Created in COMSOL Multiphysics 6.2

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.

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.

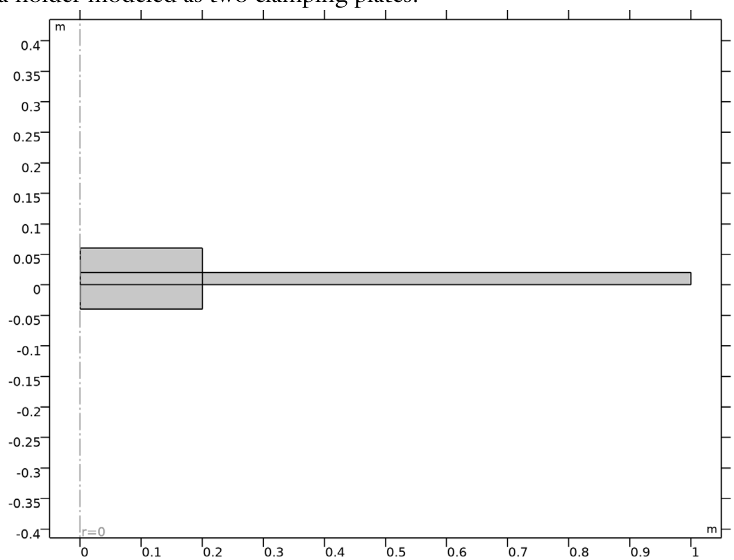


Figure 1: Geometry consisting of a large thin plate clamped between two thicker plates.

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 eigenfrequency 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.

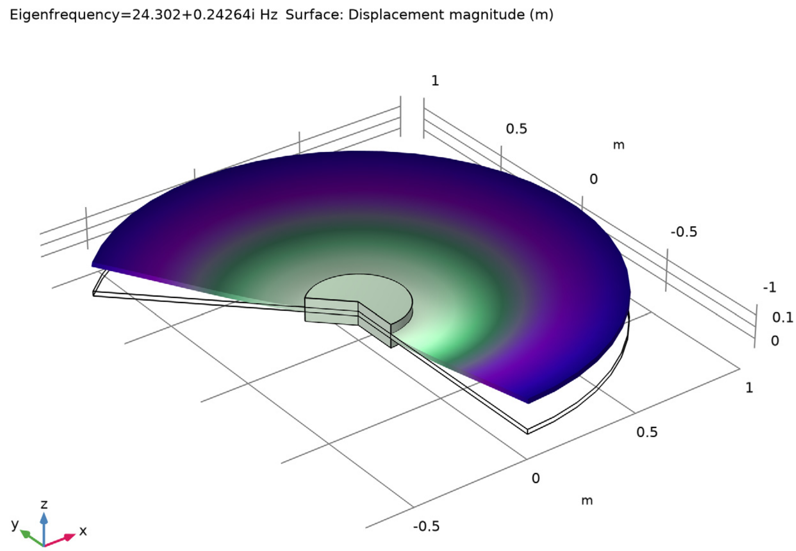


Figure 2: The first eigenmode for a plate clamped using contact. The plate holder radius is 0.2 m.

