Vibration of a Squeezed Plate

Introduction

This model illustrates how to perform eigenfrequency and frequency response analyses for a contact problem.

A circular plate is squeezed between two thicker plates. The peak of a frequency response analysis is compared with the eigenfrequency and also with the natural frequency of an ideal case, that is, an annular plate fixed at the inner ring boundary.

The modeling sequence involving all necessary study steps and model parameter updates is automated using a model method.

Model Definition

The analysis is done using an assumption of 2D axisymmetry. The geometry is shown in Figure 1. A circular plate made of steel is constrained by means of mechanical contact with a holder modeled as two clamping plates.

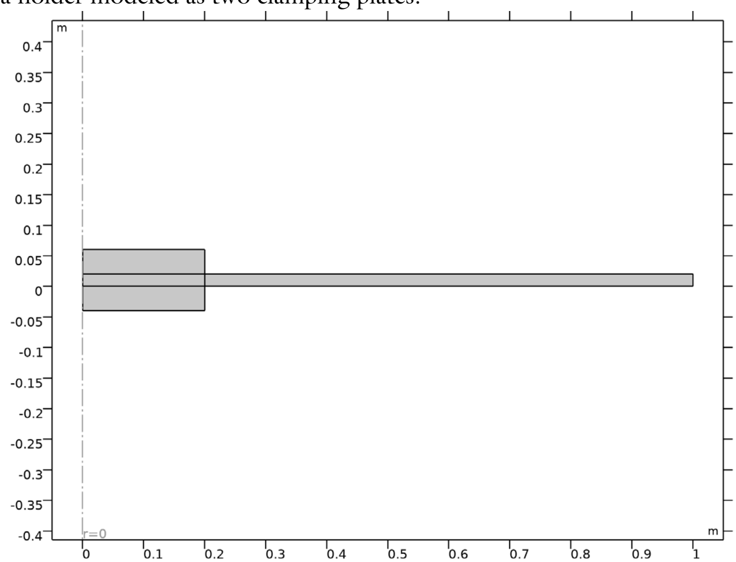

Figure 1: Geometry consisting of a large thin plate clamped between two thicker plates.

Results and Discussion

For the plate holder radius `ri` = 0.2 m, the first natural frequency and the corresponding eigenmode are shown in Figure 2. The eigenfrequency is complex valued because of the damping properties added to model. The real part of the frequency is approximately 24.3 Hz. For an equivalent annular plate fixed at the inner ring boundary, the first natural frequency is around 25.2 Hz.

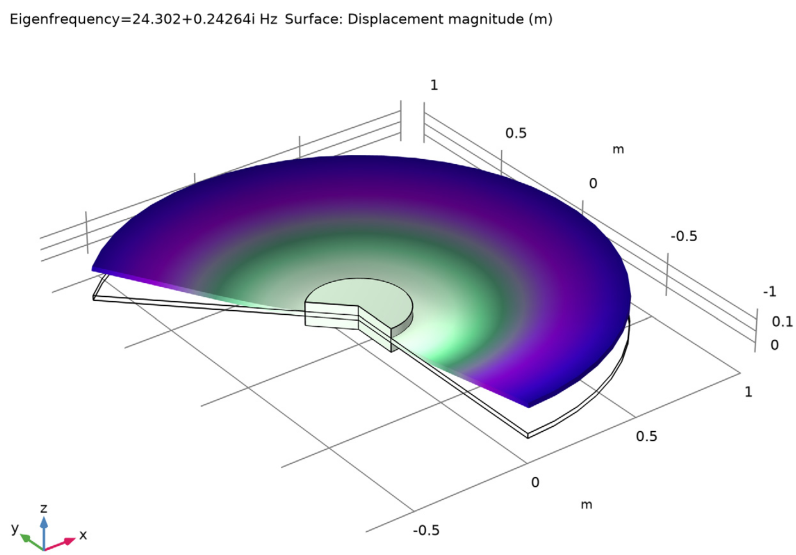

Figure 2: The first eigenmode for a plate clamped using contact. The plate holder radius is 0.2 m.

