Created in COMSOL Multiphysics 6.2

Nonlinear Transfer Impedance of a Tapered Orifice

Introduction

This model analyzes the nonlinear transfer impedance of a tapered orifice that can be part of a perforate or microperforated plate (MPP). The analysis is carried out for various degrees of tapering of the perforate and for a frequency range.

A linear analysis is first set up in the frequency domain using the Thermoviscous Acoustics, Frequency Domain interface as well as in the time domain using the Thermoviscous Acoustics, Transient interface (for the time domain, only a few selected frequencies for comparison). A full nonlinear analysis is finally carried out in the time domain using the Thermoviscous Acoustics, Transient interface and the Nonlinear Thermoviscous Acoustics Contributions feature.

The results are compared with analytical and semi-analytical models in the simple straight cylindrical perforate configuration (circular orifice).

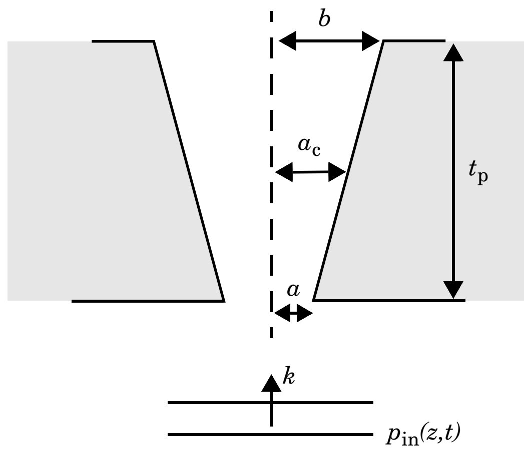


Figure 1: Schematic representation of the tapered orifice in the microperforated plate.

Model Definition

In this 2D axisymmetric model, a plane wave is incident normally on a microperforated plate (MPP) with a tapered orifice as, depicted schematically in Figure 1. The wave is propagating in the positive axial z direction, and has the form

$$p_{in}(z,t) = \sin(2\pi f_0 t - kz) \tag{1}$$

where k is the wavenumber and f_0 the excitation frequency. The top radius is b, the orifice center radius is a_c (0.4 mm in the model), the bottom radius is a, the taper ratio is a/b (the parameter ab in the model ranging from 0.5 to 1), and t_p is the plate thickness (1 mm). In the model, the corners of the orifice include a small fillet of radius $a_c/50$ in order to avoid any singularities at sharp edges.

An in dept experimental and analytical analysis of the loss mechanisms and loss regimes in a microperforated plate is given in Temiz and others (Ref. 1), for the nontapered case when the taper ratio is 1. Some details or the modeling background are given in Ref. 2. In the linear regime, all losses are due to viscous dissipation in the Stokes layer (viscous boundary layer), which has the characteristic thickness

$$\delta_v = \sqrt{\frac{\mu}{\omega\rho_0}} \tag{2}$$

where μ is the dynamic viscosity, ω is the angular frequency, and ρ_0 is the density. At 100 Hz in air, this characteristic viscous length is 0.22 mm. The ratio of the Stokes layer to the characteristic dimensions of the hole (the diameter $d = 2a$) is given by the Shear number

$$\mathrm{Sh} = \frac{d}{2\delta_v} = d\sqrt{\frac{\omega\rho_0}{4\mu}} \tag{3}$$

In an MPP, the Stokes layer spans almost the entire perforate such that $\mathrm{Sh} = O(1)$. Two cases can the be defined (Ref. 1):

- The excitation is low and the transfer impedance is linear and only depends on the excitation frequency and the perforation geometry.
- The excitation amplitude is larger than a given critical value and vortices start to form at sharp edges (local nonlinear effects). This leads to additional resistive losses.

To determine when vortices start to appear, the nondimensional Strouhal number Sr is introduced. It is defined as

$$\mathrm{Sr} = \frac{\omega d}{|\hat{u}_p|} \tag{4}$$

where $|\hat{u}_p|$ is the absolute value of the averaged acoustic particle velocity in the center of the orifice. A large Strouhal number defines a regime where the particle displacement is small compared to the orifice diameter and no vortices will appear. For very small Strouhal numbers vortices are generated and convected away from the perforate. In between these two limits when $\mathrm{Sr} = O(1)$, local vortices appear near the sharp edges of the orifice

(Ref. 1). In Temiz and others (equations 13 to 15 in Ref. 1), semi-analytical nonlinear corrections to the linear transfer impedance are given. These expressions are implemented as variables in the model under the **Definitions** node. An analytical expression for the linear transfer impedance is also set up; more details can be found in the Application Library model Transfer Impedance of a Perforate, as well as in the documentation for the **Interior Perforated Plate** feature in the *Acoustics Module User's Guide*. The necessary time-averaged quantities that enter the analytical expressions are computed using the `timeint()` operator for integration in time, as well as spatial integration coupling operators.

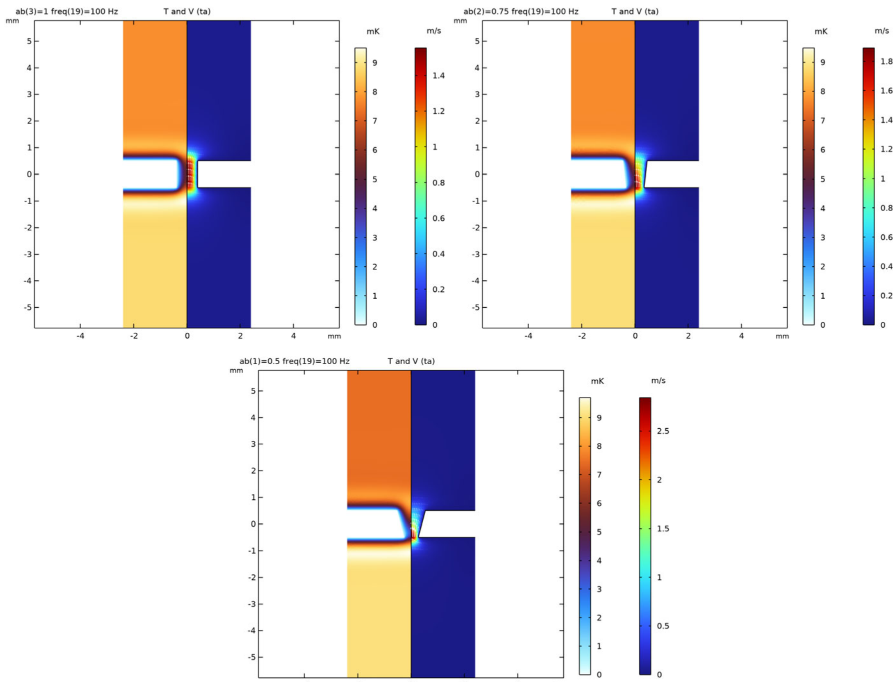


Figure 2: Acoustic temperature variation (left half) and acoustic particle velocity (right half) for the linear frequency domain model at 100 Hz. Showing the three tapering configurations with a/b set to 1, 0.75, and 0.5.

Results and Discussion

The temporal evolution of the acoustic (perturbation) fields are best visualized using an **Animation** in the user interface. As an example, acoustic temperature variation and the acoustic particle velocity is shown side by side in Figure 2 and Figure 3. In the first figure the (instantaneous) values are shown for the frequency domain simulation at 100 Hz for the three tapering configurations a/b equal to 1, 0.75, and 0.5. The time dependency is here given by the usual $\exp(i\omega t)$ and no nonlinear effects can be seen. In Figure 3 the time evolution from t = 0.045 s to 0.048 s is illustrated for the same 100 Hz excitation and for a/b equal to 0.5. The sequence of 4 images clearly shows local vortex shedding. Note that the vortex is highly damped as it moves away from the orifice, this is due to the chosen coarser mesh and the numerical stabilization. It is the dissipated energy in the orifice that dictates the transfer impedance magnitude.