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

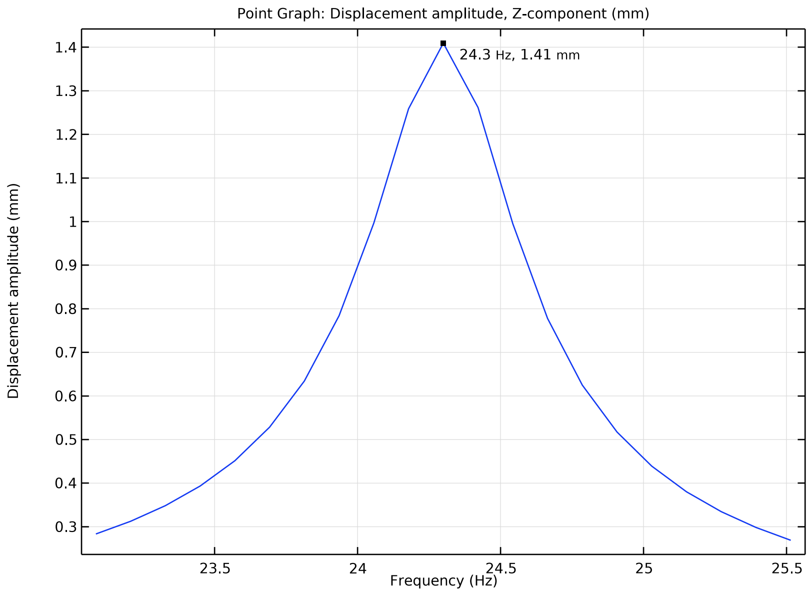


Figure 3: Vertical displacement of the plate outer edge.

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

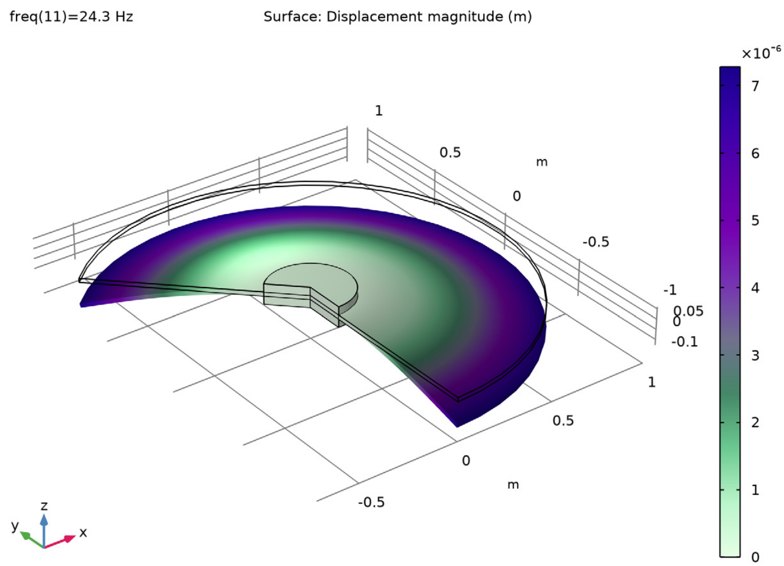


Figure 4: Response of the plate to a boundary load excitation at a frequency close to the first eigenfrequency.

The variation in the first natural frequency with respect to a change in the plate holder radius is shown in Figure 5. The results computed by means of the frequency domain analysis present a very good estimate of the eigenfrequency.

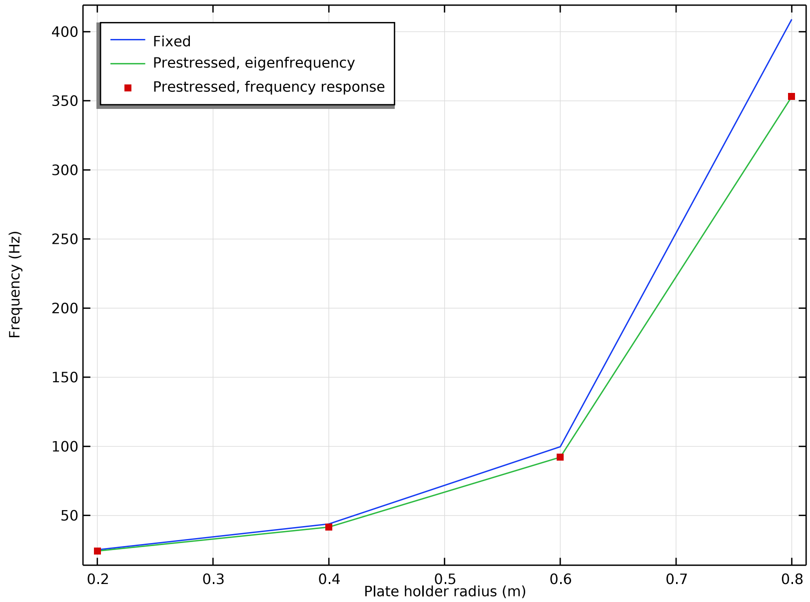


Figure 5: The first natural frequency as a function of the plate holder radius. The curves represent the results from the eigenfrequency analysis (annular constrained and squeezed plate) and from the maximum frequency response, respectively.

