



Model created in COMSOL Multiphysics 6.4

# Vibration of a Squeezed Plate

## Introduction

---

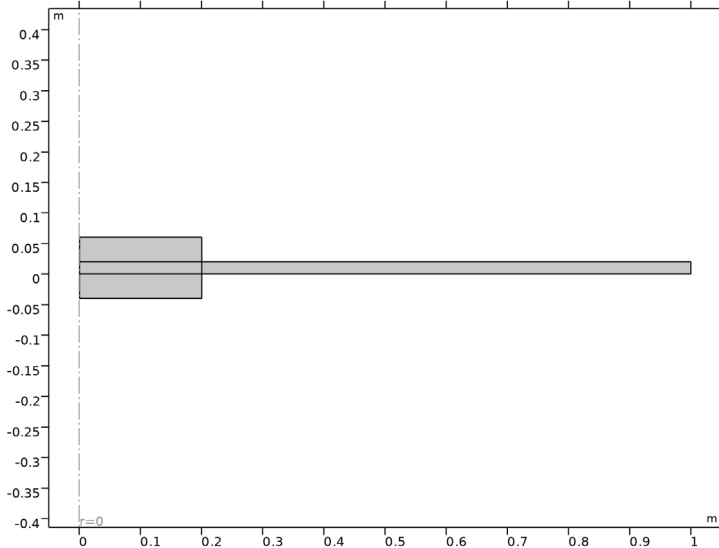
This model illustrates how to perform eigenfrequency and frequency response analyses for a contact problem.

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  $r_i = 0.2$  m, the first natural frequency and the corresponding eigenmode are shown in [Figure 2](#). The eigenfrequency is complex valued because of the damping 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.

Eigenfrequency=24.301+0.24259i Hz Displacement magnitude (m)

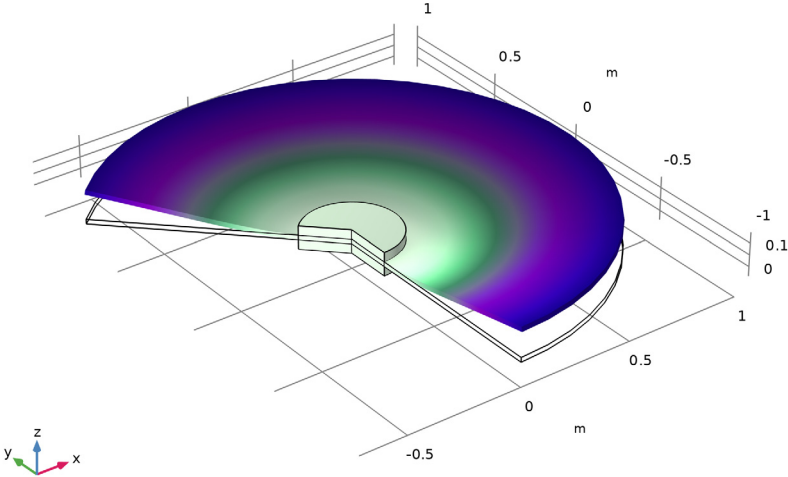
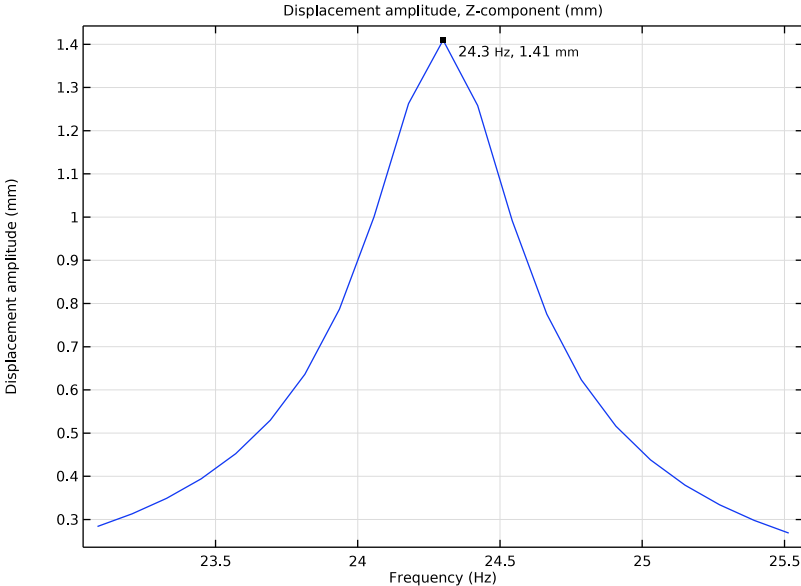


