Created in COMSOL Multiphysics 6.2

Bracket — Frequency-Response Analysis

Introduction

A frequency response analysis solves for the linear steady-state response of a structure when subjected to harmonic loads. The problem is solved in the frequency domain and you can set a range of frequencies at which to compute the structural response.

In this example you learn how to perform a frequency response analysis of a structure under harmonic loads, but also how to perform a frequency response analysis of a prestressed structure.

It is recommended that you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension of the model example described in the section “The Fundamentals: A Static Linear Analysis” in the *Introduction to the Structural Mechanics Module*.

The geometry is shown in Figure 1.

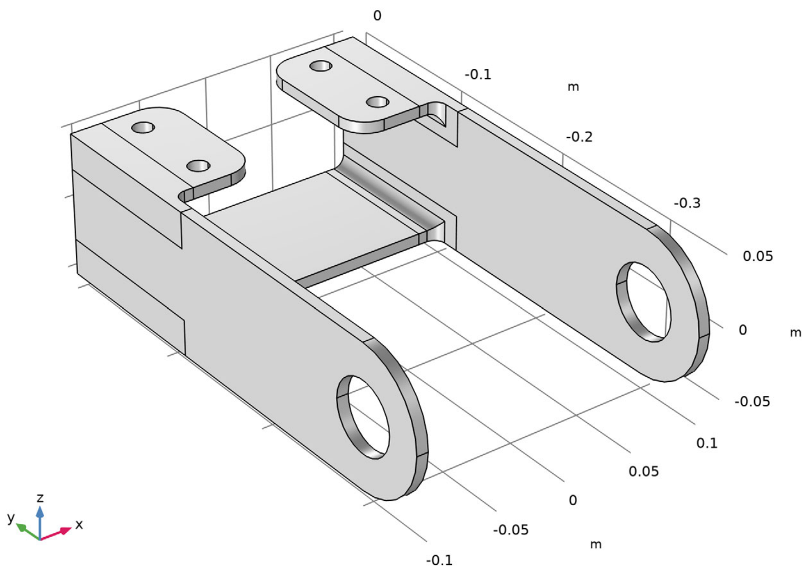

Figure 1: Bracket geometry.

You study two load cases. In the first case, a harmonic load in the X direction, with a total amplitude of 25 N, is applied to the boundaries of the bracket holes. The load is equally divided between the two arms.

The second load case consists of a combination of a static preload and the same harmonic perturbation.

An eigenfrequency analysis of this structure is performed in the tutorial Bracket — Eigenfrequency Analysis. It shows that the first resonance frequency is about 114 Hz. For the prestressed case, the eigenfrequency solution shows that the first resonance frequency is about 107 Hz when the arm is under a compressive load, and about 128 Hz when the arm is under a tensile load. In order to capture the resonance peaks properly, you can refine the frequency stepping around these values.

Results and Discussion

The default plot in a frequency-domain analysis shows the variable `<phys>.misesGp_peak`. This is a special variable that, in each point, contains the maximum von Mises stress over the whole cycle. The standard von Mises stress variable contains the stress at the current phase angle. This may be far from the peak stress, if there are significant phase shifts. In Figure 2, the stress at the last computed frequency, 750 Hz is shown. More interesting is to study the results at 114 Hz at which the first natural frequency is located. This is shown in Figure 3. Here, the peak value is around 110 MPa, to be compared with 1 MPa in the previous case.

