

# 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  $p_{in}(t) = p_0 \cdot \sin(2\pi f_0 t)$  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  $p_{re}(t)$ . 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.

$$|R|^{2} = \left(\int_{T_{s}}^{T_{s}+4T_{0}} p_{re}(t)^{2} dt\right) / \left(\int_{T_{s}}^{T_{s}+4T_{0}} p_{in}(t)^{2} dt\right)$$

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.  $\rho_t$  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

- I 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 **M** Done.

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

## GLOBAL DEFINITIONS

Parameters I - Model

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

- **3** Locate the **Parameters** section. Click *b* 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

- I In the Home toolbar, click Pi 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 I

Rectangle 1 (r1)

- I 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)

- I 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)

- I 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)

- I 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)

- I 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 I (uniI)

- I In the Geometry toolbar, click 🔲 Booleans and Partitions and choose Union.
- **2** Select the object only.
- **3** Click the **Select All** button in the **Graphics** toolbar.
- 4 In the Settings window for Union, click 틤 Build Selected.

#### Delete Entities I (dell)

- I In the Model Builder window, right-click Geometry I 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 unil, select Domains 3, 7, 10, and 13–15 only.

Delete Entities 2 (del2)

- I Right-click Geometry I and choose Delete Entities.
- 2 On the object dell, select Boundaries 24 and 25 only.
- 3 In the Settings window for Delete Entities, click 🟢 Build All Objects.

#### ADD MATERIAL

- I 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

- I In the Model Builder window, expand the Component I (compl)>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

- I In the Definitions toolbar, click 🖣 Explicit.
- 2 In the Settings window for Explicit, type Thermoviscous Acoustics in the Label text field.
- 3 Clear all domains.
- 4 Click the 🔣 Select All button in the Graphics toolbar.
- 5 Select Domains 2, 3, and 5–7 only.

#### **Pressure Acoustics**

- I 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)

- I 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 I (maxop1)

- I 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)

- I In the Model Builder window, under Component I (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, we include N0 harmonics, defined in the parameters as 6.

4 Locate the Transient Solver and Mesh Settings section. In the  $f_{\text{max}}$  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.
- IO Click OK.
- **II** 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 I

- I 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

I 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.
- 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)

 In the Model Builder window, expand the Component I (compl)>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  $f_{\text{max}}$  text field, type N0\*f0.

Plane Wave Radiation 1

- I 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

- I 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)

- I In the Physics toolbar, click Automatic Multiphysics Couplings and choose Boundary>Acoustic-Thermoviscous Acoustic Boundary.
- 2 In the Settings window for Acoustic-Thermoviscous Acoustic Boundary, locate the Boundary Selection section.



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 I

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

Size

- I 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 I

- I In the Model Builder window, right-click Free Triangular I 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

- I Right-click Free Triangular I 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

- I Right-click Free Triangular I 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

- I Right-click Free Triangular I 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.



- **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

- I 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

I 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.



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 I

Parametric Sweep

- I 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

- I In the Model Builder window, click Step I: 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

- I 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)

I 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)

- I 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

- I 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)

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

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



Acoustic Density Variations (tatd)

- I 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 I/ Parametric Solutions I (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

- I 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.



#### f0(4)=2000 Hz Time=0.0069333 s Acoustic Density Variations (tatd)

#### log 10 of Vorticity

- I 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

- I 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 **9** 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

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Pressure in Slit in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/ Parametric Solutions I (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

- I Right-click Pressure in Slit and choose Point Graph.
- 2 Select Point 16 only.
- **3** In the **Pressure in Slit** toolbar, click **I** Plot.

Pressure at Slit FFT

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Pressure at Slit FFT in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/ Parametric Solutions I (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

- I 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.
- II In the **Pressure at Slit FFT** toolbar, click **O** 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

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Incident and Reflected in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/ Parametric Solutions I (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

- I 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

- I 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**

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Reflection Coefficient in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/ Parametric Solutions I (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.
- **IO** In the **x minimum** text field, type 100.
- II In the x maximum text field, type 2500.
- **12** In the **y minimum** text field, type **0**.

#### Point Graph 1

- I 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-TO,(actd.p\_t-actd.p\_i)^2))/sqrt(timeint(Tstart,Tend-TO,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.
- **IO** In the **Reflection Coefficient** toolbar, click **O Plot**.

#### Linearity of Density Assumption

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Linearity of Density Assumption in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/ Parametric Solutions I (sol2).
- 4 Click to expand the Title section. From the Title type list, choose Label.

#### Global I

- I 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
<pre>maxop1(abs(tatd.rho_t/tatd.rho0))</pre>	1	Maximum 1

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