Squeeze-Film Damping of Perforated Plates

Introduction

This benchmark model compares the damping coefficients of perforated plates from computation results versus experimental data. The simulation includes 18 different geometric configurations from the experiments reported in Ref. 1. It uses the Bao's perforation model based on Ref. 2, which is built-in in the Thin Film Flow physics interface. Two limiting cases are also simulated: no perforation and zero relative pressure at the perforations.

Each plate is in close proximity to a stationary surface, and the damping results from the squeezing of a thin film of gas in the gap between the two surfaces. The squeezing action forces out the gas from the gap, resulting in a damping force that acts to prevent mechanical contact between the two surfaces. The opposite effect takes place when the surfaces move away from each other as gas is drawn back into the gap.

Each plate has an array of etch holes that allow the gas in the gap to escape. The **Perforations** feature of the Thin Film Flow interface simulates the effect of the etch holes without modeling them explicitly. The feature acts as a sink for gas that is proportional to both the ambient pressure and to the pressure difference with respect to the ambient on the other side of the perforated surface. The constant of proportionality is known as the admittance, and can either be defined directly or determined from the model due to Bao (Ref. 2), as shown in this model.

Model Definition

The Thin-Film Flow, Domain physics interface with the modified Reynolds equation option is used to model the gas film on a flat rectangular plate with an array of etch holes. The geometry is fully parameterized for the study to cycle through each of the 18 different configurations selected from the experimental data reported in Ref. 1.

The model studies three separate cases using two components. The first component covers the limiting case of no etch holes and the case with etch holes using Bao's formula. The second component covers the limiting case of zero relative pressure in the etch holes, with the holes explicitly modeled.

The model is 2D, with the surface normal of the plates pointing out of the paper. The plates are assumed to move only in the surface normal direction, with a prescribed sinusoidal velocity. The **Perforations** feature is used to simulate the effect of the etch holes according to the Bao's model (Ref. 2).

The frequency domain formulation is used, which assumes a small amplitude first order harmonic variation of film pressure, film height and wall velocity at the frequency of interest. The boundary conditions for this model are vanishing relative pressure ($p_f = 0$) at the border of the plate, and, for the third case where the etch holes are explicitly modeled, within every etch holes.

The model compares the computed values of damping coefficients against the experimental data. The results are found to be in good agreement.

Results and Discussion

The values of the relative pressure in the gas film for the frequency domain analysis are plotted in Figure 1, Figure 2, and Figure 3, for the three cases respectively. As expected, the pressure magnitude is the highest for the limiting case of no etch holes, and the lowest for the limiting case of zero relative pressure in the etch holes.
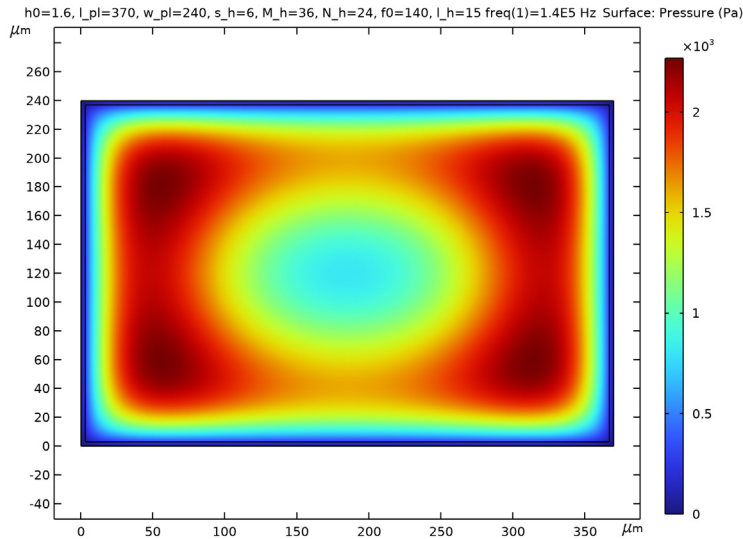