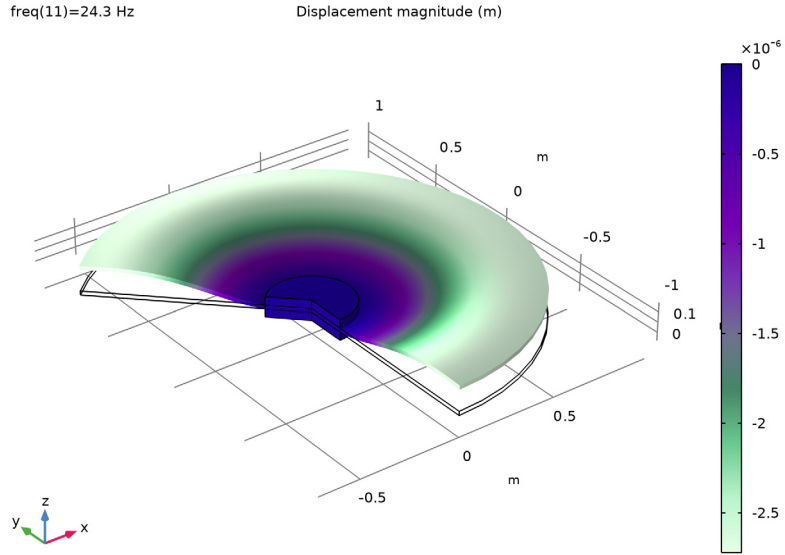
Figure 2: The first eigenmode for a plate clamped using contact. The plate holder radius is 0.2 m.

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.

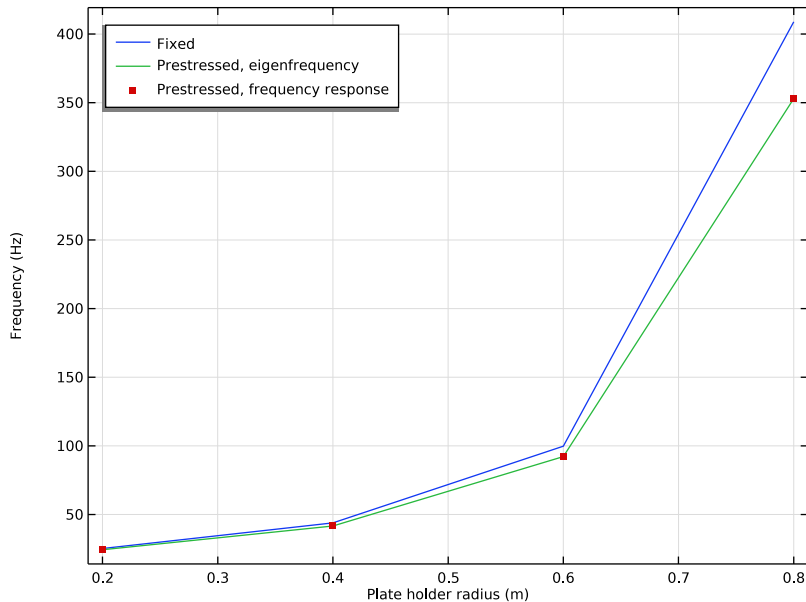


Figure 5: The first natural frequency as a function of the plate holder radius. The curves represent the results from the eigenfrequency analysis (annular constrained and squeezed plate) and from the maximum frequency response, respectively.

### 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 I*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 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 <sup>2</sup> ]	1000 N/m <sup>2</sup>	Applied harmonic load
r0	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 I


### *Rectangle I (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 r0.
- 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** checkbox.
- 8 Clear the **Layers on bottom** checkbox.
- 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  $r_i$ .
- 4 In the **Height** text field, type  $2 \cdot t_h$ .
- 5 Locate the **Position** section. In the **z** text field, type  $t_h$ .

#### *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  $-2 \cdot t_h$ .

#### *Form Union (fin)*

Since only small sliding between the plates is expected, you can build the geometry with a union operation and use the **Interior Contact** node in the Solid Mechanics interface to model contact between internal boundaries.


- 1 In the **Geometry** toolbar, click  **Build All**.

### **ADD MATERIAL**

- 1 In the **Materials** 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 **Materials** toolbar, click  **Add Material** to close the **Add Material** window.


### **SOLID MECHANICS (SOLID)**

#### *Interior Contact 1*


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Interior Contact**.
- 2 Select Boundaries 4 and 6 only.  
Use the **Augmented Lagrangian** method to increase the accuracy of the contact pressure.
- 3 In the **Settings** window for **Interior Contact**, locate the **Contact Method** section.

- 4 From the list, choose **Augmented Lagrangian**.
- 5 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 7 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 1–3 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 3 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, 4, 6, and 7 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** checkbox.

### *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** checkbox. In the associated text field, type 1.
- 4 In the **Study** toolbar, click  **Compute**.

## RESULTS

Add a plot of the mode shape from **Result Templates**.

### RESULT TEMPLATES

- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1) > Solid Mechanics > Mode Shape, 3D (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** 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** checkbox.
- 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** checkbox. In the associated text field, type 1.
- 4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.
- 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** checkbox.
- 10 In the **Study** toolbar, click  **Compute**.