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.

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

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`.

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

Rectangle 3 (r3)

1 In the **Model Builder** window, click **Rectangle 3 (r3)**.

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 **Settings** window for **Contact**, locate the **Contact Method** section.
2. From the list, choose **Augmented Lagrangian**.
3. Locate the **Contact Pressure Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

Roller 1

1. In the **Model Builder** window, right-click **Solid Mechanics (solid)** 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. From the **Displacement in z direction** list, choose **Prescribed**.
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

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.

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

Add a predefined plot of the mode shape.

ADD PREDEFINED PLOT

1 In the **Home** 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/Solution 1 (sol1)>Solid Mechanics>Mode Shape, 3D (solid)**.

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

Mode Shape, Fixed

In the **Settings** window for **3D Plot Group**, type `Mode Shape, Fixed` in the **Label** text field. The value of the real part should be close to 25.2 Hz.

ADD STUDY

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

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 **Model Builder** window, click **Study 2**.

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

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

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

ADD PREDEFINED PLOT

1 In the **Home** 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 2/Solution 2 (sol2)>Solid Mechanics>Mode Shape, 3D (solid)**.

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

Mode Shape, Prestressed

In the **Settings** window for **3D Plot Group**, type `Mode Shape, Prestressed` in the **Label** text field. The value of the real part should be close to 24.3 Hz, which is slightly lower than for the fixed annular plate.

SOLID MECHANICS (SOLID)

Next, prepare to perform a frequency response analysis.

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**.

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
freq_ref	24.3[Hz]	24.3 Hz	Reference frequency

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>Frequency Domain, Prestressed**.

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

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

STUDY 3

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: Frequency-Domain Perturbation

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

1 In the **Model Builder** window, click **Step 2: Frequency-Domain Perturbation**.

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

3 In the **Frequencies** text field, type `freq_ref*range(0.95,5e-3,1.05)`.

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 **Model Builder** window, click **Study 3**.

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

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

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

RESULTS

For comparison with the previously computed eigenmodes, modify the predefined stress plot to show the displacement at the reference frequency.

ADD PREDEFINED PLOT

1 In the **Home** 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 3/Solution 4 (sol4)>Solid Mechanics>Stress, 3D (solid)**.
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

Displacement, Frequency Response

1 In the **Settings** window for **3D Plot Group**, type `Displacement, Frequency Response` in the **Label** text field.
2 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **24.3**.

Surface 1

1 In the **Model Builder** window, expand the **Displacement, Frequency Response** 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, Frequency Response** toolbar, click **Plot**.

ADD PREDEFINED PLOT

The contact forces at the linearization point can be inspected by adding a predefined plot.

1 In the **Home** 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 2/Solution Store 1 (sol3)>Solid Mechanics>Contact Forces (solid)**.
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

Contact Forces (solid)

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

Displacement Amplitude

Create a frequency response curve of the displacement amplitude at a point on the outer edge of the plate.

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Displacement Amplitude` in the **Label** text field.
3 Locate the **Data** section. 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)`.
6 Select the **y-axis label** check box. In the associated text field, type `Displacement amplitude (mm)`.

