 25 only.

3 In the **Settings** window for **Delete Entities**, click **Build All Objects**.

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.

Set up several selections to simplify the model setup. Define an integral operator for the inlet as well as a maximum operator. Both will be used in postprocessing.

DEFINITIONS

Inlet

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Selections>Explicit**.

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

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 10 only.

Thermoviscous Acoustics

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

2 In the **Settings** window for **Explicit**, type `Thermoviscous Acoustics` in the **Label** text field.

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

4 Click the **Select All** button in the **Graphics** toolbar.

5 Select Domains 2, 3, and 5–7 only.

Pressure Acoustics

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

2 In the **Settings** window for **Explicit**, type `Pressure Acoustics` in the **Label** text field.

3 Select Domains 1 and 4 only.

Integration 1 (intop1)

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

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

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

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

Maximum 1 (maxop1)

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

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

3 From the **Selection** list, choose **Thermoviscous Acoustics**.

THERMOVISCOUS ACOUSTICS, TRANSIENT (TATD)

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

2 In the **Settings** window for **Thermoviscous Acoustics, Transient**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Thermoviscous Acoustics**.

For nonlinear transient models, it is important the set the **Maximum frequency to resolve** for the solver, taking the harmonic generation into account. In this model, include `N0` harmonics, defined in the parameters as 6.

4 Locate the **Transient Solver and Mesh Settings** section. In the f_{max} text field, type `N0*f0`.

To speed up the solution time, it can also be advantageous to switch to an all linear discretization (P1-P1-P1). This is particularly true for nonlinear problems. Note that this is only possible if stabilization is used.

5 Click to expand the **Discretization** section. From the **Element order for velocity** list, choose **Linear**.

6 From the **Element order for temperature** list, choose **Linear**.

Turn on the stabilization as the problem solved is highly nonlinear and the model uses P1-P1-P1 discretization.

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

8 In the **Show More Options** dialog box, select **Physics>Stabilization** in the tree.

9 In the tree, select the check box for the node **Physics>Stabilization**.

10 Click **OK**.

11 In the **Settings** window for **Thermoviscous Acoustics, Transient**, click to expand the **Stabilization** section.

12 From the **Stabilization method** list, choose **Galerkin least-squares (GLS) stabilization**.

Nonlinear Thermoviscous Acoustics Contributions 1

1 In the **Physics** toolbar, click **Domains** and choose **Nonlinear Thermoviscous Acoustics Contributions**.

2 In the **Settings** window for **Nonlinear Thermoviscous Acoustics Contributions**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Thermoviscous Acoustics**.

Wall 2

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

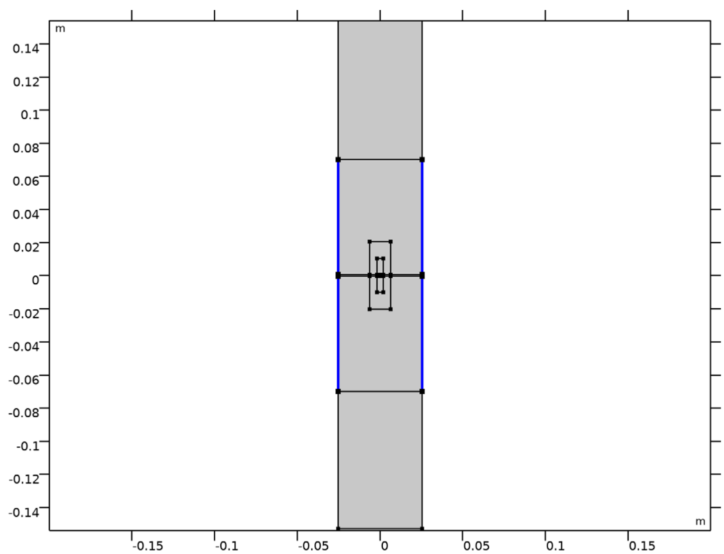
Use adiabatic and slip conditions on the walls away from the slit. This is also necessary to get a physically valid coupling to Pressure Acoustics.

2 In the **Settings** window for **Wall**, locate the **Mechanical** section.

3 From the **Mechanical condition** list, choose **Slip (perfect)**.

4 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

5 Select Boundaries 3, 6, 36, and 37 only.



PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Pressure Acoustics, Transient (actd)** node, then click **Pressure Acoustics, Transient (actd)**.