#### **RESULT TEMPLATES**

- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study 2/Solution 2 (sol2) > Solid Mechanics > Mode Shape, 3D (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** 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 13 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 I** and choose **Harmonic Perturbation**.



## GLOBAL DEFINITIONS

### Parameters I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 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** checkbox.
- 3 In the tree, select **Component I (comp1) > Solid Mechanics (solid), Controls spatial frame > Fixed Constraint I**.
- 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** checkbox.
- 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** checkbox.
- 10 In the **Study** toolbar, click  **Compute**.

## RESULTS

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

## RESULT TEMPLATES

- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study 3/Solution 4 (sol4) > Solid Mechanics > Stress, 3D (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** 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. From the **Color table** list, choose **AuroraBorealis**.
- 5 In the **Displacement, Frequency Response** toolbar, click  **Plot**.

## RESULT TEMPLATES

The contact forces at the linearization point can be inspected by adding a plot from **Result Templates**.

- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study 2/Solution Store 1 (sol3) > Solid Mechanics > Contact Forces (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** 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 **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type Frequency (Hz).
- 6 Select the **y-axis label** checkbox. In the associated text field, type Displacement amplitude (mm).


*Point Graph 1*

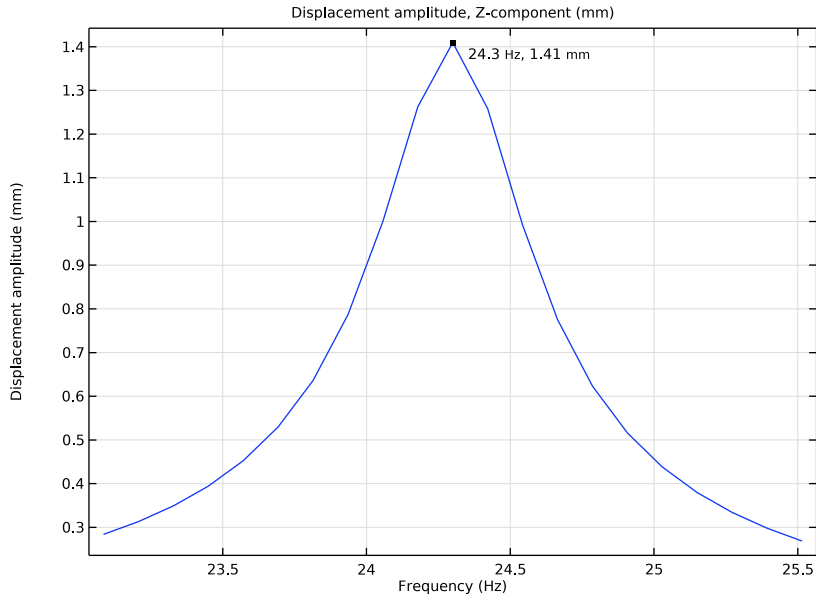
- 1 Right-click **Displacement Amplitude** and choose **Point Graph**.
- 2 Select Point 10 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Displacement >**

**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** checkbox.
- 5 Select the **Include unit** checkbox.
- 6 In the **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  $r_i$  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 **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, Prestressed*

- 1 In the **Model Builder** window, right-click **Eigenfrequencies, Fixed** and choose **Duplicate**.
- 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 (sol1)**.
- 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  **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 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 1 and column number 1.
- 6 In the **Unit** text field, type m.
- 7 In the table, click to select the cell at row number 2 and column number 1.
- 8 In the **Name** text field, type `freq_ref`.
- 9 In the **Unit** text field, type Hz.
- 10 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Nearest neighbor**.

### *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)	24.301 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 **ID 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** checkbox.
- 4 Find the **Include** subsection. Clear the **Point** checkbox.
- 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 box, 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** checkbox.
- 10 In the **Eigenfrequencies, Frequency Response** toolbar, click  **Evaluate**.

*Point Evaluation 2*

- 1 Right-click **Point Evaluation 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
- 3 In the **Parameter values (ri (m))** list box, select **0.4**.


*Point Evaluation 3*

- 1 Right-click **Point Evaluation 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
- 3 In the **Parameter values (ri (m))** list box, select **0.6**.

*Point Evaluation 4*


- 1 Right-click **Point Evaluation 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Evaluation**, locate the **Data** section.
- 3 In the **Parameter values (ri (m))** list box, 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 **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

---

<b>Legends</b>
Fixed

---

*Eigenfrequencies, Prestressed*

- 1 Right-click **Eigenfrequencies, Fixed** and choose **Duplicate**.
- 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:

---

<b>Legends</b>
Prestressed, eigenfrequency

---

*Eigenfrequencies, Frequency Response*

- 1 Right-click **Eigenfrequencies, Prestressed** and choose **Duplicate**.
- 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 box, 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**.