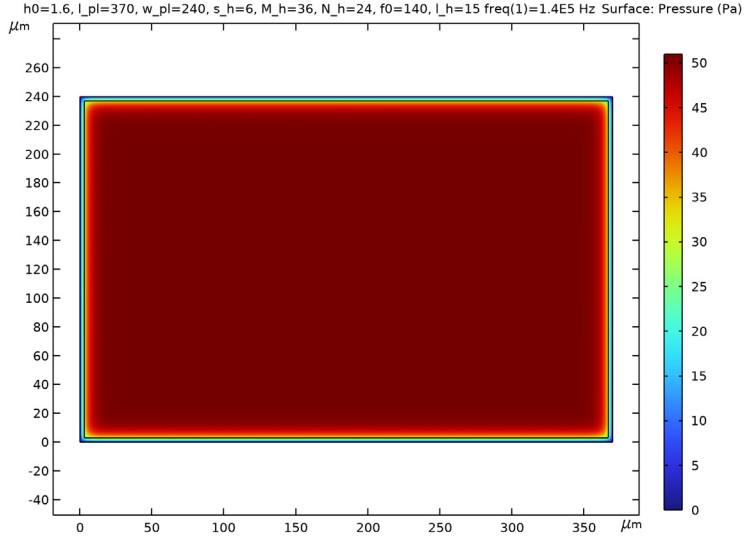Figure 1: Relative gas film pressure for the limiting case of no etch holes.

h0=1.6, l_pl=370, w_pl=240, s_h=6, M_h=36, N_h=24, f0=140, l_h=15 freq(1)=1.4E5 Hz Surface: Pressure (Pa)

Figure 2: Relative gas film pressure for the case of Bao's model.

h0=1.6, l_pl=370, w_pl=240, s_h=6, M_h=36, N_h=24, f0=140, l_h=15 freq(1)=1.4E5 Hz Surface: Pressure (Pa)
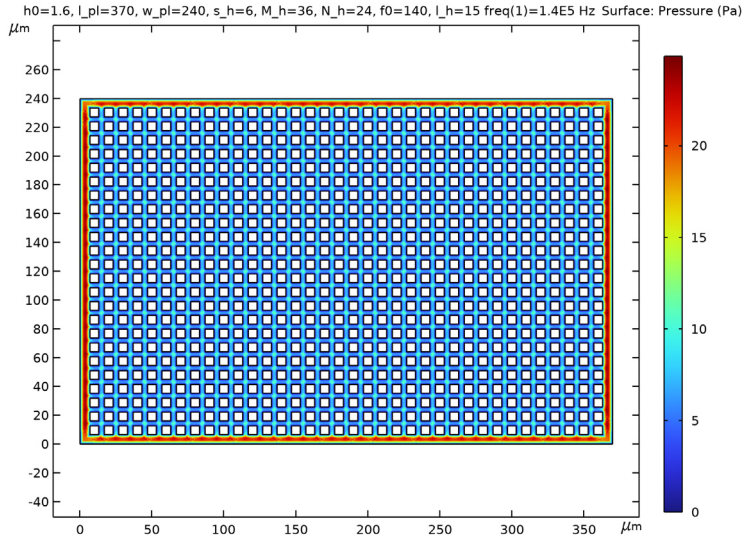

Figure 3: Relative gas film pressure for the limiting case of zero relative pressure in the etch holes.

The computed damping coefficients for the three cases are plotted in Figure 4 with color symbols and lines. The corresponding experimental data are plotted as black dots. The simulation results using Bao's model for the etch holes agree reasonably well with the experimental data.
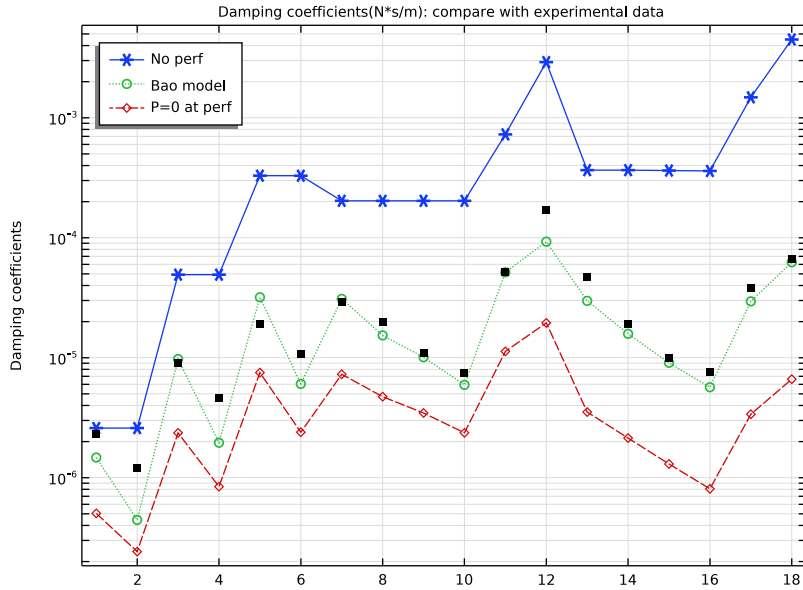