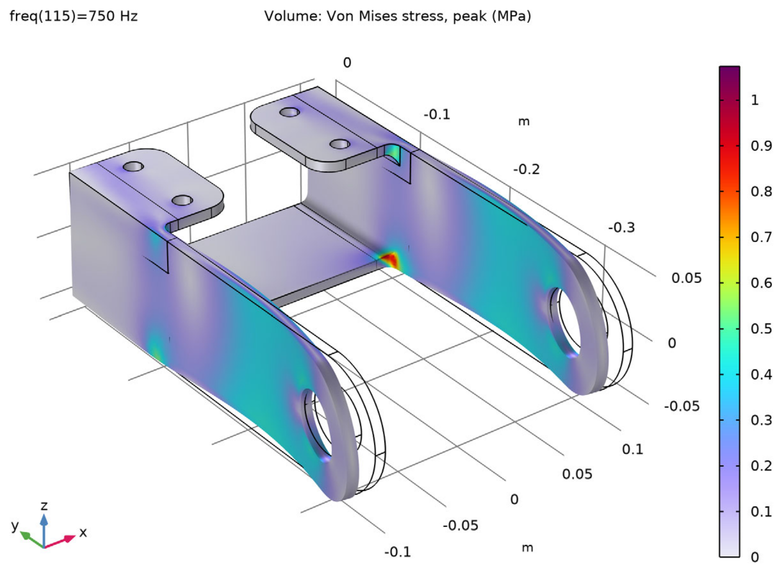

Figure 2: von Mises stress at 750 Hz.

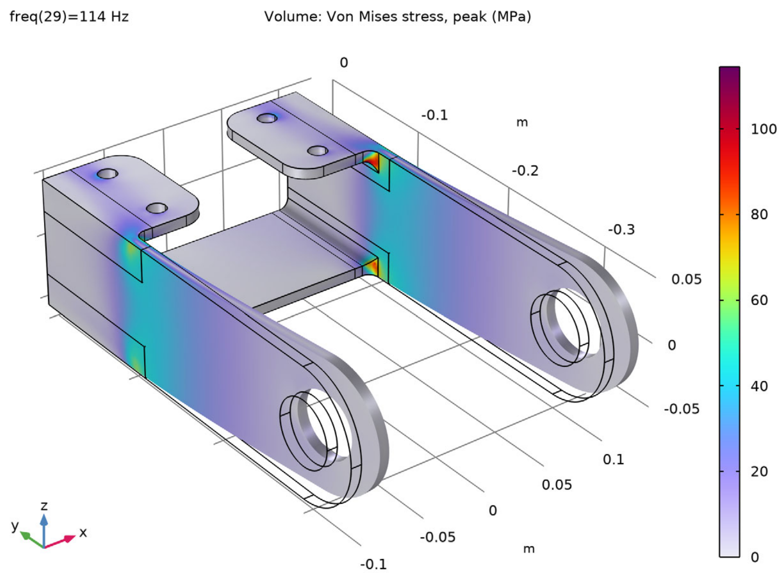

Figure 3: von Mises stress at 114 Hz.

[Figure 4](#) shows the root mean square of the displacement at the tip of the arms of the bracket around the first resonance for both the pure harmonic load case and the combined harmonic and static load cases.