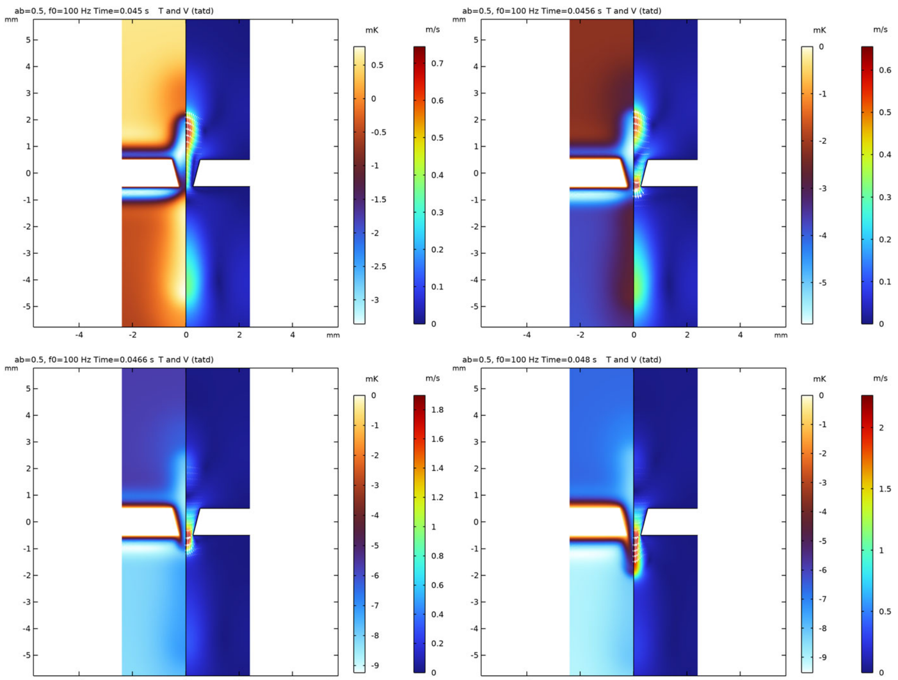


Figure 3: Time evolution (from t = 0.045 s to 0.048 s) for the acoustic temperature variation (left half) and acoustic particle velocity (right half) for the nonlinear time domain simulation for the 100 Hz excitation and for a/b equal to 0.5.

The model implements and compares the COMSOL simulation results with an analytical linear and a semi-analytical nonlinear transfer impedance model. Some of the important results are given in the following Figure 4 to Figure 7.

The linear analytical model (see **Definitions > Variables 1 - Linear Analytical Model**) is first compared to the (linear) frequency domain results for the straight pipe configuration (a/b = 1). Figure 4 shows the real, imaginary, and absolute values of the transfer impedance as function of frequency. The results show good correlation.

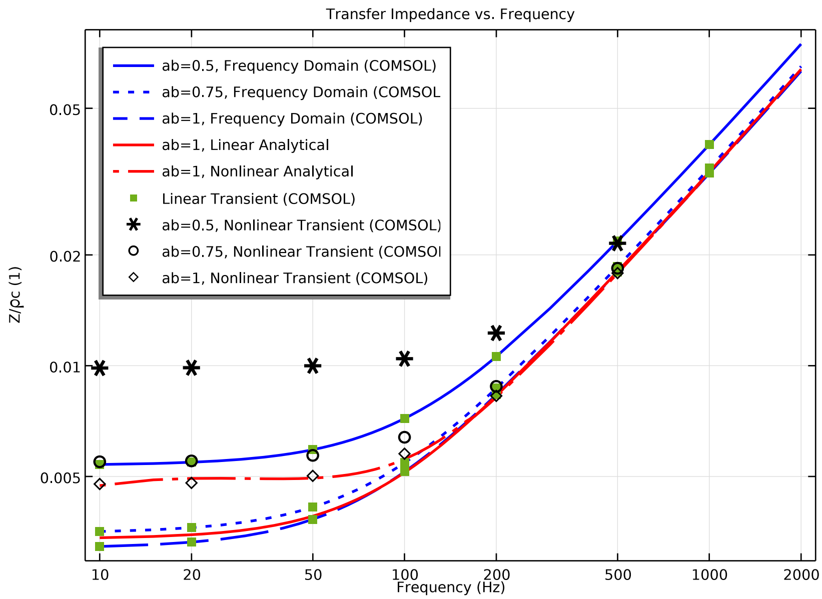


Figure 4: Comparison of the simulated and analytical linear transfer impedance as function of frequency. Image showing the real, imaginary, and absolute value for the straight orifice configuration.

A comparison between all the transfer impedance models is shown in Figure 5 and Figure 6, as function of frequency and Strouhal Sr number, respectively. The figures show the analytical linear and analytical nonlinear (see **Definitions > Variables 3 - Nonlinear Analytical Model**) models for the straight pipe configuration (no analytical models exist for the tapered configurations). The figure also shows simulation results for the linear frequency domain and time domain, as well as the nonlinear time domain simulation results, all for the three tapering cases (a/b = 1, 0.75, and 0.5). The comparison between the linear frequency domain and time domain results is carried out to validate the computation procedure for the time domain transfer impedance. The results agree perfectly. The nonlinear time domain simulations results are seen to agree well with the semi analytical model from Ref. 1 (straight pipe configuration), validating the simulation method based on the *Nonlinear Thermoviscous Acoustic* setup. The method can be used to predict the linear and nonlinear acoustic behavior of any perforate configuration and geometry. As described above the nonlinear results deviate from the linear results for small Strouhal numbers (see Figure 6) where the dynamics exhibit stronger (local) nonlinear behavior, here local vortices are generated and shed from the orifice edges.

Finally, Figure 7 sows the relation between the Shear number Sh and Strouhal number Sr, for the range of studied frequencies (from 10 Hz to 500 Hz), for the frequency domain model.

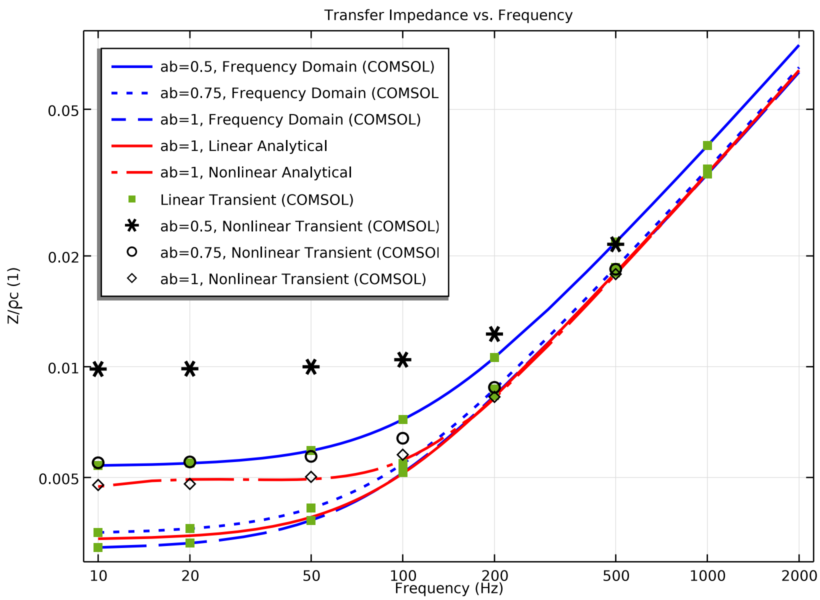


Figure 5: Comparison of all the analytical and numerical simulation results. Analytical linear and nonlinear models for a/b = 1, simulation results sowing the linear frequency and time domain (for a/b = 1, 0.75, and 0.5), an the nonlinear time domain results (for a/b = 1, 0.75, and 0.5). Shown as function of frequency.

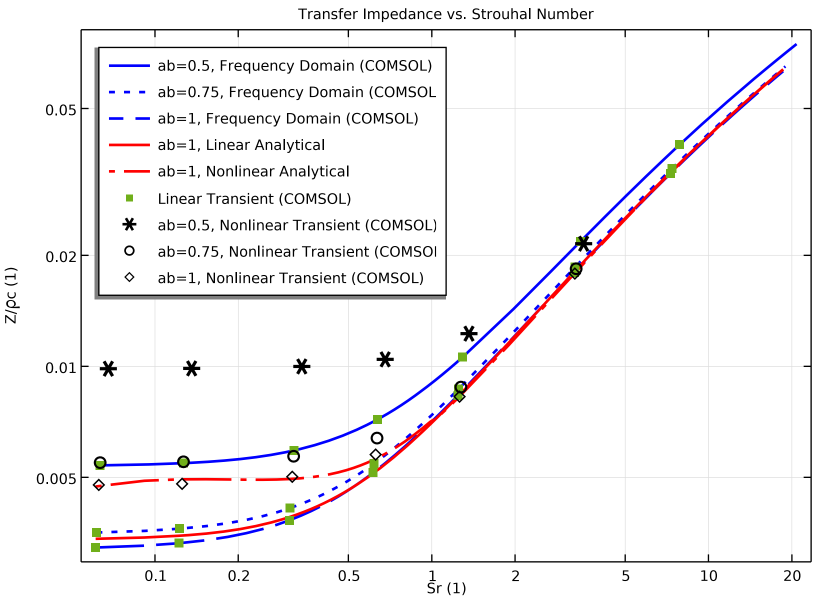


Figure 6: Comparison of all the analytical and numerical simulation results. Analytical linear and nonlinear models for a/b = 1, simulation results sowing the linear frequency and time domain (for a/b = 1, 0.75, and 0.5), an the nonlinear time domain results (for a/b = 1, 0.75, and 0.5). Shown as function of Strouhal number.