Figure 4: Computed damping coefficients and experimental data (black dots).

References

1. G.D. Pasquale, T. Veijola, and A. Soma, "Modelling and validation of air damping in perforated gold and silicon MEMS plates," *J. Micromech. Microeng*, vol. 20, p. 015010, 2010.

2. M. Bao and H. Yang, "Squeeze film air damping in MEMS," *Sensors and Actuators A: Physical*, vol. 136, no. 1, pp. 3–27, 2007.

Application Library path: MEMS_Module/Actuators/ squeeze_film_perforated_plates

Modeling Instructions

Model three separate cases using two components. The first component covers the limiting case of no perforation and the case with perforation using Bao's formula. The second component covers the limiting case of zero relative pressure in the etch holes, with the holes explicitly modeled. Create the first component using the Model Wizard and the second component later.

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Thin-Film Flow>Thin-Film Flow, Domain (tff)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 Click **Done**.

GEOMETRY 1

The Model Wizard brings up the COMSOL Desktop main user interface and starts at the **Geometry 1** node. Set the length unit to a convenient one.

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 **μm**.

Before building the geometry, first import some global parameters from a file.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

The geometry in **Component 1** is a simple rectangle, since the etch holes will not be modeled explicitly.

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 l_pl.

4 In the **Height** text field, type w_pl.

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 l_per.

4 In the **Height** text field, type w_per.

5 Locate the **Position** section. In the **x** text field, type s2p-s1p.

6 In the **y** text field, type s_2-s_1.

Create a global material to be shared by the two components.

G L O B A L   D E F I N I T I O N S

Material 1 (mat1)

In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

From the first component, create a material link to the global material.

M A T E R I A L S

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

Fill in material data in the global material. Only the viscosity will be needed for gases.

G L O B A L   D E F I N I T I O N S

Material 1 (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	mu0	Pa·s	Basic

First set up the physics for the limiting case of no perforation.

THIN-FILM FLOW, DOMAIN - NO PERFORATION

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thin-Film Flow, Domain (tff)**.

2 In the **Settings** window for **Thin-Film Flow, Domain**, type `Thin-Film Flow, Domain - no perforation` in the **Label** text field.

The modified Reynolds equation is suitable for gases.

3 Locate the **Physical Model** section. From the **Equation type** list, choose **Modified Reynolds equation**.

4 Locate the **Reference Pressure** section. In the p_{ref} text field, type `pRef`.

Fluid-Film Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thin-Film Flow, Domain - no perforation (tff)** click **Fluid-Film Properties 1**.

2 In the **Settings** window for **Fluid-Film Properties**, locate the **Wall Properties** section.

3 In the h_w text field, type `h0`.

4 From the $\mathbf{v}_w$ list, choose **User defined**. Specify the vector as

vf	z

Next set up the physics for the case with perforation, using Bao's model.

ADD PHYSICS

1 In the **Home** toolbar, click ⚛ **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Fluid Flow>Thin-Film Flow>Thin-Film Flow, Domain (tff)**.

4 Click **Add to Component 1** in the window toolbar.

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

THIN-FILM FLOW, DOMAIN - BAO MODEL

1 In the **Settings** window for **Thin-Film Flow, Domain**, type `Thin-Film Flow, Domain - Bao model` in the **Label** text field.