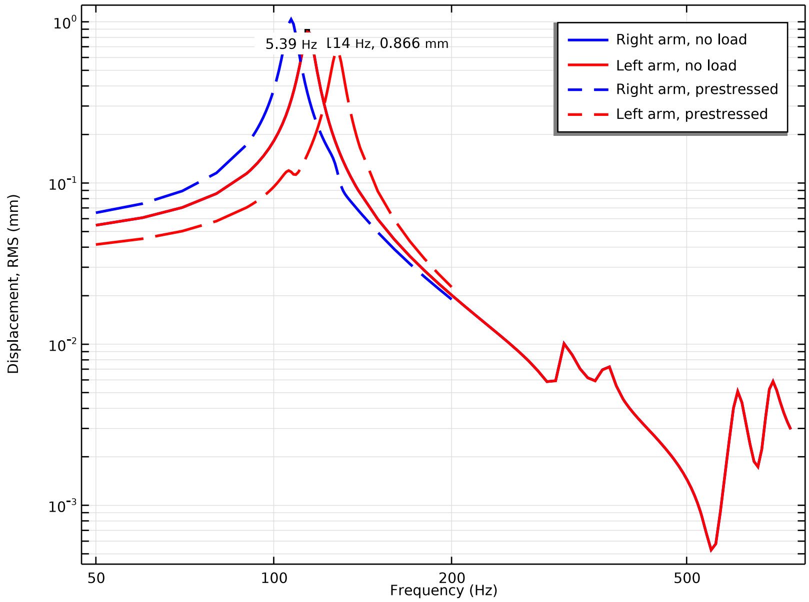

Figure 4: Root mean square of the displacement at the tip of the left (red) and right (blue) arms for both pure harmonic loaded case (solid) and a combined static and harmonic loaded case (dashed).

The curves show resonance peaks around 114 Hz for the unloaded structure in both bracket arms and a frequency shift for the loaded structure. These results are in agreement with the values predicted by the eigenfrequency solution. The curves for the left and right arms coincide as long as there is no prestress.

You can also verify that the deformation remains small even around the resonance frequency. Thus, the linearity assumption within the frequency-domain studies is fulfilled.

Figure 5 shows the phase of the x-displacement at the tips of both arms.

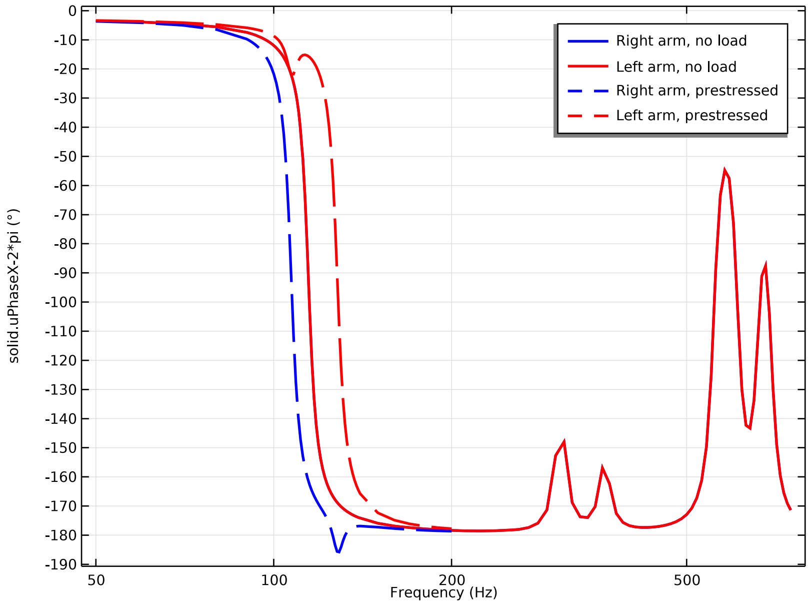

Figure 5: Phase of x-displacement at the tip of the bracket right arm.

Note the smooth transition where the displacement is in phase with the load at lower frequencies and in counterphase for higher frequencies. This is an effect of the damping, where a 5% loss factor is used. The prestressed load case solution shows interesting properties where the phase flips at different frequencies in each arm. This can be interpreted so that the two arms move synchronously for low and high frequencies, but against each other for intermediate frequencies.

In Figure 6 and Figure 7, the perturbation of the von Mises stress is shown at 107 Hz and 128 Hz. This result is the linearized deviation from the constant stress caused by the static preload, and thus the values are both positive and negative. Each arm dominates the response close to its own eigenfrequency.