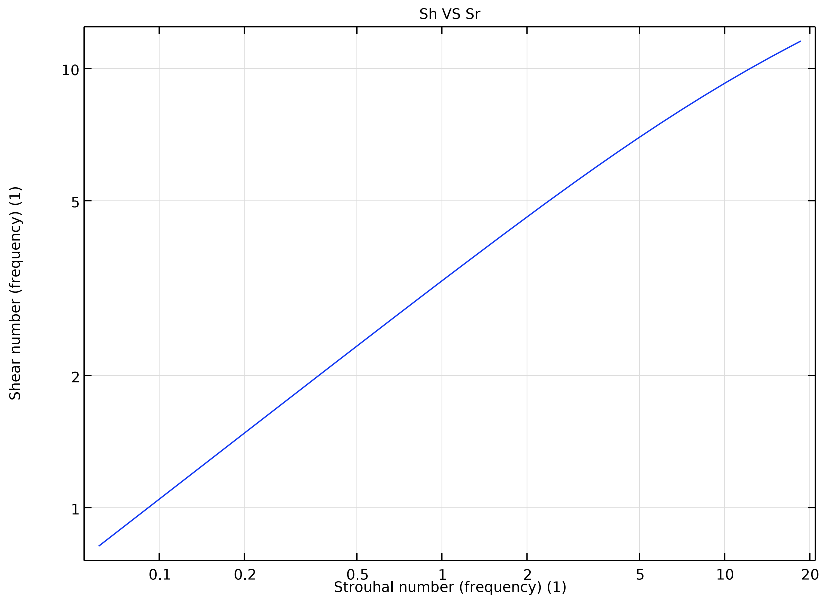


Figure 7: Shear number Sh versus Strouhal number Sr for all the studied frequencies from 10 Hz to 500 Hz, for the linear frequency domain model.

References

1. M.A. Temiz, J. Tournadre, I.L. Arteaga, and A. Hirschberg, "Non-linear acoustic transfer impedance of micro-perforated plates with circular orifices," *J. Sound Vib.*, vol. 366, pp. 418–428, 2016.

2. M. Herring Jensen, "Simulation of Perforates: An Application of Nonlinear Thermoviscous Acoustics," *Acoustics Today*, vol. 17, Summer 2021.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/nonlinear_transfer_impedance`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Frequency Domain (ta)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Transient (tatd)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Frequency Domain**.
10 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1 - Geometry

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Parameters 1 - 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_transfer_impedance_geometry_parameters.txt`.

Parameters 2 - Model

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.
2 In the **Settings** window for **Parameters**, type `Parameters 2 - 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_transfer_impedance_model_parameters.txt`.

GEOMETRY 1

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

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

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

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (mm)	z (mm)
0	-tp/2
a	-tp/2
b	tp/2
0	tp/2
0	-tp/2

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **r** text field, type `ac`.

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 `NR*ac`.

4 In the **Height** text field, type `NH*ac`.

5 Locate the **Position** section. In the **z** text field, type `tp/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 `NR*ac`.

4 In the **Height** text field, type `NH*ac`.

5 Locate the **Position** section. In the **z** text field, type `tp/2-NH*ac-tp`.

Union 1 (uni1)

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

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

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **uni1**, select Boundaries 4 and 8 only.

Fillet 1 (fil1)

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

2 On the object **del1**, select Points 6 and 8 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `r0`.

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 `NR*ac`.

4 In the **Height** text field, type `NH*ac`.

5 Locate the **Position** section. In the **z** text field, type `tp/2+NH*ac`.

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 `NR*ac`.

4 In the **Height** text field, type `NH*ac`.

5 Locate the **Position** section. In the **z** text field, type `tp/2-NH*ac-tp-NH*ac`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 Click to select the **Activate Selection** toggle button for **Start vertex**.

4 On the object **fil1**, select Point 2 only.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **r** text field, type `a`.

7 In the **z** text field, type `-tp/2`.

Line Segment 3 (ls3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **fil**, select Point 4 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 From the **Specify** list, choose **Coordinates**.

5 In the **r** text field, type `b`.

6 In the **z** text field, type `tp/2`.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

DEFINITIONS

Variables 1 - Linear Analytical Model

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

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, type `Variables 1 - Linear Analytical Model` in the **Label** text field.

4 Locate the **Variables** section. Click **Load from File**.

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

Variables 2 - Material Parameters

1 Right-click **Definitions** and choose **Variables**.

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

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

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

Variables 3 - Nonlinear Analytical Model

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables 3 - Nonlinear Analytical Model` in the **Label** text field.

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

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

Variables 4 - Time Domain Impedance

1 Right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables 4 - Time Domain Impedance` in the **Label** text field.

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

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

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, type `intop_mid` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 8 only.

Integration 2 (intop2)

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

2 In the **Settings** window for **Integration**, type `intop_in` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6 and 15 only.

Integration 3 (intop3)

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

2 In the **Settings** window for **Integration**, type `intop_out` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 10 and 17 only.

Integration 4 (intop4)

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

2 In the **Settings** window for **Integration**, type `intop_pnt` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Point 11 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Air**.

3 Right-click and choose **Add to Component 1 (comp1)**.

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

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Select Domains 2 and 3 only.

Wall 2

1 Right-click **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** and choose **Wall**.

2 Select Boundaries 19 and 20 only.

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

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

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

Port 1

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Plane wave**.

5 Locate the **Incident Mode Settings** section. In the A^{in} text field, type `pin`.

Port 2

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

2 Select Boundary 12 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Plane wave**.

THERMOVISCOUS ACOUSTICS, TRANSIENT (TATD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Transient (tatd)**.
2 Select Domains 2 and 3 only.
3 Click the **Show More Options** button in the **Model Builder** toolbar.
4 In the **Show More Options** dialog box, select **Physics>Stabilization** in the tree.
5 In the tree, select the check box for the node **Physics>Stabilization**.
6 Click **OK**.
7 In the **Settings** window for **Thermoviscous Acoustics, Transient**, click to expand the **Stabilization** section.
8 From the **Stabilization method** list, choose **Galerkin least-squares (GLS) stabilization**.
9 Locate the **Transient Solver and Mesh Settings** section. In the f_{max} text field, type `f0`.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.
2 Select Boundaries 19 and 20 only.
3 In the **Settings** window for **Wall**, locate the **Mechanical** section.
4 From the **Mechanical condition** list, choose **Slip (perfect)**.
5 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

Nonlinear Thermoviscous Acoustics Contributions 1

1 In the **Physics** toolbar, click **Domains** and choose **Nonlinear Thermoviscous Acoustics Contributions**.
2 Select Domains 2 and 3 only.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.
2 Select Domains 1 and 4 only.
3 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Transient Solver and Mesh Settings** section.
4 In the f_{max} text field, type `f0`.

Plane Wave Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.
2 Select Boundaries 2 and 13 only.

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 `pin`.
4 From the c list, choose **From material**.
5 From the **Material** list, choose **Air (mat1)**.
6 Specify the $\mathbf{e}_k$ vector as

1	z

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

MULTIPHYSICS

Proceed to set up the **Multiphysics Coupling** that couples the **Pressure Acoustics, Transient (actd)** and the **Thermoviscous Acoustics, Transient (tatd)**.

Acoustic–Thermoviscous Acoustic Boundary 1 (atb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic–Thermoviscous Acoustic Boundary**.
2 Select Boundaries 4 and 12 only.
3 In the **Settings** window for **Acoustic–Thermoviscous Acoustic Boundary**, locate the **Coupled Interfaces** section.
4 From the **Thermoviscous acoustics** list, choose **Thermoviscous Acoustics, Transient (tatd)**.

MESH 1

In this model, the mesh is set up manually. Proceed by directly adding the desired mesh components.

Corner Refinement 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Corner Refinement**.
2 In the **Settings** window for **Corner Refinement**, locate the **Boundary Selection** section.
3 Click to select the **Activate Selection** toggle button.
4 Select Boundaries 14–17, 22, and 23 only.

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.

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, use the orifice bottom radius as the maximum element size because it is the limiting distance. It is much smaller than the smallest wavelength corresponding to the maximum study frequency.