2 Locate the **Physical Model** section. From the **Equation type** list, choose **Modified Reynolds equation**.

3 Locate the **Reference Pressure** section. In the p_{ref} text field, type pRef.

Fluid-Film Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thin-Film Flow, Domain - Bao model (tff2)** click **Fluid-Film Properties 1**.

2 In the **Settings** window for **Fluid-Film Properties**, locate the **Wall Properties** section.

3 In the h_w text field, type h0.

4 From the $\mathbf{v}_w$ list, choose **User defined**. Specify the vector as

vf	z

Set up the perforation model.

Perforations 1

1 In the **Physics** toolbar, click ⬤ **Domains** and choose **Perforations**.

2 Select Domain 2 only.

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

4 From the **Perforation admittance** list, choose **Bao model**.

5 In the s_h text field, type s_h.

6 In the l_h text field, type l_h.

7 In the n_h text field, type 1/(l_per*w_per/M_h/N_h).

DEFINITIONS

Define some useful variables.

Integration 1 (intop1)

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

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

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

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
f_tot_no_perf	intop1(tff.fwallz)	N	Total force without perforation
f_tot_Bao	intop1(tff2.fwallz)	N	Total force with perforation; Bao's model

Enter experimental data into an interpolation function for plotting against the simulation result.

Interpolation of experimental data

1 In the **Definitions** toolbar, click ◠. **Interpolation**.

2 In the **Settings** window for **Interpolation**, type Interpolation of experimental data in the **Label** text field.

3 Locate the **Definition** section. From the **Data source** list, choose **File**.

4 Click 📂 **Browse**.

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

6 Click 📥 **Import**.

7 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Nearest neighbor**.

8 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	1

9 In the **Function** table, enter the following settings:

Function	Unit
int1	uN*s/m

Add a 2D component for the limiting case of zero relative pressure at the perforations.

ADD COMPONENT

Right-click **Interpolation of experimental data** and choose **Add Component>2D**.

GEOMETRY 2

Build the geometry with explicit perforations.

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

2 From the **Length unit** list, choose **µm**.

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 l_pl.

4 In the **Height** text field, type w_pl.

Square 1 (sq1)

1 In the **Geometry** toolbar, click ▢ **Square**.

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

3 In the **Side length** text field, type s_h.

4 Locate the **Position** section. In the **x** text field, type s2p.

5 In the **y** text field, type s_2.

Array 1 (arr1)

1 In the **Geometry** toolbar, click ⟳ **Transforms** and choose **Array**.

2 Select the object **sq1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type M_h.

5 In the **y size** text field, type N_h.

6 Locate the **Displacement** section. In the **x** text field, type s_h+s1p.

7 In the **y** text field, type s_h+s_1.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 From the **Objects to subtract** list, choose **Array 1**.

5 Click ▦ **Build All Objects**.

Add a material link to the global material.

MATERIALS

Material Link 2 (matlnk2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

Set up the physics.

ADD PHYSICS

1 In the **Home** toolbar, click ⚛ **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Fluid Flow>Thin-Film Flow>Thin-Film Flow, Domain (tff)**.

4 Click **Add to Component 2** in the window toolbar.

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

THIN-FILM FLOW, DOMAIN - P = 0 AT PERF

1 In the **Settings** window for **Thin-Film Flow, Domain**, type `Thin-Film Flow, Domain - P = 0 at perf` in the **Label** text field.

2 Locate the **Physical Model** section. From the **Equation type** list, choose **Modified Reynolds equation**.

3 Locate the **Reference Pressure** section. In the p_{ref} text field, type `pRef`.

Fluid-Film Properties 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Thin-Film Flow, Domain - P = 0 at perf (tff3)** click **Fluid-Film Properties 1**.

2 In the **Settings** window for **Fluid-Film Properties**, locate the **Wall Properties** section.

3 In the h_w text field, type `h0`.

4 From the $\mathbf{v}_w$ list, choose **User defined**. Specify the vector as

vf	z

DEFINITIONS (COMP2)

Define some useful variables.

Integration 2 (intop2)

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

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

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

Variables 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
f_tot_p0	intop2(tff3.fwallz)	N	Total force with zero relative pressure at perforations

Add a **Parametric Sweep** node to cycle through the geometric dimensions used in the experiments.