The frequency response for a frequency range containing the first eigenmode is shown in Figure 3.

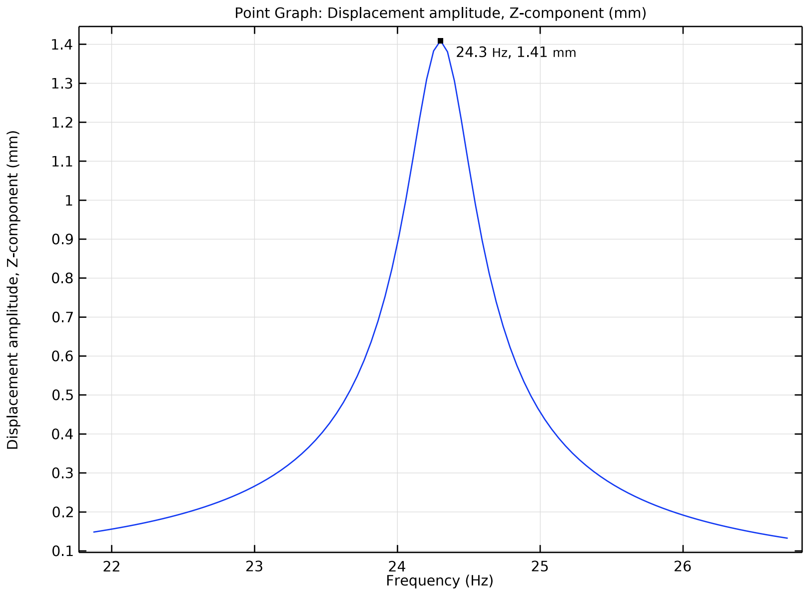

Figure 3: Vertical displacement of the plate outer edge.

The maximum value occurs at the frequency value close to the eigenfrequency. The corresponding deformation of the plate is shown in Figure 4.

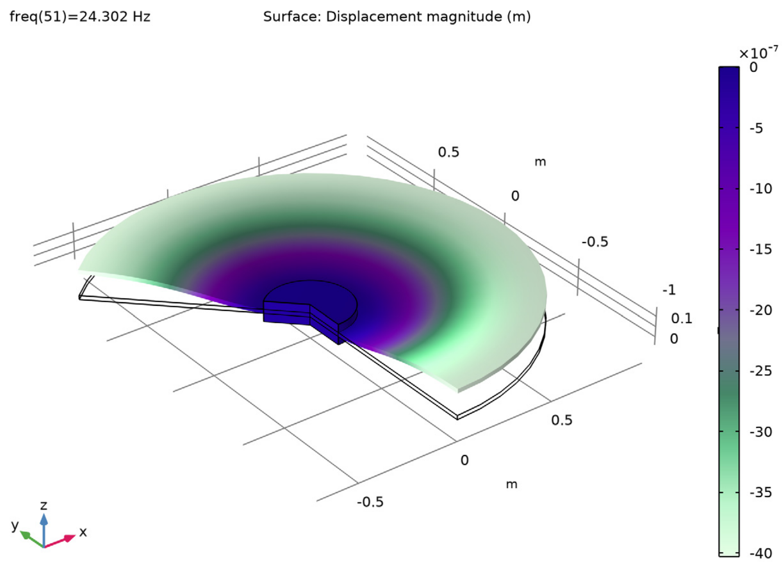

Figure 4: Response of the plate to a boundary load excitation at a frequency close to the first eigenfrequency.

The variation in the first natural frequency with respect to a change in the plate holder radius is shown in Figure 4. The results computed by means of the frequency domain analysis present a very good estimate of the eigenfrequency.