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

5 In the **Minimum element size** text field, type `4.8E-6`.

6 In the **Maximum element growth rate** text field, type `1.2`.

Free Triangular 1

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

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 5–8 and 10 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 `a/7`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

4 Select Domains 2 and 3 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 `a/3`.

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 Select Boundaries 14, 16, 22, and 23 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`.

Boundary Layers 1

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

Boundary Layer Properties

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

2 Select Boundaries 14–17, 22, and 23 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 - FREQUENCY DOMAIN

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

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

3 Clear the **Generate default plots** check box.

Turn off the generation of default plots for each study. If turned on all the default plots for each physics interface will be generated. In the following steps, some of the default plots will be added and slightly changed after the computation.

4 In the **Label** text field, type `Study 1 - Frequency Domain`.

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
ab (Ratio a/b)	`0.5 0.75 1`	

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type 10 in the **Start** text field.

5 In the **Step** text field, type 5.

6 In the **Stop** text field, type 95.

7 Click **Add**.

8 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

9 Click **Range**.

10 In the **Range** dialog box, type 100 in the **Start** text field.

11 In the **Step** text field, type 10.

12 In the **Stop** text field, type 190.

13 Click **Add**.

14 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

15 Click **Range**.

16 In the **Range** dialog box, type 200 in the **Start** text field.

17 In the **Step** text field, type 100.

18 In the **Stop** text field, type 2000.

19 Click **Add**.

20 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

21 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Thermoviscous Acoustics, Frequency Domain (ta)	√	Automatic (Frequency domain)
Thermoviscous Acoustics, Transient (tatd)		Automatic (Time dependent)
Pressure Acoustics, Transient (actd)		Automatic (Time dependent)

22 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)		Automatic (Frequency domain)

Solution 1 (sol1)

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

ADD STUDY

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 Click **Add Study** in the window toolbar.

6 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 - TIME DOMAIN LINEAR

1 In the **Settings** window for **Study**, type `Study 2 - Time Domain Linear` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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
ab (Ratio a/b)	0.5 0.75 1	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Frequency)	10 20 50 100 200 500 1000	Hz

7 From the **Sweep type** list, choose **All combinations**.

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 Click **Range**.

4 In the **Range** dialog box, type `T0` in the **Step** text field.

5 In the **Stop** text field, type `3*T0`.

6 Click **Replace**.

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

8 Click **Range**.

9 In the **Range** dialog box, type `4*T0` in the **Start** text field.

10 In the **Step** text field, type `T0/50`.

11 In the **Stop** text field, type `5*T0`.

12 Click **Add**.

13 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

14 Select the **Modify model configuration for study step** check box.

15 In the tree, select **Component 1 (comp1)>Thermoviscous Acoustics, Transient (tatd)> Nonlinear Thermoviscous Acoustics Contributions 1**.

16 Right-click and choose **Disable**.

Solution 2 (sol2)

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

STUDY 3 - TIME DOMAIN NONLINEAR

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - Time Domain Nonlinear` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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
ab (Ratio a/b)	`0.5 0.75 1`	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0 (Frequency)	10 20 50 100 200 500	Hz

7 From the **Sweep type** list, choose **All combinations**.

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 Click **Range**.

4 In the **Range** dialog box, type `T0` in the **Step** text field.

5 In the **Stop** text field, type `3*T0`.

6 Click **Replace**.

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

8 Click **Range**.

9 In the **Range** dialog box, type `4*T0` in the **Start** text field.

10 In the **Step** text field, type `T0/50`.

11 In the **Stop** text field, type `5*T0`.

12 Click **Add**.

Solution 3 (sol3)

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

STUDY 1 - FREQUENCY DOMAIN

Click **Compute**.

STUDY 2 - TIME DOMAIN LINEAR

Click **Compute**.

STUDY 3 - TIME DOMAIN NONLINEAR

Click **Compute**.

RESULTS

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

First, datasets need to be defined to better analyze some of the results: the temperature and velocity for the linear and the nonlinear cases, the acoustic pressure and temperature for the **Thermoviscous Acoustics, Transient** case and for the 3D velocity representation. The three mirror datasets depend on the three corresponding studies' results.

Mirror 2D - Frequency

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More 2D Datasets>Mirror 2D**.

3 In the **Settings** window for **Mirror 2D**, type `Mirror 2D - Frequency` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Frequency Domain/ Parametric Solutions 1 (sol4)**.

Mirror 2D - Linear

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

2 In the **Settings** window for **Mirror 2D**, type `Mirror 2D - Linear` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Time Domain Linear/ Parametric Solutions 2 (sol8)**.

Mirror 2D - Nonlinear

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

2 In the **Settings** window for **Mirror 2D**, type `Mirror 2D - Nonlinear` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

Revolution 2D 1

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

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

3 From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

4 Click to expand the **Revolution Layers** section. In the **Revolution angle** text field, type `130`.

ADD PREDEFINED PLOT

1 In the **Results** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1 - Frequency Domain/Parametric Solutions 1 (sol4)>Thermoviscous Acoustics, Frequency Domain>Acoustic Pressure (ta)**.

4 Click **Add Plot** in the window toolbar.

5 In the **Results** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Acoustic Pressure (ta)

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

2 From the **Parameter value (freq (Hz))** list, choose **200**.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

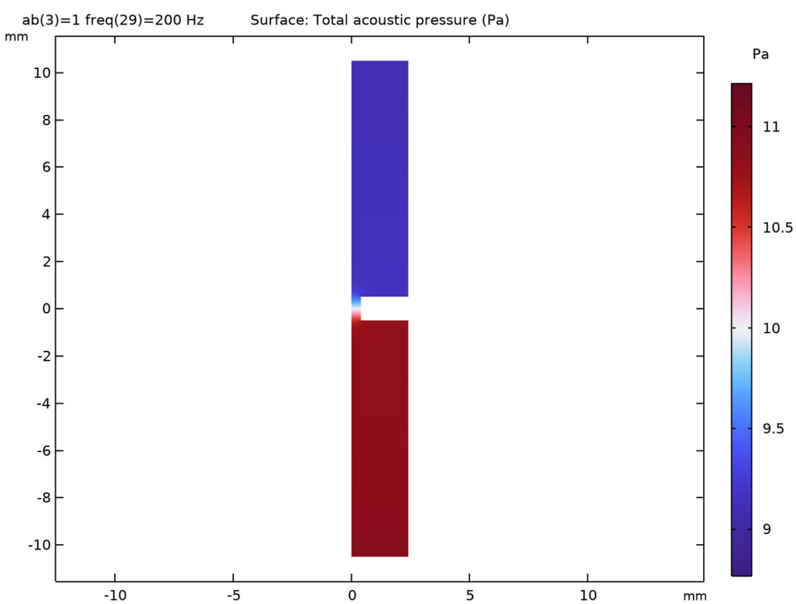
Surface

1 In the **Model Builder** window, expand the **Acoustic Pressure (ta)** node, then click **Surface**.

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

3 From the **Scale** list, choose **Linear**.

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



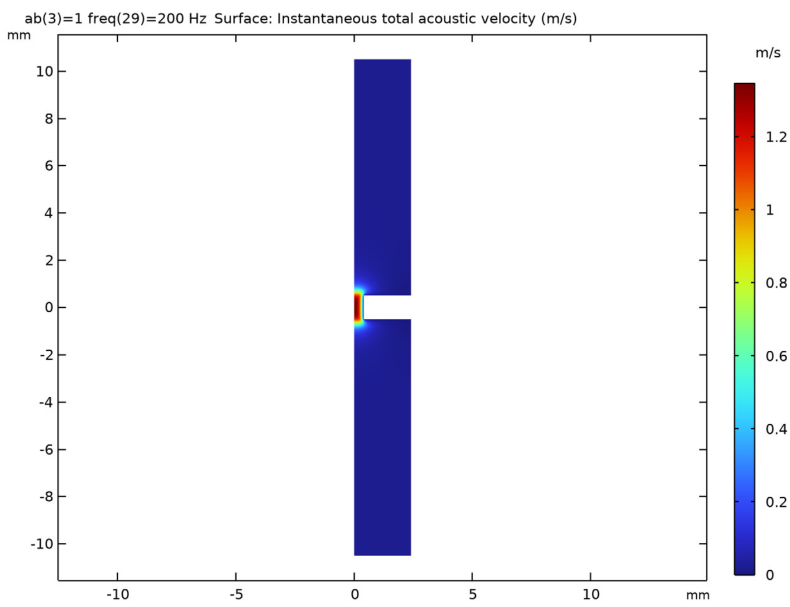
ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.
2 Go to the **Add Predefined Plot** window.
3 In the tree, select **Study 1 - Frequency Domain/Parametric Solutions 1 (sol4)>Thermoviscous Acoustics, Frequency Domain>Acoustic Velocity (ta)**.
4 Click **Add Plot** in the window toolbar.
5 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Acoustic Velocity (ta)

1 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
2 From the **Parameter value (freq (Hz))** list, choose **200**.
3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.
4 In the **Acoustic Velocity (ta)** toolbar, click **Plot**.