2 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Pressure Acoustics**.

4 Locate the **Transient Solver and Mesh Settings** section. In the f_{max} text field, type `N0*f0`.

Plane Wave Radiation 1

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

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

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

Incident Pressure Field 1

1 In the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

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

4 From the c list, choose **From material**.

5 From the **Material** list, choose **Air (mat1)**.

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

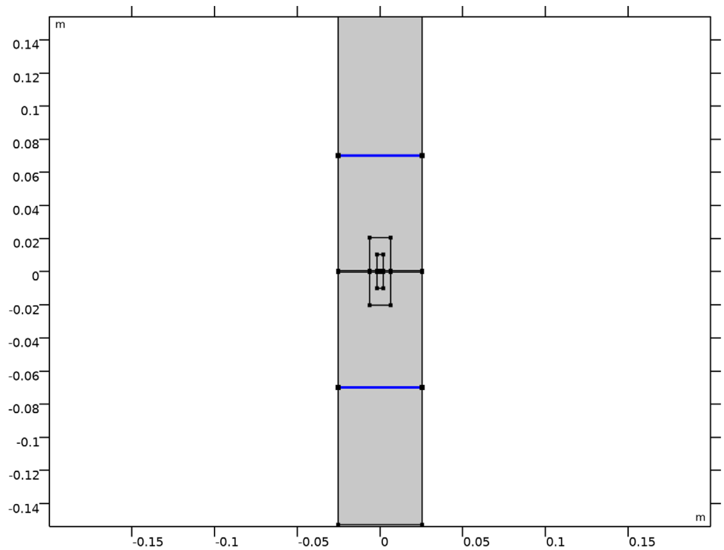
MULTIPHYSICS

Acoustic–Thermoviscous Acoustic Boundary 1 (atb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic–Thermoviscous Acoustic Boundary**.

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

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



Create a mesh that resolves the characteristic length scales in the model. The mesh needs to be fine near the slit where vortex shedding occurs. The thickness of the thermal and viscous boundary layers (parameter `dvisc`) dictate the length scale. At the wall create a boundary layer mesh with a thickness that is a fraction of the boundary layer thickness (here using `0.1*dvisc`). Remember that linear elements are used in the model.

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lam0/6`.

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

6 In the **Maximum element growth rate** text field, type `1.1`.

7 In the **Resolution of narrow regions** text field, type `3`.

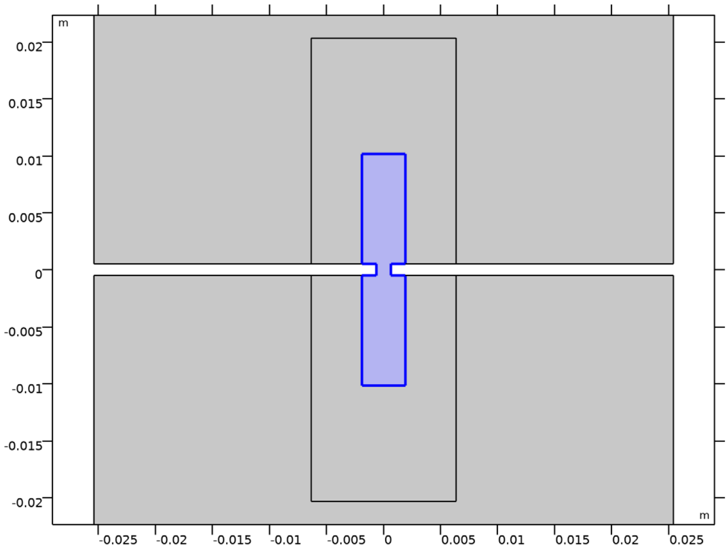
Size 1

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 Domain 7 only.



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

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `2.5*dvisc`.