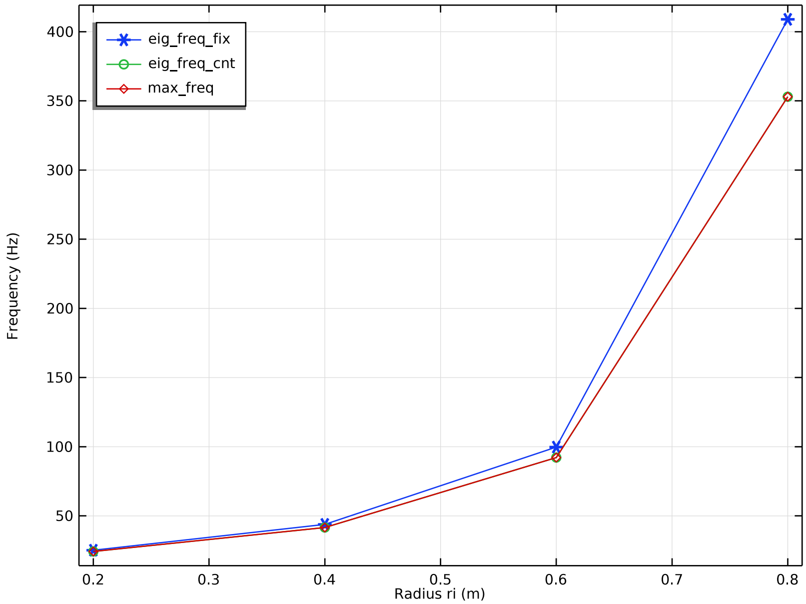

Figure 5: The first natural frequency as a function of the plate holder radius. The curves represent the results from the eigenfrequency analysis (annular constrained and squeezed plate) and from the maximum frequency response, respectively.

Notes About the COMSOL Implementation

The contact state is modeled via a stationary study step using the augmented Lagrangian method. The results are then used as a linearization point for both the eigenfrequency and the frequency domain study steps. A parametric sweep including all study steps is automated using a model method. The method also performs all necessary evaluations and model parameter updates between the study steps. You enter the Java® code for the method using the method editor, which is only available in the Windows® version of the COMSOL Desktop®.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/squeezed_plate_response`

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 **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
F	1[kN/m^2]	1000 N/m²	Applied harmonic load
ro	1[m]	1 m	Plate radius
ri	0.2[m]	0.2 m	Plate holder radius
th	0.02[m]	0.02 m	Plate thickness
freq_ref	24.302	24.302	Reference frequency
disp_max	0.0014454	0.0014454	Peak displacement

GEOMETRY 1

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r0`.

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

5 In the **Width** text field, type `ro`.

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

Layer name	Thickness (m)
Layer 1	ri

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

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

9 Click **Build Selected**.

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 `ri`.

4 In the **Height** text field, type `2*th`.

5 Locate the **Position** section. In the **z** text field, type `th`.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **z** text field, type `-2*th`.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 In the **Geometry** toolbar, click **Build All**.

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundaries 3 and 13 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 6 and 7 only.

6 Locate the **Advanced** section. From the **Mapping method** list, choose **Initial configuration**.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Structural steel**.

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

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

SOLID MECHANICS (SOLID)

Contact 1

Use the augmented Lagrangian method to increase the accuracy of the contact pressure.

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Contact 1**.

2 In the **Settings** window for **Contact**, locate the **Contact Method** section.

3 From the list, choose **Augmented Lagrangian**.

4 Locate the **Contact Pressure Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

Roller 1

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

2 Select Boundary 2 only.

Prescribed Displacement 1

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

2 Select Boundary 14 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

5 In the u_{0z} text field, type `-1[um]`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 From the **Damping type** list, choose **Isotropic loss factor**.

MATERIALS

Structural steel (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Structural steel (mat1)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Isotropic structural loss factor	eta_s	0.02	1	Basic

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Physics** toolbar, click **Domains** and choose **Fixed Constraint**.

2 Select Domains 1, 2, and 4 only.

This constraint makes the geometry equivalent to an annular plate fixed at the inner ring boundary.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 5 only.

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

4 In the **Number of elements** text field, type 4.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 2, 6, 13, and 14 only.

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

4 In the **Number of elements** text field, type 24.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Extremely fine**.

4 Click **Build All**.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type 1.

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

RESULTS

Mode Shape (solid)

Add a numerical evaluation to extract the natural frequency.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
freq	1/s	Frequency

4 Click **Evaluate**.

The value of the real part should be close to 25.2 Hz.