Temperature (ta)

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

2 In the **Settings** window for **2D Plot Group**, type `Temperature (ta)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Frequency Domain/ Parametric Solutions 1 (sol4)**.

4 From the **Parameter value (freq (Hz))** list, choose **200**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

Surface 1

1 In the **Temperature (ta)** toolbar, click **Surface**.

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

3 In the **Expression** text field, type `ta.T_t`.

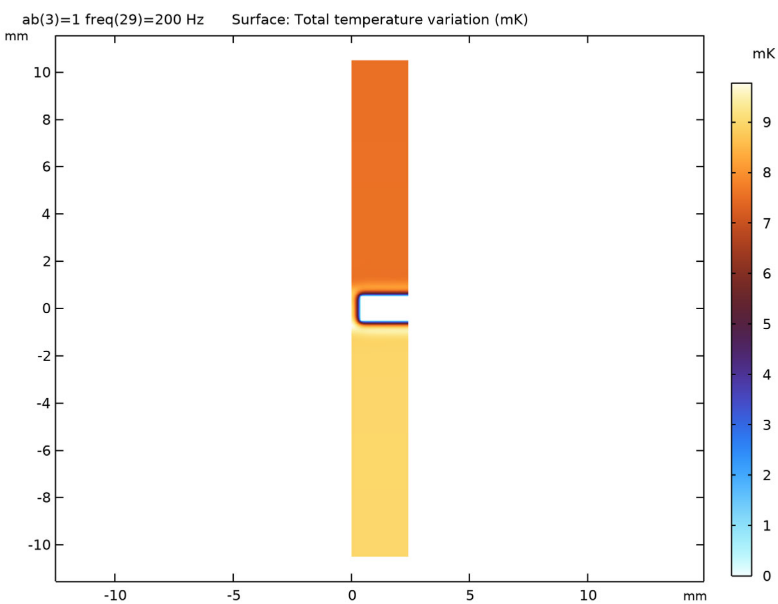
4 From the **Unit** list, choose **mK**.

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

6 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.

7 Click **OK**.

8 In the **Temperature (ta)** toolbar, click **Plot**.



T and V (ta)

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

2 In the **Settings** window for **2D Plot Group**, type `T and V (ta)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1 - Frequency Domain/ Parametric Solutions 1 (sol4)**.

4 From the **Parameter value (ab)** list, choose **0.5**.

5 From the **Parameter value (freq (Hz))** list, choose **100**.

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

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

Surface 1

1 In the **T and V (ta)** toolbar, click **Surface**.

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

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

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Expression** section. In the **Expression** text field, type `ta.T_t`.

6 From the **Unit** list, choose **mK**.

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

8 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.

9 Click **OK**.

T and V (ta)

In the **T and V (ta)** toolbar, click **Surface**.

Surface 2

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

2 In the **Expression** text field, type `ta.v_inst`.

T and V (ta)

In the **T and V (ta)** toolbar, click **Arrow Surface**.

Arrow Surface 1

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

2 Find the **R grid points** subsection. From the **Entry method** list, choose **Coordinates**.

3 Click **Range**.

4 In the **Range** dialog box, type `0` in the **Start** text field.

5 In the **Step** text field, type `0.06`.

6 In the **Stop** text field, type `0.6`.

7 Click **Replace**.

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

9 Find the **Z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

10 Click **Range**.

11 In the **Range** dialog box, type `-1.2` in the **Start** text field.

12 In the **Step** text field, type `0.24`.

13 In the **Stop** text field, type `1.2`.

14 Click **Replace**.

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

16 Select the **Scale factor** check box. In the associated text field, type `0.15`.

17 From the **Color** list, choose **White**.

Selection 1

1 In the **T and V (ta)** toolbar, click **Selection**.

2 Select Domains 2 and 3 only.

T and V (ta)

In the **T and V (ta)** toolbar, click **Line**.

Line 1

1 In the **Settings** window for **Line**, locate the **Expression** section.

2 In the **Expression** text field, type `1`.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Black**.

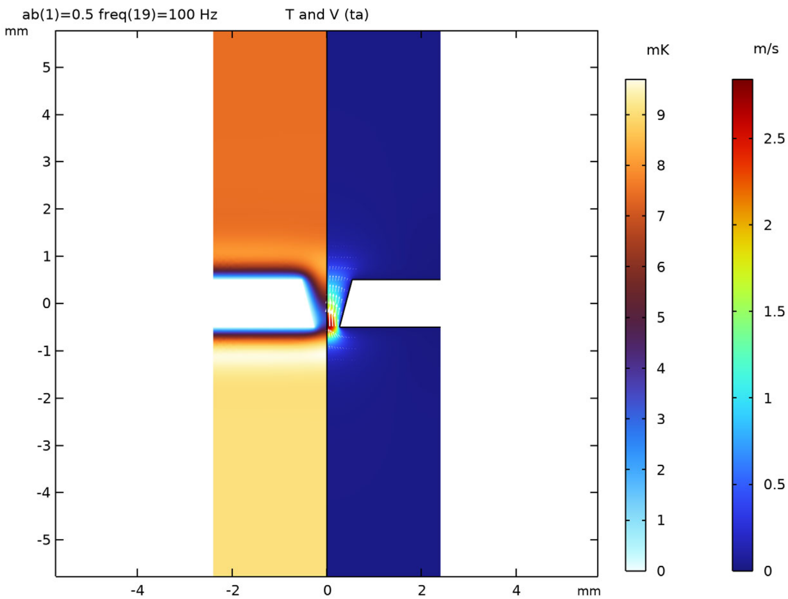
Selection 1

1 In the **T and V (ta)** toolbar, click **Selection**.

2 Select Boundaries 3, 5, 7, 9, 14–17, 22, and 23 only.

3 In the **T and V (ta)** toolbar, click **Plot**.

The instantaneous acoustic temperature fluctuations and the particle velocity of the linear (frequency domain) model at f = 100 Hz, should look like the following figure.



Transfer Impedance (Frequency Domain)

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

2 In the **Settings** window for **1D Plot Group**, type `Transfer Impedance (Frequency Domain)` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

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

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

7 Locate the **Axis** section. Select the **x-axis log scale** check box.

8 Select the **y-axis log scale** check box.

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

Global 1

1 In the **Transfer Impedance (Frequency Domain)** toolbar, click **Global**.

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

3 From the **Dataset** list, choose **Study 1 - Frequency Domain/Parametric Solutions 1 (sol4)**.

4 From the **Parameter selection (ab)** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`real(Ztrans)`	1	`COMSOL, real(Z)`
`imag(Ztrans)`	1	`COMSOL, imag(Z)`
`abs(Ztrans)`	1	`COMSOL, abs(Z)`

6 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

Transfer Impedance (Frequency Domain)

In the **Transfer Impedance (Frequency Domain)** toolbar, click **Global**.

Global 2

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 1 - Frequency Domain/Solution 1 (sol1)**.