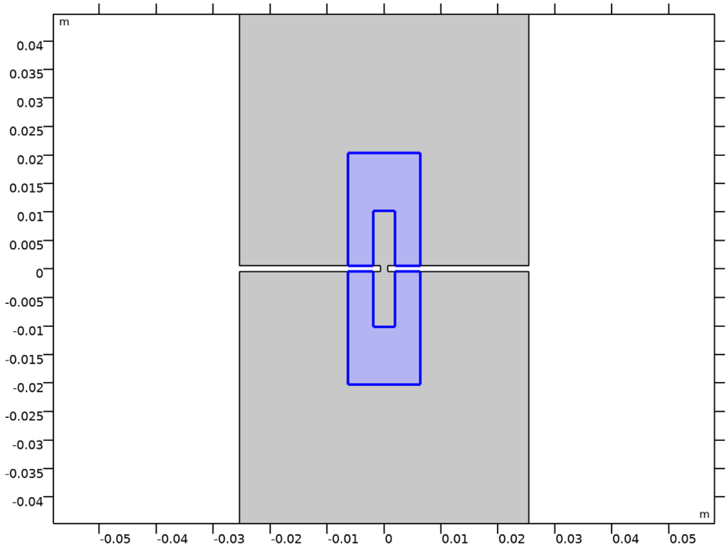
Size 2

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 5 and 6 only.



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

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `12*dvisc`.

Size 3

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 **Boundary**.

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

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

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `w_tube/8`.

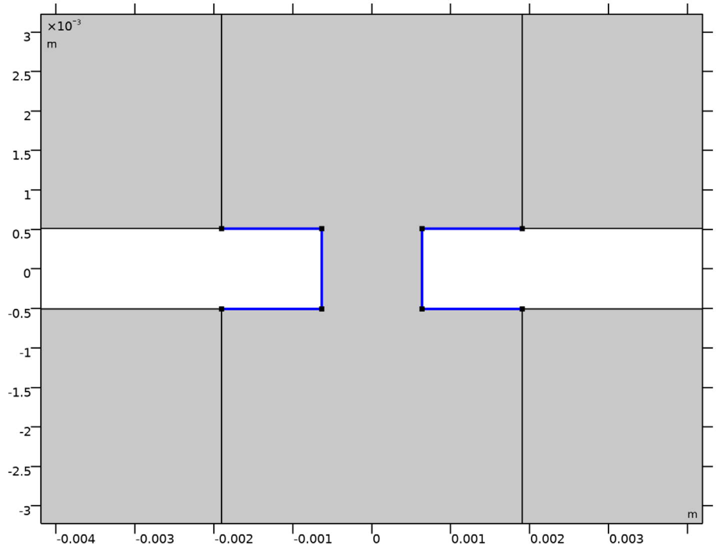
Size 4

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 **Boundary**.

4 Select Boundaries 19, 21, and 23–26 only.



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

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `dvisc`.

8 Click **Build All**.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

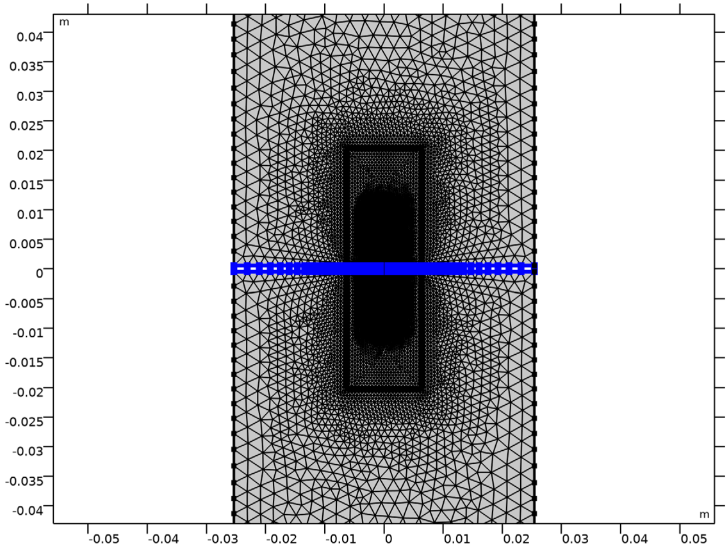
2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

3 Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundaries 5, 7, 13, 15, 19, 21, 23–26, 28, 30, 32, and 34 only.



3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

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

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `0.1*dvisc`.

7 Click **Build All**.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Driving frequency)	500 1000 1500 2000	Hz

Step 1: Time Dependent

1 In the **Model Builder** window, 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,T0/30,Tend)`.

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

Study the plots in the model. Switch between the excitation frequency values and step through the times. It is also useful to create an **Animation** to visualize the transient behavior in the model.