Next, compute the first natural frequency for a plate clamped by using contact.

ADD STUDY

1 In the **Home** 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 Selected Physics Interfaces>Eigenfrequency, Prestressed**.

4 Right-click and choose **Add Study**.

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

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

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Fixed Constraint 1**.

4 Click **Disable**.

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `1`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Fixed Constraint 1**.

6 Click **Disable**.

7 In the **Home** toolbar, click **Compute**.

RESULTS

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Click next to **Evaluate**, then choose **New Table**.

TABLE

1 Go to the **Table** window.

The value of the real part should be close to 24.3 Hz. Compare the result with the first natural frequency of the equivalent annular plate.

Next, prepare to perform a frequency response analysis.

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 Select Boundary 11 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	r
-F	z

5 Right-click **Boundary Load 1** and choose **Harmonic Perturbation**.

ADD STUDY

1 In the **Home** 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 **Empty Study**.

4 Right-click and choose **Add Study**.

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

STUDY 3

Frequency Domain Perturbation

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain Perturbation**.

Compute the solution for a frequency range that contains the reference frequency.

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

3 In the **Frequencies** text field, type `freq_ref*range(0.9,2e-3,1.1)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Fixed Constraint 1**.

6 Click **Disable**.

Use the same prestress static solution as for the Eigenfrequency study step.

7 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

8 From the **Method** list, choose **Solution**.

9 From the **Study** list, choose **Study 2, Eigenfrequency**.

10 From the **Solution** list, choose **Solution 2 (sol2)**.

Solution 4 (sol4)

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

Step 1: Frequency Domain Perturbation

1 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Study 3>Step 1: Frequency Domain Perturbation**.

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

3 Find the **Values of linearization point** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 2, Eigenfrequency**.

6 From the **Solution** list, choose **Solution 2 (sol2)**.

7 From the **Use** list, choose **Solution Store 1 (sol3)**.

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `solid.mises`.

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

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

6 Click **OK**.

Displacement, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **24.302**.

4 In the **Label** text field, type `Displacement, 3D (solid)`.

Surface 1

1 In the **Model Builder** window, expand the **Displacement, 3D (solid)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `solid.disp`.

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

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

6 Click **OK**.

7 In the **Displacement, 3D (solid)** toolbar, click **Plot**.

8 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

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

2 In the tree, select **Study 2/Solution Store 1 (sol3)>Solid Mechanics>Contact Forces (solid)**.

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

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Contact Forces (solid)

1 In the **Model Builder** window, under **Results** click **Contact Forces (solid)**.

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

3 From the **Dataset** list, choose **Study 2/Solution Store 1 (sol3)**.

4 In the **Contact Forces (solid)** toolbar, click **Plot**.

1D Plot Group 8

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

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

3 From the **Dataset** list, choose **Study 3/Solution 4 (sol4)**.

4 Locate the **Plot Settings** section.

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

Point Graph 1

1 Right-click **1D Plot Group 8** and choose **Point Graph**.

2 Select Point 10 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Displacement>Displacement amplitude (material and geometry frames) - m>solid.uAmpZ - Displacement amplitude, Z-component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.

Graph Marker 1

1 Right-click **Point Graph 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Display** list, choose **Max**.

4 Locate the **Text Format** section. Select the **Show x-coordinate** check box.

5 Select the **Include unit** check box.

6 In the **Display precision** text field, type `3`.

7 In the **1D Plot Group 8** toolbar, click **Plot**.

In the following steps, you will evaluate the peak frequency which you can see in the newly generated plot.

Point Evaluation 1

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

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

3 From the **Dataset** list, choose **Study 3/Solution 4 (sol4)**.

4 Select Point 10 only.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`solid.uAmpZ`	`m`	`Displacement amplitude, Z component`

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

7 Click **Evaluate**.

Point Evaluation 2

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

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

3 From the **Dataset** list, choose **Study 3/Solution 4 (sol4)**.

4 Select Point 10 only.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq*(solid.uAmpZ>=disp_max-eps)`	Hz	Computed peak frequency

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

7 Click **Evaluate**.

TABLE

Go to the **Table** window, compare the computed value with the real part of the eigenfrequency.