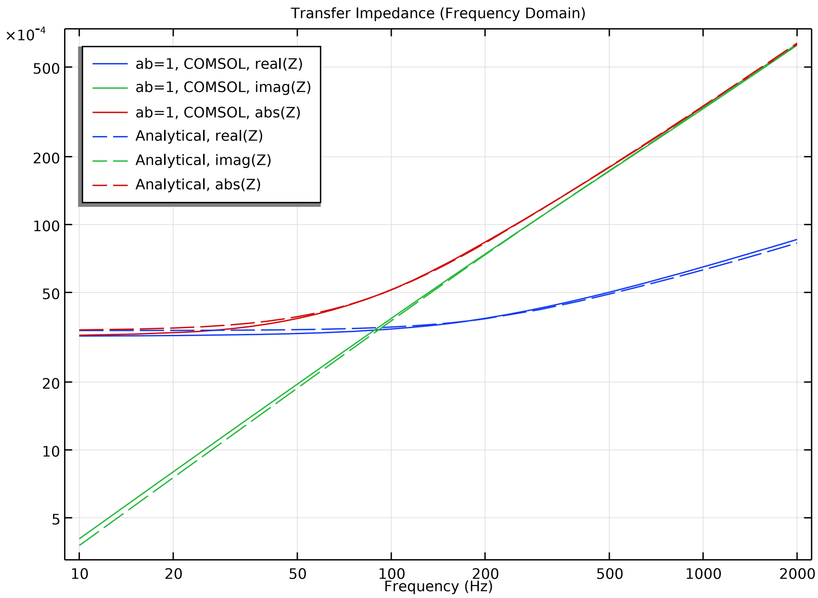
3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`real(Zlin)`	1	`Analytical, real(Z)`
`imag(Zlin)`	1	`Analytical, imag(Z)`
`abs(Zlin)`	1	`Analytical, abs(Z)`

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

6 In the **Transfer Impedance (Frequency Domain)** toolbar, click **Plot**.



Transfer Impedance vs. Frequency

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

2 In the **Settings** window for **1D Plot Group**, type `Transfer Impedance vs. Frequency` in the **Label** text field.

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

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

5 Locate the **Plot Settings** section.

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

7 Select the **y-axis label** check box. In the associated text field, type `Z/\rho c (1)`.

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

9 Select the **y-axis log scale** check box.

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

Global 1

1 In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

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

3 From the **Dataset** list, choose **Study 1 - Frequency Domain/Parametric Solutions 1 (sol4)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
abs(Ztrans)	1	

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

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

7 From the **Color** list, choose **Blue**.

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

9 Click to expand the **Legends** section. Find the **Prefix and suffix** subsection. In the **Suffix** text field, type `, Frequency Domain (COMSOL)`.

Transfer Impedance vs. Frequency

In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

Global 2

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 1 - Frequency Domain/Parametric Solutions 1 (sol4)**.

3 From the **Parameter selection (ab)** list, choose **Last**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
abs(Zlin)	1	Linear Analytical

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

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

7 From the **Color** list, choose **Red**.

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

Transfer Impedance vs. Frequency

In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

Global 3

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 1 - Frequency Domain/Parametric Solutions 1 (sol4)**.

3 From the **Parameter selection (ab)** list, choose **Last**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
abs(Rnonlin+i*Dnonlin)	1	Nonlinear Analytical

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

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

7 From the **Color** list, choose **Red**.

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

Transfer Impedance vs. Frequency

In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

Global 4

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 2 - Time Domain Linear/Parametric Solutions 2 (sol8)**.

3 From the **Time selection** list, choose **First**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
Ztime	1	Transfer impedance (time domain RMS based)

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

6 From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `f0`.

8 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

9 From the **Color** list, choose **Custom**.

10 Click **Define custom colors**.

11 Set the RGB values to 112, 175, and 26, respectively.

12 Click **Add to custom colors**.

13 Click **Show color palette only** or **OK** on the cross-platform desktop.

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

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

16 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

17 In the table, enter the following settings:

Legends
`Linear Transient (COMSOL)`

Transfer Impedance vs. Frequency

In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

Global 5

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

3 From the **Parameter selection (ab)** list, choose **First**.

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

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`Ztime`	`1`	`Transfer impedance (time domain RMS based)`

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

7 From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `f0`.

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

10 From the **Color** list, choose **Black**.

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

12 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

14 In the table, enter the following settings:

Legends
`ab=0.5, Nonlinear Transient (COMSOL)`

Transfer Impedance vs. Frequency

In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

Global 6

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

3 From the **Parameter selection (ab)** list, choose **From list**.

4 In the **Parameter values (ab)** list, select **0.75**.

5 From the **Time selection** list, choose **First**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
Ztime	1	Transfer impedance (time domain RMS based)

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

8 From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `f0`.

10 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

11 From the **Color** list, choose **Black**.

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

13 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

14 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

15 In the table, enter the following settings:

Legends
ab=0.75, Nonlinear Transient (COMSOL)

Transfer Impedance vs. Frequency

In the **Transfer Impedance vs. Frequency** toolbar, click **Global**.

Global 7

1 In the **Settings** window for **Global**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

3 From the **Parameter selection (ab)** list, choose **Last**.

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

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`Ztime`	`1`	`Transfer impedance (time domain RMS based)`

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

7 From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `f0`.

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

10 From the **Color** list, choose **Black**.

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

12 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

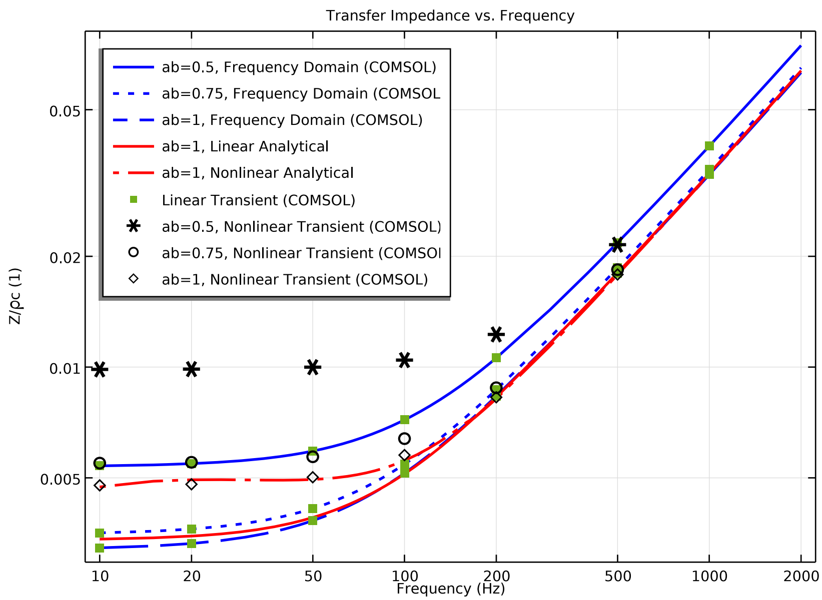
13 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

14 In the table, enter the following settings:

Legends
`ab=1, Nonlinear Transient (COMSOL)`

15 In the **Transfer Impedance vs. Frequency** toolbar, click **Plot**.

The transfer impedance for all the setups, as a function of the frequency, should look like the following figure.



Transfer Impedance vs. Frequency

In the **Model Builder** window, right-click **Transfer Impedance vs. Frequency** and choose **Duplicate**.

Transfer Impedance vs. Strouhal Number

1 In the **Model Builder** window, expand the **Results>Transfer Impedance vs. Frequency 1** node, then click **Transfer Impedance vs. Frequency 1**.

2 In the **Settings** window for **1D Plot Group**, type `Transfer Impedance vs. Strouhal Number` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `Sr (1)`.

Global 1

1 In the **Model Builder** window, expand the **Transfer Impedance vs. Strouhal Number** node, then click **Global 1**.

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

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `Sr`.

Global 2

1 In the **Model Builder** window, click **Global 2**.

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

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `Sr`.

Global 3

1 In the **Model Builder** window, click **Global 3**.

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

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `Sr`.

Global 4

1 In the **Model Builder** window, click **Global 4**.

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

3 In the **Expression** text field, type `Sr2`.

Global 5

1 In the **Model Builder** window, click **Global 5**.

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

3 In the **Expression** text field, type `Sr2`.

Global 6

1 In the **Model Builder** window, click **Global 6**.

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

3 In the **Expression** text field, type `Sr2`.

Global 7

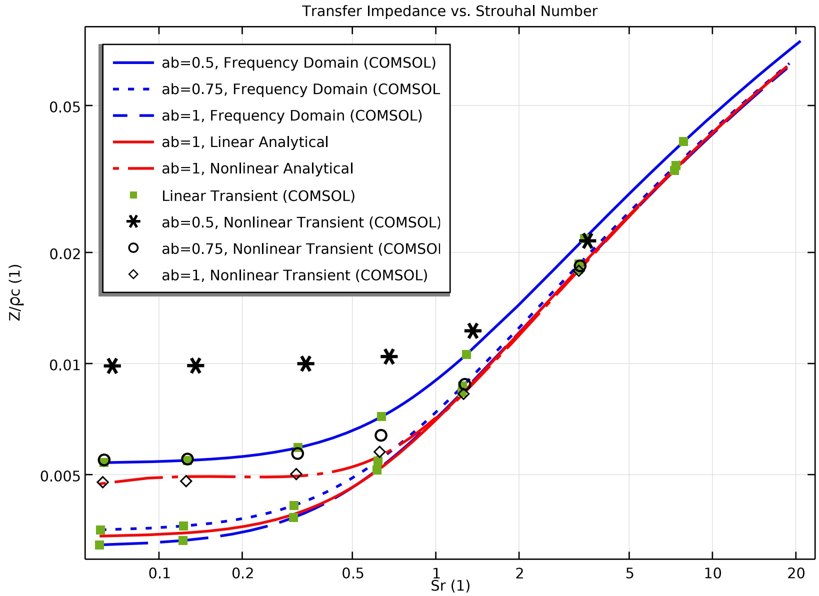
1 In the **Model Builder** window, click **Global 7**.