RESULTS

Surface 2

1 In the **Model Builder** window, expand the **Results>Acoustic Pressure (tatd)** node.

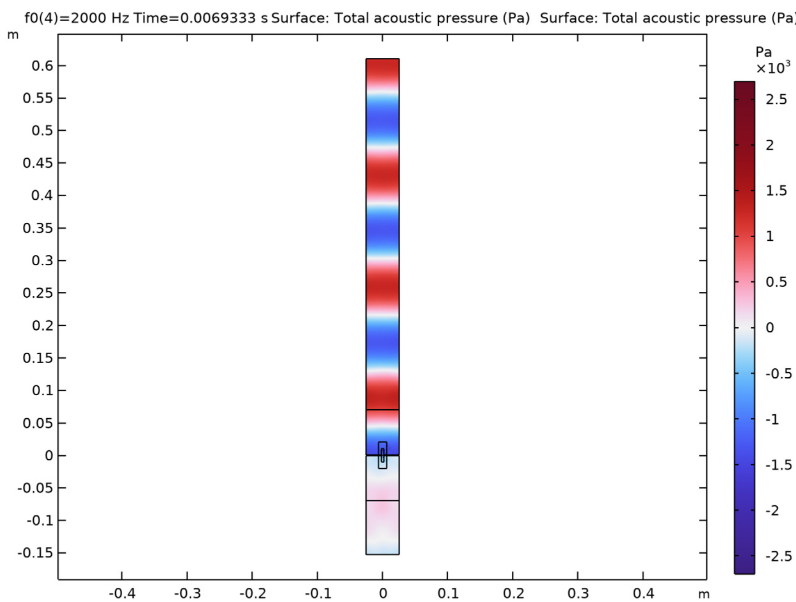
2 Right-click **Acoustic Pressure (tatd)** and choose **Surface**.

3 In the **Settings** window for **Surface**, locate the **Expression** section.

4 In the **Expression** text field, type `actd.p_t`.

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

6 In the **Acoustic Pressure (tatd)** toolbar, click **Plot**.



Acoustic Pressure (tatd)

1 In the **Model Builder** window, click **Acoustic Pressure (tatd)**.

2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Label**.

Acoustic Velocity (tatd)

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

2 In the **Settings** window for **2D Plot Group**, click to expand the **Selection** section.

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

4 From the **Selection** list, choose **Thermoviscous Acoustics**.

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

Arrow Surface 1

1 Right-click **Acoustic Velocity (tatd)** and choose **Arrow Surface**.

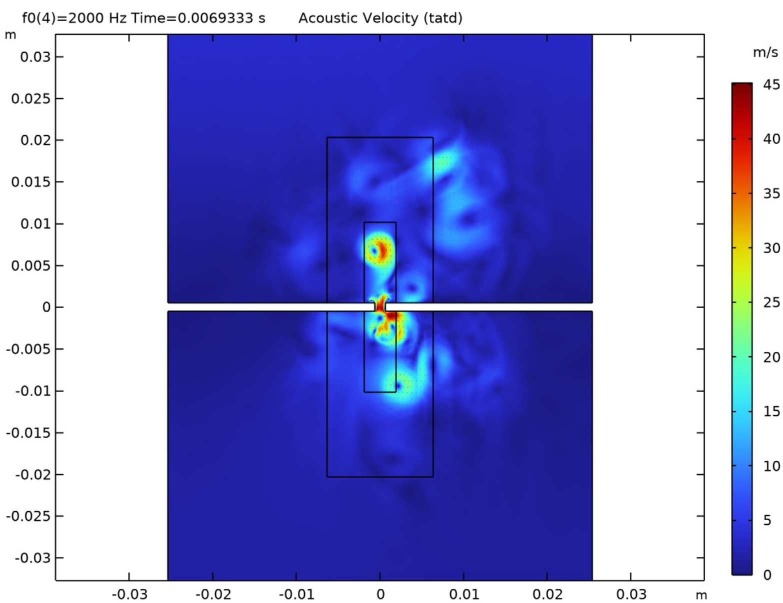
2 In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.