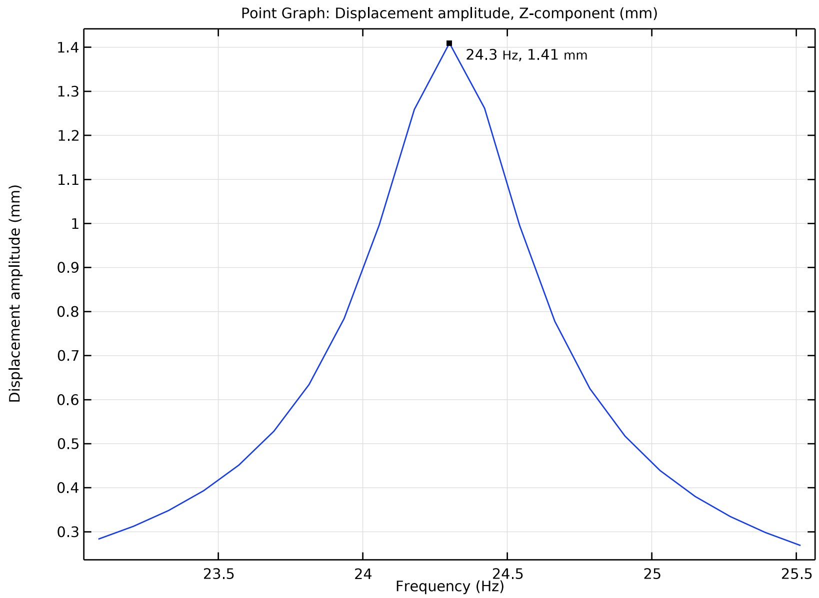
Point Graph 1

1 Right-click **Displacement Amplitude** 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 **Displacement Amplitude** toolbar, click **Plot**.



STUDY 1

Next, extend all studies with a **Parametric Sweep** to study how the radius of the clamping plate `ri` affects the natural frequency of the plate.

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
ri (Plate holder radius)	`range(0.2,0.2,0.8)`	`m`

5 In the table, click to select the cell at row number 1 and column number 3.

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

RESULTS

Add an **Evaluation Group** to extract the natural frequency.

Eigenfrequencies, Fixed

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

2 In the **Settings** window for **Evaluation Group**, type `Eigenfrequencies, Fixed` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol6)**.

4 Click to expand the **Format** section. From the **Include parameters** list, choose **Off**.

Global Evaluation 1

1 Right-click **Eigenfrequencies, Fixed** and choose **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
`ri`	`m`	`Plate holder radius`
`real(freq)`	`Hz`	`Frequency`

4 In the **Eigenfrequencies, Fixed** toolbar, click **Evaluate**.

STUDY 2

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
ri (Plate holder radius)	`range(0.2,0.2,0.8)`	`m`

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

RESULTS

Eigenfrequencies, Fixed

In the **Model Builder** window, under **Results** right-click **Eigenfrequencies, Fixed** and choose **Duplicate**.

Eigenfrequencies, Prestressed

1 In the **Model Builder** window, under **Results** click **Eigenfrequencies, Fixed 1**.

2 In the **Settings** window for **Evaluation Group**, type `Eigenfrequencies, Prestressed` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 2 (sol11)**.

4 In the **Eigenfrequencies, Prestressed** toolbar, click **Evaluate**.

GLOBAL DEFINITIONS

In order to add a parametric sweep to the prestressed frequency response study, we first need to redefine the reference frequency `freq_ref` using an interpolation function to capture the change in natural frequency with the holder plate radius.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **Result table**.

4 From the **Table from** list, choose **Eigenfrequencies, Prestressed**.

5 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
freq_ref	1

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

7 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
freq_ref	Hz

8 In the **Argument** table, enter the following settings:

Argument	Unit
Column 1	m

Parameters 1

1 In the **Model Builder** window, 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
`freq_ref`	`freq_ref(ri)`	24.302 Hz	`Reference frequency`

STUDY 3

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
ri (Plate holder radius)	`range(0.2,0.2,0.8)`	`m`

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

RESULTS

Displacement Amplitude

1 In the **Model Builder** window, under **Results** click **Displacement Amplitude**.

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

3 From the **Dataset** list, choose **Study 3/Parametric Solutions 3 (sol16)**.

Point Graph 1

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

2 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 Find the **Include** subsection. Clear the **Point** check box.