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

3 In the **Expression** text field, type `Sr2`.

4 In the **Transfer Impedance vs. Strouhal Number** toolbar, click **Plot**.

The transfer impedance for all the setups, as a function of the Strouhal number, should look like the following figure.



Acoustic Pressure (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 Pressure (tatd)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D - Nonlinear**.

4 From the **Parameter value (ab)** list, choose **0.75**.

5 From the **Parameter value (f0 (Hz))** list, choose **50**.

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

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

Surface 1

1 In the **Acoustic Pressure (tatd)** toolbar, click **Surface**.

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

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

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

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

6 Click **OK**.

Acoustic Pressure (tatd)

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

Surface 2

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

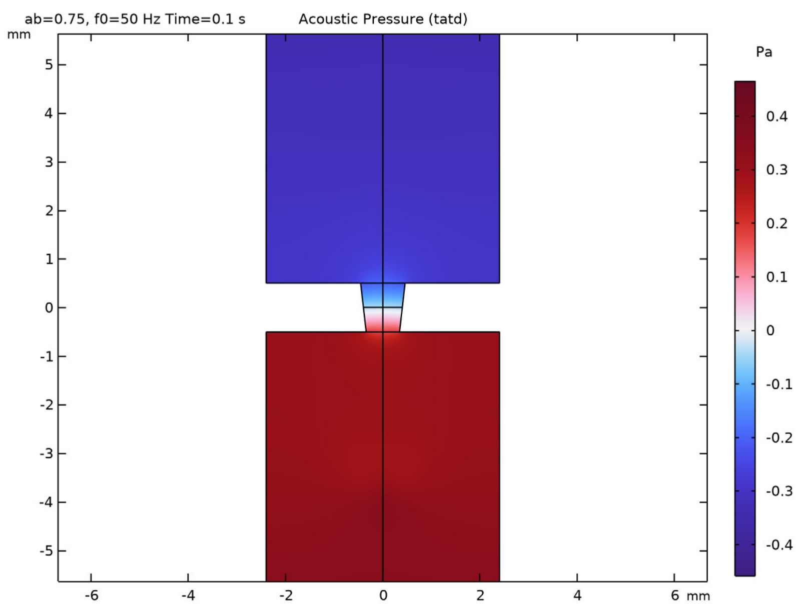
2 In the **Expression** text field, type `actd.p_t`.

3 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Acoustic Pressure (tatd)

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

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



T and V (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 `T and V (tatd)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

4 From the **Parameter value (ab)** list, choose **0.5**.

5 From the **Parameter value (f0 (Hz))** list, choose **100**.

6 From the **Time (s)** list, choose **0.045**.

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

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

Surface 1

1 In the **T and V (tatd)** toolbar, click **Surface**.

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

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

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Expression** section. In the **Expression** text field, type `tatd.T_t`.

6 From the **Unit** list, choose **mK**.

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

8 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.

9 Click **OK**.

T and V (tatd)

In the **T and V (tatd)** toolbar, click **Surface**.

Surface 2

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

2 In the **Expression** text field, type `tatd.v_inst`.

T and V (tatd)

In the **T and V (tatd)** toolbar, click **Arrow Surface**.

Arrow Surface 1

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

2 In the **R-component** text field, type `tatd.u_tr`.

3 In the **Z-component** text field, type `tatd.u_tz`.

4 Locate the **Arrow Positioning** section. Find the **R grid points** subsection. From the **Entry method** list, choose **Coordinates**.

5 Click **Range**.

6 In the **Range** dialog box, type `0` in the **Start** text field.

7 In the **Step** text field, type `0.06`.

8 In the **Stop** text field, type `0.6`.

9 Click **Replace**.

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

11 Find the **Z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

12 Click **Range**.

13 In the **Range** dialog box, type `-1.2` in the **Start** text field.

14 In the **Step** text field, type `3.2/15`.

15 In the **Stop** text field, type `2`.

16 Click **Replace**.

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

18 From the **Color** list, choose **White**.

Selection 1

1 In the **T and V (tatd)** toolbar, click **Selection**.

2 Select Domains 2 and 3 only.

T and V (tatd)

In the **T and V (tatd)** toolbar, click **Line**.

Line 1

1 In the **Settings** window for **Line**, locate the **Expression** section.

2 In the **Expression** text field, type `1`.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Black**.

Selection 1

1 In the **T and V (tatd)** toolbar, click **Selection**.

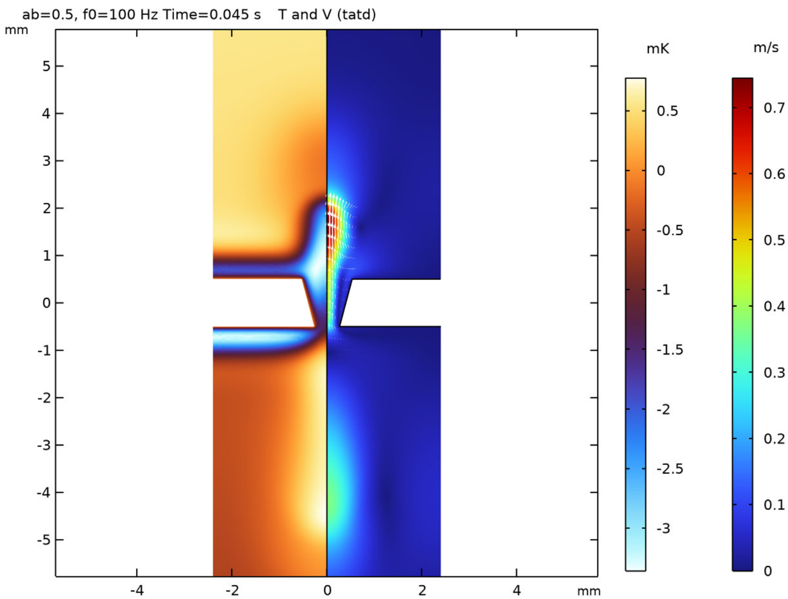
2 Select Boundaries 3, 5, 7, 9, 14–17, 22, and 23 only.

T and V (tatd)

1 In the **Model Builder** window, under **Results** click **T and V (tatd)**.

2 In the **T and V (tatd)** toolbar, click **Plot**.

The instantaneous acoustic temperature fluctuations and the particle velocity of the nonlinear model, at t = 0.045 s and for f = 100 Hz, should look like the following figure.



Acoustic Temperature (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 Temperature (tatd)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D - Nonlinear**.

4 From the **Parameter value (ab)** list, choose **0.75**.

5 From the **Parameter value (f0 (Hz))** list, choose **20**.

6 From the **Time (s)** list, choose **0.212**.

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

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

Surface 1

1 In the **Acoustic Temperature (tatd)** toolbar, click **Surface**.

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

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

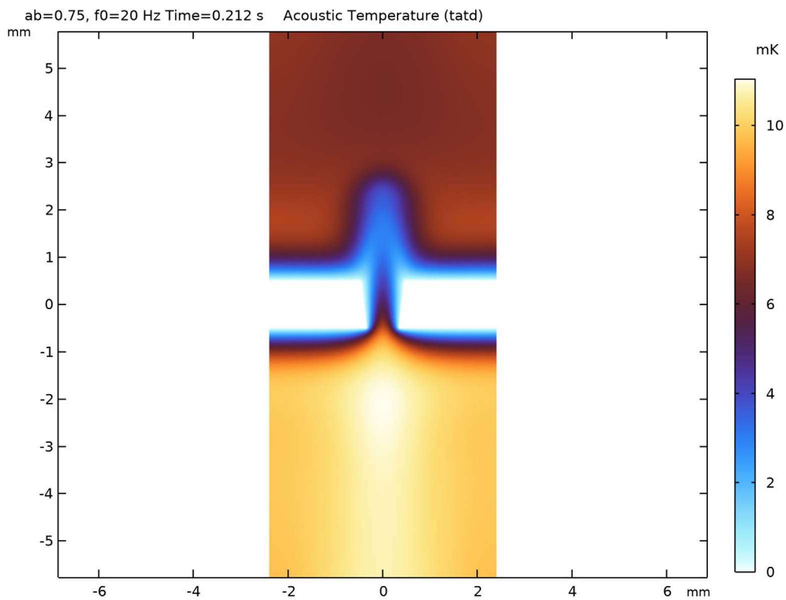
4 From the **Unit** list, choose **mK**.