3 Find the **X grid points** subsection. In the **Points** text field, type 100.

4 Find the **Y grid points** subsection. In the **Points** text field, type 200.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

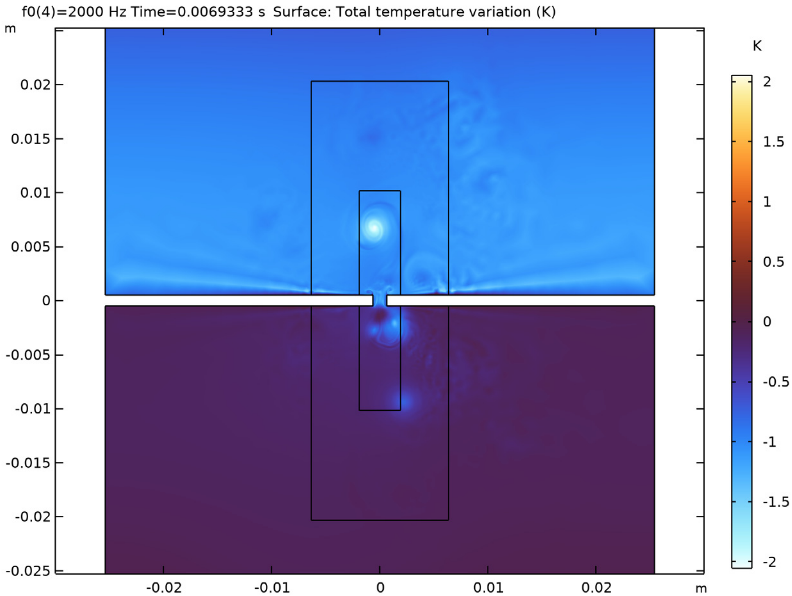
6 In the **Acoustic Velocity (tatd)** toolbar, click **Plot**.



Temperature Variation (tatd)

1 In the **Model Builder** window, under **Results** click **Temperature Variation (tatd)**.

2 In the **Temperature Variation (tatd)** toolbar, click **Plot**.



Acoustic Density Variations (tatd)

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

2 In the **Settings** window for **2D Plot Group**, type `Acoustic Density Variations (tatd)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

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

5 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 Right-click **Acoustic Density Variations (tatd)** and choose **Surface**.

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

3 In the **Expression** text field, type `tatd.rho`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

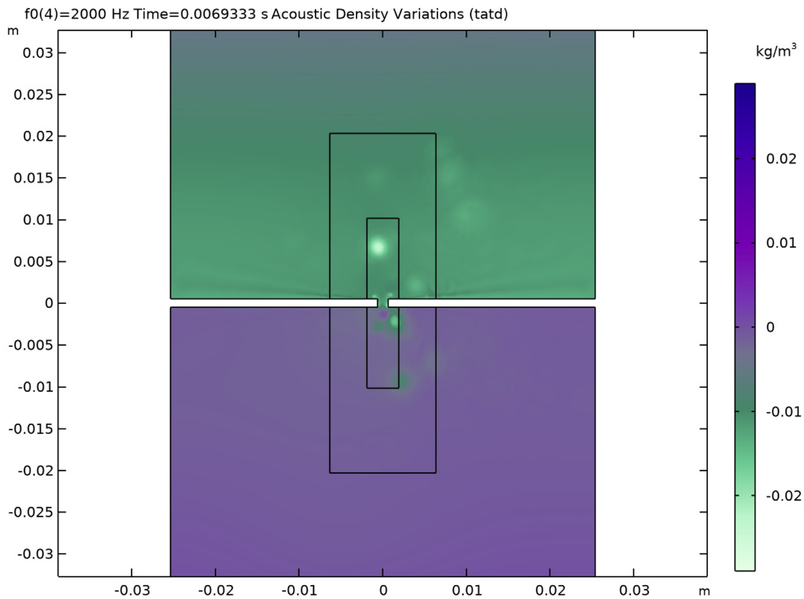
5 In the **Color Table** dialog box, select **Aurora>AuroraBorealis** in the tree.

6 Click **OK**.

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

8 From the **Scale** list, choose **Linear symmetric**.

9 In the **Acoustic Density Variations (tatd)** toolbar, click **Plot**.



log10 of Vorticity

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

2 In the **Settings** window for **2D Plot Group**, type `log10 of Vorticity` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