5 In the **Displacement Amplitude** toolbar, click **Plot**.

Eigenfrequencies, Frequency Response

Finally, create a plot to show the dependence of the natural frequency on `ri` for the three studies. First, find the frequency that yields the maximum displacement amplitude in the prestressed frequency response study.

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

2 In the **Settings** window for **Evaluation Group**, type `Eigenfrequencies, Frequency Response` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3/ Parametric Solutions 3 (sol16)**.

4 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

5 From the **Concatenation** list, choose **Vertical**.

Point Evaluation 1

1 Right-click **Eigenfrequencies, Frequency Response** and choose **Point Evaluation**.

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

3 From the **Dataset** list, choose **Study 3/Parametric Solutions 3 (sol16)**.

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

5 In the **Parameter values (ri (m))** list, select **0.2**.

6 Select Point 10 only.

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

Expression	Unit	Description
`solid.uAmpZ`	`mm`	`Displacement amplitude, Z-component`

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

9 Select the **Include parameters** check box.

10 In the **Eigenfrequencies, Frequency Response** toolbar, click **Evaluate**.

11 Right-click **Point Evaluation 1** and choose **Duplicate**.

Point Evaluation 2

1 In the **Model Builder** window, click **Point Evaluation 2**.

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

3 In the **Parameter values (ri (m))** list, select **0.4**.

4 Right-click **Point Evaluation 2** and choose **Duplicate**.

Point Evaluation 3

1 In the **Model Builder** window, click **Point Evaluation 3**.

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

3 In the **Parameter values (ri (m))** list, select **0.6**.

4 Right-click **Point Evaluation 3** and choose **Duplicate**.

Point Evaluation 4

1 In the **Model Builder** window, click **Point Evaluation 4**.
2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
3 In the **Parameter values (ri (m))** list, select **0.8**.

Eigenfrequencies, Frequency Response

1 In the **Model Builder** window, click **Eigenfrequencies, Frequency Response**.
2 In the **Eigenfrequencies, Frequency Response** toolbar, click **= Evaluate**.

Natural Frequencies

Use the data in the three **Evaluation Groups** to plot the natural frequency as a function of the plate holder radius using **Table Graphs**.

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Natural Frequencies` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **None**.
4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Eigenfrequencies, Fixed

1 Right-click **Natural Frequencies** and choose **Table Graph**.
2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Source** list, choose **Evaluation group**.
4 In the **Label** text field, type `Eigenfrequencies, Fixed`.
5 Click to expand the **Legends** section. Select the **Show legends** check box.
6 From the **Legends** list, choose **Manual**.
7 In the table, enter the following settings:

Legends
`Fixed`

8 Right-click **Eigenfrequencies, Fixed** and choose **Duplicate**.

Eigenfrequencies, Prestressed

1 In the **Model Builder** window, under **Results>Natural Frequencies** click **Eigenfrequencies, Fixed 1**.
2 In the **Settings** window for **Table Graph**, type `Eigenfrequencies, Prestressed` in the **Label** text field.

3 Locate the **Data** section. From the **Evaluation group** list, choose **Eigenfrequencies, Prestressed**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Prestressed, eigenfrequency`

5 Right-click **Eigenfrequencies, Prestressed** and choose **Duplicate**.

Eigenfrequencies, Frequency Response

1 In the **Model Builder** window, under **Results>Natural Frequencies** click **Eigenfrequencies, Prestressed 1**.

2 In the **Settings** window for **Table Graph**, type `Eigenfrequencies, Frequency Response` in the **Label** text field.

3 Locate the **Data** section. From the **x-axis data** list, choose **Plate holder radius (m)**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, select **Frequency (Hz)**.

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

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

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

Legends
`Prestressed, frequency response`

9 In the **Natural Frequencies** toolbar, click **Plot**.