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

6 In the **Color Table** dialog box, select **Thermal>ThermalWave** in the tree.

7 Click **OK**.

8 In the **Acoustic Temperature (tatd)** toolbar, click **Plot**.



Acoustic Velocity 3D

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Velocity 3D` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (f0 (Hz))** list, choose **100**.

4 From the **Time (s)** list, choose **0.0484**.

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

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

Surface 1

1 In the **Acoustic Velocity 3D** toolbar, click **Surface**.

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

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

Acoustic Velocity 3D

In the **Acoustic Velocity 3D** toolbar, click **Arrow Surface**.

Arrow Surface 1

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

2 In the **R-component** text field, type `tatd.u_tr`.

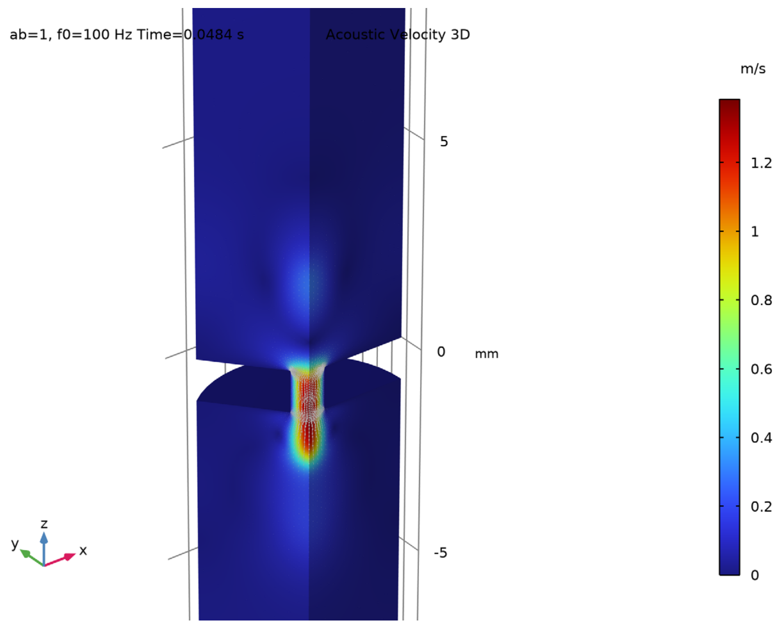
3 In the **PHI-component** text field, type `0`.

4 In the **Z-component** text field, type `tatd.u_tz`.

5 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

7 In the **Acoustic Velocity 3D** toolbar, click **Plot**.



Pressure in Orifice

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

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

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

5 Locate the **Axis** section. Select the **x-axis log scale** check box.

Point Graph 1

1 In the **Pressure in Orifice** toolbar, click **Point Graph**.

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

3 From the **Dataset** list, choose **Study 2 - Time Domain Linear/Parametric Solutions 2 (sol8)**.

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

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

6 In the **Parameter values (f0 (Hz))** list, select **100**.

7 From the **Time selection** list, choose **Manual**.

8 Click **Range**.

9 In the **Integer Range** dialog box, type `5` in the **Start** text field.

10 In the **Stop** text field, type `55`.

11 Click **Add**.

12 In the **Settings** window for **Point Graph**, locate the **Selection** section.

13 Click **Paste Selection**.

14 In the **Paste Selection** dialog box, type `3,4,5` in the **Selection** text field.

15 Click **OK**.

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

17 In the **Expression** text field, type `tatd.p_t`.

18 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Time**.

Pressure in Orifice

In the **Pressure in Orifice** toolbar, click **Point Graph**.

Point Graph 2

1 In the **Settings** window for **Point Graph**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 3 - Time Domain Nonlinear/ Parametric Solutions 3 (sol30)**.

3 From the **Parameter selection (ab)** list, choose **First**.

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

5 In the **Parameter values (f0 (Hz))** list, select **100**.

6 From the **Time selection** list, choose **Manual**.

7 Click **Range**.

8 In the **Integer Range** dialog box, type `5` in the **Start** text field.

9 In the **Stop** text field, type `55`.

10 Click **Replace**.

11 In the **Settings** window for **Point Graph**, locate the **Selection** section.

12 Click **Paste Selection**.

13 In the **Paste Selection** dialog box, type `3,4,5` in the **Selection** text field.

14 Click **OK**.

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

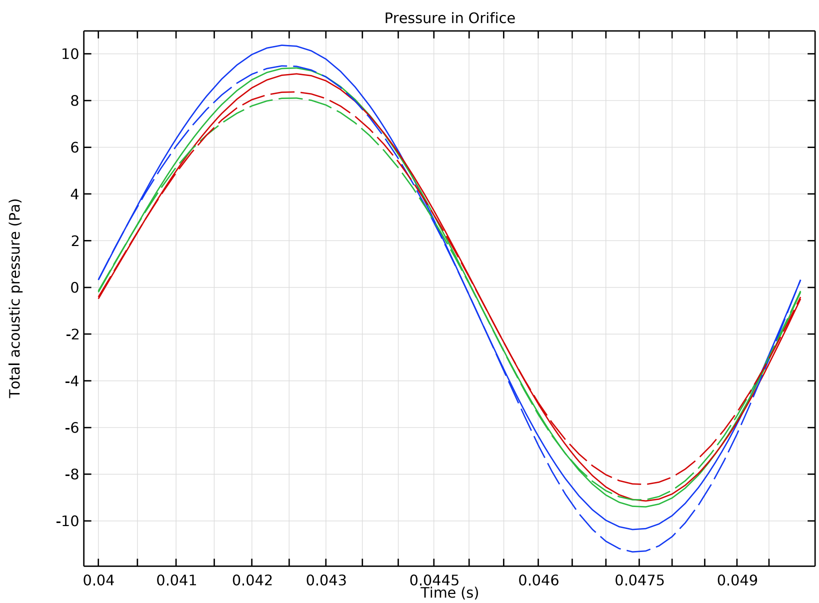
16 In the **Expression** text field, type `tatd.p_t`.

17 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Time**.

18 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

19 From the **Color** list, choose **Cycle (reset)**.

20 In the **Pressure in Orifice** toolbar, click **Plot**.



Sh VS Sr

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

2 In the **Settings** window for **1D Plot Group**, type `Sh VS Sr` in the **Label** text field.

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

4 Locate the **Axis** section. Select the **x-axis log scale** check box.

5 Select the **y-axis log scale** check box.

6 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 In the **Sh VS Sr** toolbar, click **Global**.

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

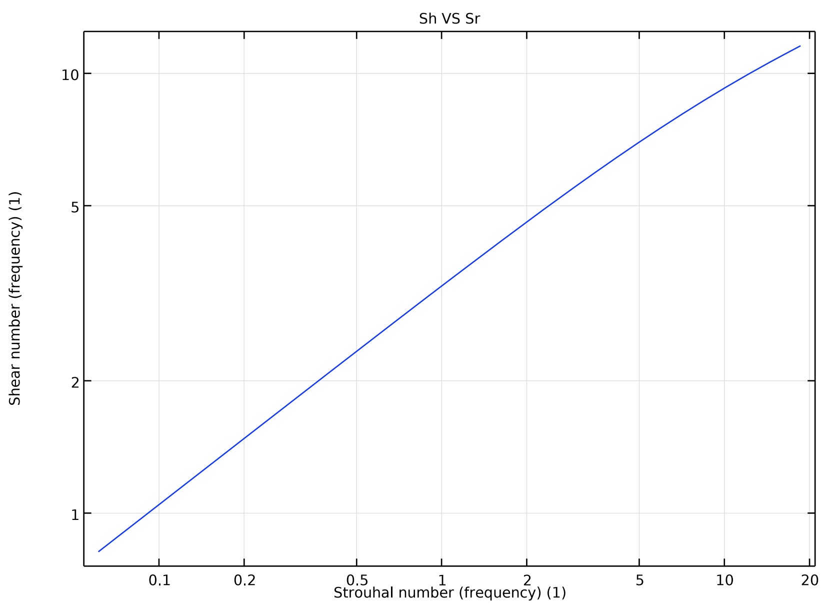
3 In the table, enter the following settings:

Expression	Unit	Description
`Sh`	1	`Shear number (frequency)`

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

5 In the **Expression** text field, type `Sr`.

6 In the **Sh VS Sr** toolbar, click **Plot**.



In the final plot group a nice model thumbnail image is set up. For the setup details see the plot in the model.