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

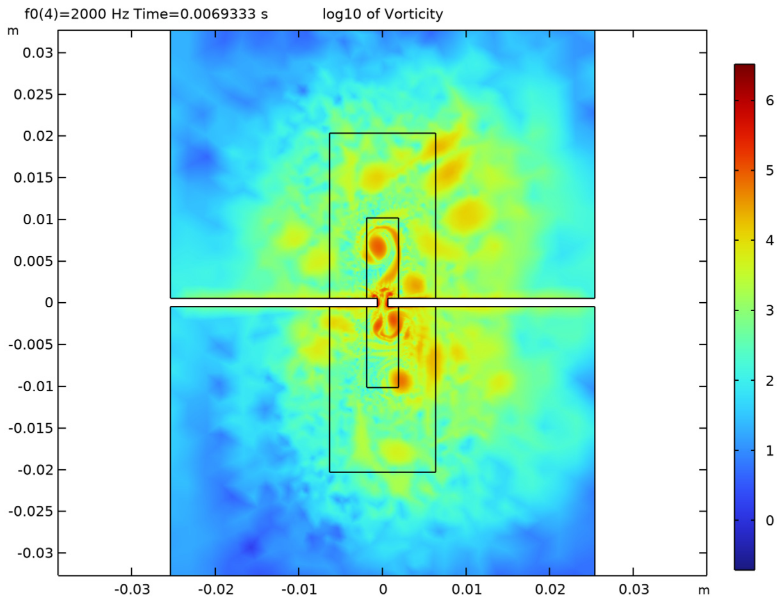
Surface 1

1 Right-click **log10 of Vorticity** and choose **Surface**.

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

3 In the **Expression** text field, type `log10(abs(uy-vx))`.

4 In the **log10 of Vorticity** toolbar, click **Plot**.



Proceed to create plots that analyze the pressure signal. All the plots are shown in the Results and Discussion section. Create a plot of the pressure in the slit and its FFT. Plot the incident and reflected wave signals. Compute the reflection coefficient and plot it as a function of the excitation frequency. Finally, plot the ratio of the maximum density variation to the equilibrium density.

Pressure in Slit

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 in Slit` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (f0)** list, choose **First**.

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

Point Graph 1

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

2 Select Point 16 only.

3 In the **Pressure in Slit** toolbar, click **Plot**.

Pressure at Slit FFT

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 at Slit FFT` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

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

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

Point Graph 1

1 Right-click **Pressure at Slit FFT** and choose **Point Graph**.

2 Select Point 16 only.

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

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

8 In the **Minimum** text field, type `100`.

9 In the **Maximum** text field, type `10000`.

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

11 In the **Pressure at Slit FFT** toolbar, click **Plot**.

12 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

The plot shows the FFT of the pressure response in the point.

Incident and Reflected

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

2 In the **Settings** window for **1D Plot Group**, type `Incident and Reflected` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (f0)** list, choose **First**.

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

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `t (s)`.

8 Select the **y-axis label** check box. In the associated text field, type `Pressure (Pa)`.

Point Graph 1

1 Right-click **Incident and Reflected** and choose **Point Graph**.

2 Select Point 6 only.

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

4 In the **Expression** text field, type `actd.p_i`.

5 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
`Incident`

Point Graph 2

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

2 Select Point 6 only.

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

4 In the **Expression** text field, type `actd.p_t-actd.p_i`.

5 Locate 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
`Reflected`

8 In the **Incident and Reflected** toolbar, click **Plot**.

Reflection Coefficient

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

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

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

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `f (Hz)`.

8 Select the **y-axis label** check box. In the associated text field, type `|R| (1)`.

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

10 In the **x minimum** text field, type `100`.

11 In the **x maximum** text field, type `2500`.

12 In the **y minimum** text field, type `0`.

Point Graph 1

1 Right-click **Reflection Coefficient** and choose **Point Graph**.

2 Select Point 6 only.

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

4 In the **Expression** text field, type `sqrt(timeint(Tstart,Tend-T0,(actd.p_t-actd.p_i)^2))/sqrt(timeint(Tstart,Tend-T0,actd.p_i^2))`.

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

6 In the **Reflection Coefficient** toolbar, click **Plot**.

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

8 From the **Width** list, choose **2**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

10 In the **Reflection Coefficient** toolbar, click **Plot**.

Linearity of Density Assumption

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

2 In the **Settings** window for **1D Plot Group**, type `Linearity of Density Assumption` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

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

Global 1

1 Right-click **Linearity of Density Assumption** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`maxop1(abs(tatd.rho_t/tatd.rho0))`	`1`	`Maximum 1`

4 In the **Linearity of Density Assumption** toolbar, click **Plot**.