STUDY 1

Parametric Sweep

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

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

3 Click 📂 **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file squeeze_film_perforated_plates_study_data.txt.
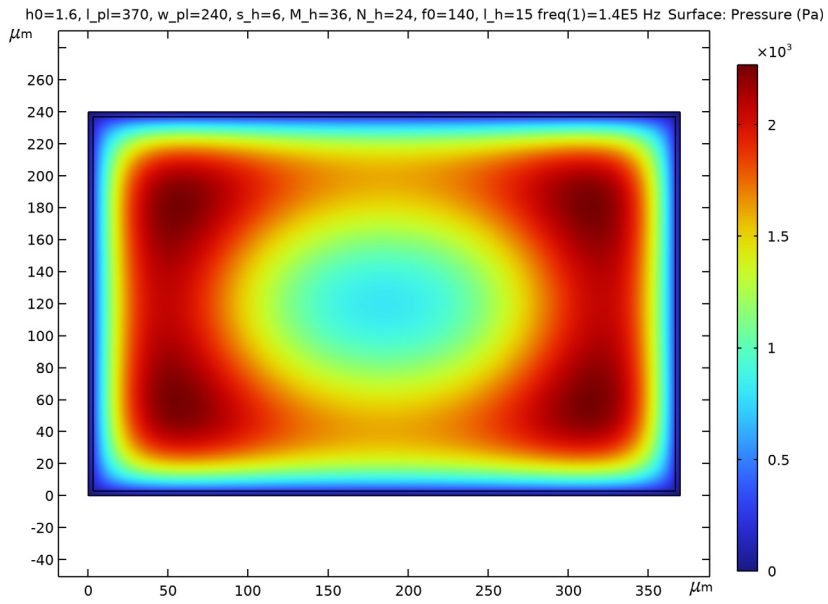
Step 1: Frequency Domain

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 In the **Frequencies** text field, type f0.

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

RESULTS

2D Plot Group 1 - no perf
Examine the results.

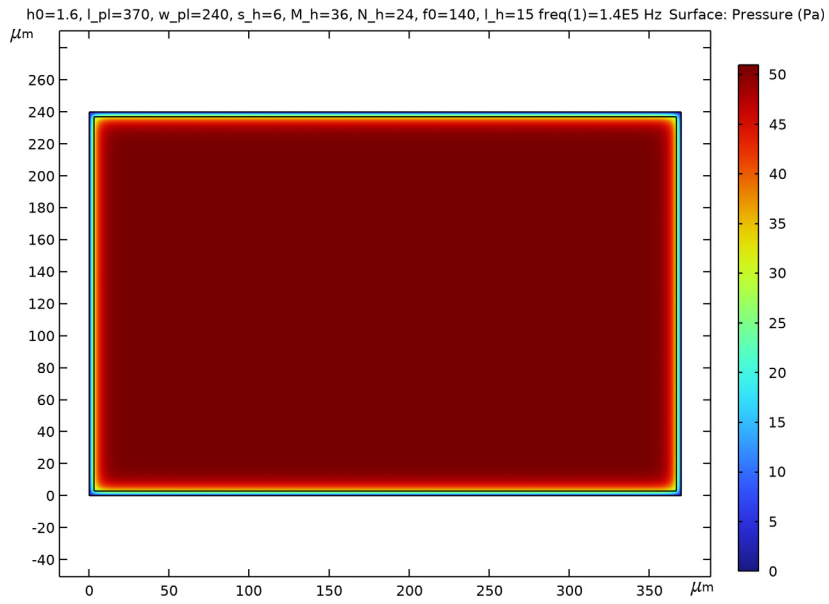**1** In the **Settings** window for **2D Plot Group**, type 2D Plot Group 1 - no perf in the **Label** text field.

h0=1.6, l_pl=370, w_pl=240, s_h=6, M_h=36, N_h=24, f0=140, l_h=15 freq(1)=1.4E5 Hz Surface: Pressure (Pa)

2D Plot Group 2 - Bao model

1 In the **Model Builder** window, under **Results** click **Fluid Pressure (tff2)**.

2 In the **Settings** window for **2D Plot Group**, type 2D Plot Group 2 - Bao model in the **Label** text field.

h0=1.6, l_pl=370, w_pl=240, s_h=6, M_h=36, N_h=24, f0=140, l_h=15 freq(1)=1.4E5 Hz Surface: Pressure (Pa)