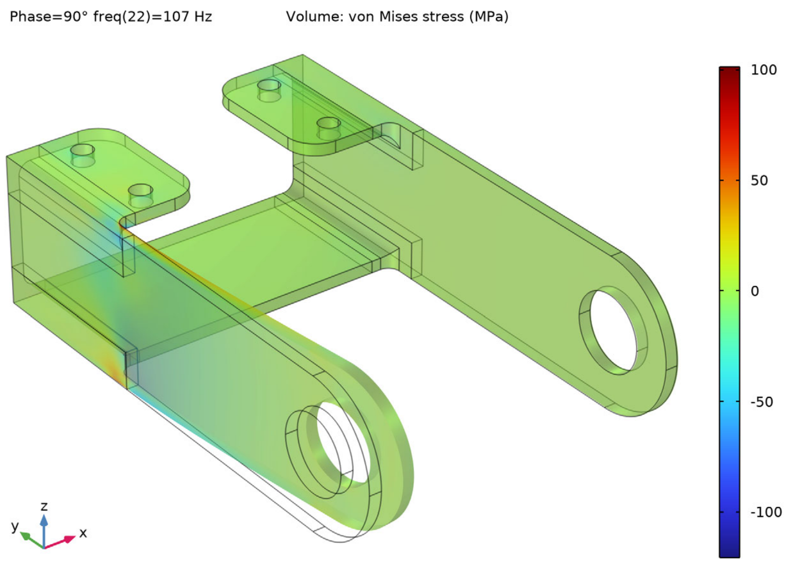

Figure 6: Perturbation in von Mises stress at first eigenfrequency, 107 Hz.

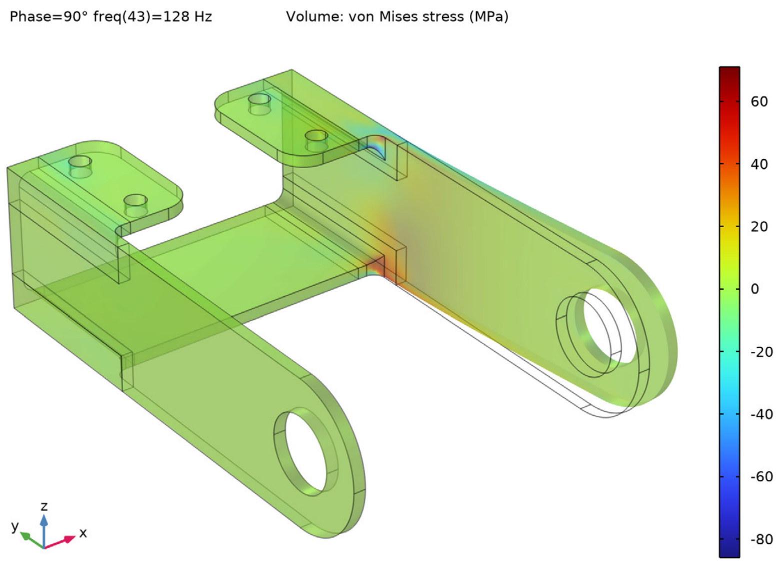

Figure 7: Perturbation in von Mises stress at second eigenfrequency, 128 Hz.

Figure 8 shows the root mean square of the x-component of the velocity of the arm of the bracket over the whole solved frequency range for the analysis without prestress as a one-third octave band plot. The band centered at 630 Hz shows a local maximum related to the second flexural mode of the arms.