In the remaining part of the modeling, you will create a **Model Method**. For each value of the plate holder radius, `ri`, the method computes the first eigenfrequency and then estimates the peak frequency using the frequency sweep around the eigenfrequency.

The method editor is only available in the Windows® version of the COMSOL Desktop.

NEW METHOD

1 In the **Developer** toolbar, click **New Method**.

2 In the **New Method** dialog box, type `runModel` in the **Name** text field.

3 Click **OK**.

APPLICATION BUILDER

runModel

1 In the **Application Builder** window, under **Methods** click **runModel**.

2 Copy the following code into the **runModel** window:

```
// Clean up and recreate the tables in the model
model.result().table("tbl1").clearTableData();
model.result().table("tbl2").clearTableData();
model.result().table("tbl3").clearTableData();
model.result().table("tbl4").clearTableData();
model.result().numerical("gev1").set("table", "tbl1");
model.result().numerical("gev2").set("table", "tbl2");
model.result().numerical("pev1").set("table", "tbl3");
model.result().numerical("pev2").set("table", "tbl4");
// You will decrease the plate holder radius, ri, in several steps
```

```
double rimax = 0.8;
double rimin = 0.2;
int n_steps = 4;
double dri = (rimax-rimin)/(n_steps-1);
double ri = rimax;
// Data structure for the tables
double[][] data_eig_freq_fix = new double[n_steps][2];
double[][] data_eig_freq_cnt = new double[n_steps][2];
double[][] data_max_freq = new double[n_steps][2];
for (int n = 0; n < n_steps; n++) {
 data_eig_freq_fix[n][0] = ri;
 data_eig_freq_cnt[n][0] = ri;
 data_max_freq[n][0] = ri;
 model.param().set("ri", ri);
 // Compute the eigenfrequency
 model.study("std1").run();
 double eig_freq_fix = model.result().numerical("gev1").getReal()[0][0];
 data_eig_freq_fix[n][1] = eig_freq_fix;
 model.study("std2").run();
 double eig_freq_cnt = model.result().numerical("gev2").getReal()[0][0];
 data_eig_freq_cnt[n][1] = eig_freq_cnt;
 model.param().set("freq_ref", eig_freq_cnt);
 // Compute the frequency response
 model.study("std3").run();
 double disp_max = model.result().numerical("pev1").getReal()[0][0];
 model.result().numerical("pev1").setResult();
 model.param().set("disp_max", disp_max);
 model.sol("sol4").updateSolution();
 data_max_freq[n][1] = model.result().numerical("pev2").getReal()[0][0];
 model.result("pg5").run();
 ri = ri-dri;
}
// Populate the tables

model.result().table("tbl1").setColumnHeaders(new String[]{"ri",
"eig_freq_fix"});
model.result().table("tbl1").setTableData(data_eig_freq_fix);

model.result().table("tbl2").setColumnHeaders(new String[]{"ri",
"eig_freq_cnt"});
model.result().table("tbl2").setTableData(data_eig_freq_cnt);

model.result().table("tbl4").setColumnHeaders(new String[]{"ri", "max_freq"});
model.result().table("tbl4").setTableData(data_max_freq);
```

METHODS

In the **Home** toolbar, click **Model Builder** to switch to the main desktop.

GLOBAL DEFINITIONS

Click **Method Call** and choose **runModel**.

RunModel 1

Click **Run Method** and choose **runModel**.

Finally, plot the computed frequencies.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

1D Plot Group 9

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Results>1D Plot Group 9**.

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

3 Select the **x-axis label** check box. In the associated text field, type `Radius ri (m)`.

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

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

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

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

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

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

Table Graph 2

1 Right-click **Results>1D Plot Group 9>Table Graph 1** and choose **Duplicate**.

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

3 From the **Table** list, choose **Table 2**.

Table Graph 3

1 Right-click **Table Graph 2** and choose **Duplicate**.

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

3 From the **Table** list, choose **Table 4**.

4 In the **1D Plot Group 9** toolbar, click **Plot**.