

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.



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



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

- I 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 I

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

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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.
- 6 | SQUEEZE-FILM DAMPING OF PERFORATED PLATES

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

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 1_p1.
- 4 In the **Height** text field, type w_pl.

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 1_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.

GLOBAL DEFINITIONS

Material I (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.

MATERIALS

Material Link I (matlnk I)

In the Model Builder window, under Component I (compl) 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.

GLOBAL DEFINITIONS

Material I (mat1)

I 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

- I In the Model Builder window, under Component I (compl) 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 **Fluid type** list, choose **Gas (modified Reynolds equation)**.
- 4 Locate the Reference Pressure section. In the p_{ref} text field, type pRef.

Fluid-Film Properties 1

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

- I 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 I in the window toolbar.
- 5 In the Home toolbar, click 🖄 Add Physics to close the Add Physics window.

THIN-FILM FLOW, DOMAIN - BAO MODEL

I 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 Fluid type list, choose Gas (modified Reynolds equation).
- 3 Locate the **Reference Pressure** section. In the p_{ref} text field, type pRef.

Fluid-Film Properties 1

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

- I 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 1_h.
- 7 In the n_h text field, type 1/(1_per*w_per/M_h/N_h).

DEFINITIONS

Define some useful variables.

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 Selection list, choose All domains.

Variables I

- I 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	<pre>intop1(tff.fwallz)</pre>	N	Total force without perforation
f_tot_Bao	<pre>intop1(tff2.fwallz)</pre>	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

- I 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
intl	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.

- I In the Settings window for Geometry, locate the Units section.
- 2 From the Length unit list, choose µm.

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 1_p1.
- 4 In the **Height** text field, type w_pl.

Square 1 (sq1)

- I 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 I (arr1)

- I In the Geometry toolbar, click 💭 Transforms and choose Array.
- 2 Select the object sql 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 I (dif1)

- I In the Geometry toolbar, click P Booleans and Partitions and choose Difference.
- 2 Select the object rI only.
- 3 In the Settings window for Difference, locate the Difference section.
- 4 From the Objects to subtract list, choose Array I.
- 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

- I 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

- I 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 Fluid type list, choose Gas (modified Reynolds equation).
- 3 Locate the Reference Pressure section. In the p_{ref} text field, type pRef.

Fluid-Film Properties 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)

- 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 Selection list, choose All domains.

Variables 2

- I 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	<pre>intop2(tff3.fwallz)</pre>	Ν	Total force with zero relative pressure at perforations

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

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 **b** 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

- I In the Model Builder window, click Step I: 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. I In the **Settings** window for **2D Plot Group**, type **2D Plot Group 1** - no perf in the **Label** text field.



h0=1.6 um, l_pl=370 um, w_pl=240 um, s_h=6 um, M_h=36, N_h=24, f0=140 kHz, l_h=15 um freq(1)=1.4E5 H;

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

²D Plot Group 2 - Bao model

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



h0=1.6 um, l_pl=370 um, w_pl=240 um, s_h=6 um, M_h=36, N_h=24, f0=140 kHz, l_h=15 um freq(1)=1.4E5 H;

2D Plot Group 3 - P=0 at perf

I 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=O at perf in the Label text field.



h0=1.6 um, l_pl=370 um, w_pl=240 um, s_h=6 um, M_h=36, N_h=24, f0=140 kHz, l_h=15 um freq(1)=1.4E5 H:

Plot the damping coefficients and compare with the experimental data.

Damping coefficients

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Damping coefficients in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study I/ Parametric Solutions I (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.
- 7 Select the y-axis label check box. 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 I

- I 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.

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

Global 2

- I Right-click Global I and choose Duplicate.
- 2 In the Settings window for Global, locate the Data section.
- 3 From the Dataset list, choose Study I/Parametric Solutions I (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 (COMPI)

Interpolation of experimental data (int I)

- I In the Model Builder window, under Component I (compl)>Definitions click Interpolation of experimental data (intl).
- 2 In the Settings window for Interpolation, click 🐻 Create Plot.

RESULTS

ID Plot Group 5

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

Function I

- I In the Model Builder window, expand the ID Plot Group 5 node, then click Function I.
- 2 In the Settings window for Function, locate the Data section.
- 3 From the Dataset list, choose Grid ID I.
- 4 Right-click Function I and choose Copy.

Function I

- I 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. From the Width list, choose 6.
- 6 In the **Damping coefficients** toolbar, click **I** Plot.