2D Plot Group 3 - P=0 at perf

1 In the **Model Builder** window, under **Results** click **Fluid Pressure (tff3)**.

2 In the **Settings** window for **2D Plot Group**, type 2D Plot Group 3 - P=0 at perf in the **Label** text field.

h0=1.6, l_pl=370, w_pl=240, s_h=6, M_h=36, N_h=24, f0=140, l_h=15 freq(1)=1.4E5 Hz Surface: Pressure (Pa)
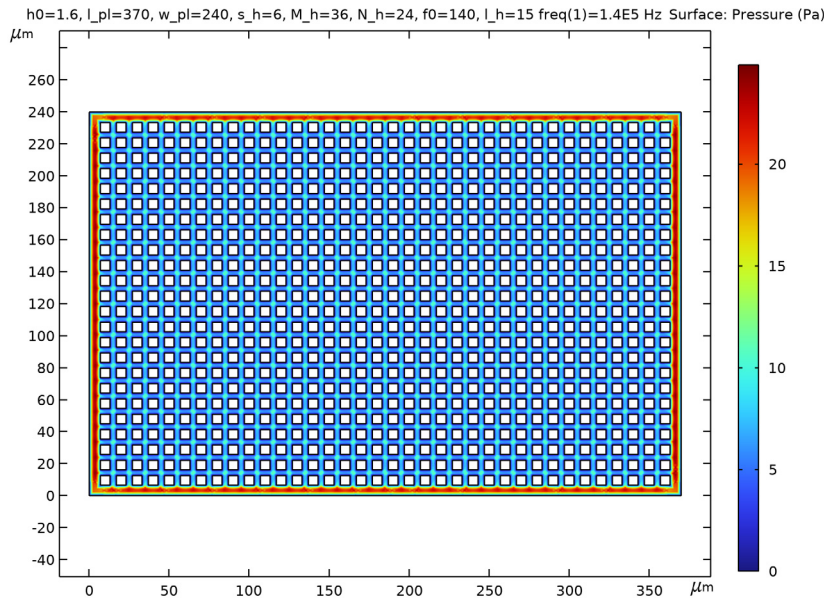
Plot the damping coefficients and compare with the experimental data.

Damping coefficients

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

2 In the **Settings** window for **1D Plot Group**, type Damping coefficients in the **Label** text field.

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

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

5 In the **Title** text area, type Damping coefficients(N*s/m): compare with experimental data.

6 Locate the **Plot Settings** section. Select the **y-axis label** check box.

7 In the associated text field, type Damping coefficients.

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

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

Global 1

1 Right-click **Damping coefficients** 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
abs(f_tot_no_perf)/vf	N*s/m	No perf
abs(f_tot_Bao)/vf	N*s/m	Bao model

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

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

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

7 From the **Positioning** list, choose **In data points**.

The result from the second component is stored in a different dataset, which can be plotted in a separate global plot.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (4) (sol2)**.

4 Locate the **y-Axis Data** section. Click 🧹 **Clear Table**.

5 In the table, enter the following settings:

Expression	Unit	Description
abs(f_tot_p0)/vf	N*s/m	P=0 at perf

Plot the experimental data.

DEFINITIONS (COMP1)

Interpolation of experimental data (int1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Interpolation of experimental data (int1)**.

2 In the **Settings** window for **Interpolation**, click 📊 **Create Plot**.

1D Plot Group 5

Copy and paste the plot of the experimental data to compare with the simulation results.

Function 1

1 In the **Model Builder** window, expand the **1D Plot Group 5** node, then click **Function 1**.

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

3 From the **Dataset** list, choose **Grid 1D 1**.

4 Right-click **Function 1** and choose **Copy**.

Function 1

1 In the **Model Builder** window, right-click **Damping coefficients** and choose **Paste Function**.

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

3 In the **Unit** field, type N*s/m.

4 Locate the **Output** section. From the **Display** list, choose **Points**.

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type 6.

6 In the **Damping coefficients** toolbar, click ➡️ **Plot**.

Damping coefficients(N*s/m): compare with experimental data