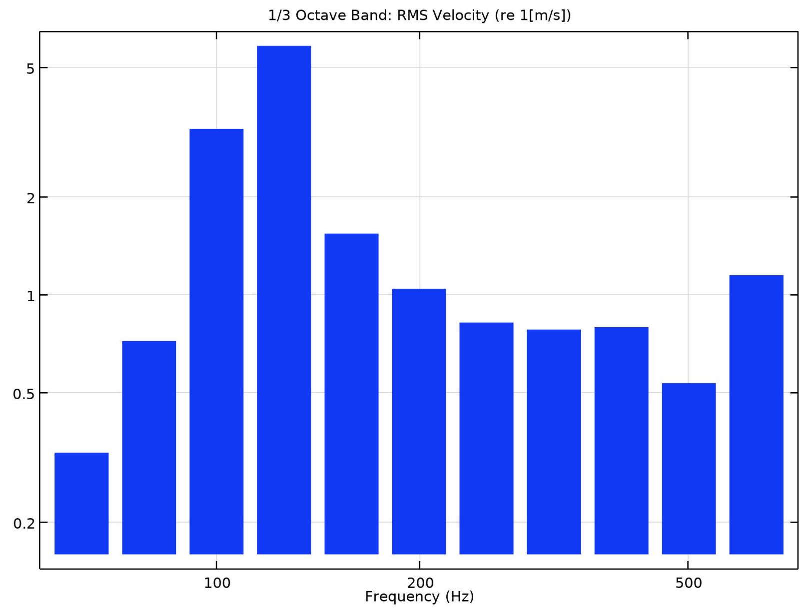

Figure 8: One-third octave band plot of the x-component of the velocity at the arm tip.

Notes About the COMSOL Implementation

For structural mechanics physics interfaces in COMSOL Multiphysics, there are six predefined study types available for frequency-response analysis: **Frequency Domain**; **Frequency Domain Modal**; **Frequency Domain, Prestressed**; **Frequency Domain, Prestressed, Modal**; **Frequency Domain, Modal Reduced-Order Model**; and **Frequency Domain, AWE Reduced-Order Model**;

The modal analysis uses the modal solver to compute the frequency response. This analysis type speeds up the computation significantly when compared to the regular frequency-domain analysis if the number of frequencies is large. In this example, the modal solver is used in the first study, and the direct solver in the second study. This is purely for comparison. If the modal solver had been selected also for the second study, it would run more than 10 times faster.

Use the prestressed frequency-response analysis when a structure is subjected to both static and harmonic loads, and the stiffness induced by the static load case can affect the structural response to the harmonic load.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_frequency`

Modeling Instructions

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **Structural Mechanics Module>Tutorials>bracket_basic** in the tree.
3. Click **Open**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

Damping 1

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

   In the frequency domain you can use loss factor damping, viscous damping, or Rayleigh damping. For this example, use loss factor 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, expand the **Component 1 (comp1)>Materials** node, then 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	0.05	1	Basic

You can now apply an external harmonic load to the bracket arms.

SOLID MECHANICS (SOLID)

Boundary Load, Harmonic

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. In the **Settings** window for **Boundary Load**, type `Boundary Load, Harmonic` in the **Label** text field.
3. Locate the **Boundary Selection** section. From the **Selection** list, choose **Pin Holes**.
4. Locate the **Force** section. From the **Load type** list, choose **Total force**.
5. Specify the $\mathbf{F}_{tot}$ vector as

25[N]	x

To define a harmonic load in the frequency domain modal analysis, you need to mark the load as being a harmonic perturbation.

6. Right-click **Boundary Load, Harmonic** 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 **Preset Studies for Selected Physics Interfaces>Frequency Domain, Modal**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Eigenfrequency

1. In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
2. Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `12`.

   For a mode superposition, it is the undamped eigenvalues that should be used. Disable the damping in this study.

3. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
4. In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1>Damping 1**.
5. Right-click and choose **Disable**.

Step 2: Frequency Domain, Modal

The frequency range will be 50 Hz–750 Hz with a refined frequency sweep step between 90 Hz and 140 Hz.

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

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

3 In the **Frequencies** text field, type `range(50,10,90) range(91,1,139) range(150,10,750)`.

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

RESULTS

Stress (solid)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

The default plot group shows the stress distribution on a deformed geometry for the final frequency (750 Hz). You can change the frequency for the plot evaluation in the **Parameter value** list in the settings for the plot group.

Volume 1

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

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

3 From the **Unit** list, choose **MPa**.

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

Stress (solid)

Plot the stresses at 114 Hz too.

1 In the **Model Builder** window, click **Stress (solid)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **114**.

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

Plot the root mean square of the displacement at the tip of the left arm of the bracket.

Displacement, RMS

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, RMS` in the **Label** text field.

3 Locate the **Plot Settings** section.

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

Frequency response curves are often presented using a logarithmic scale.

5 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

6 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

Point Graph 1

1 Right-click **Displacement, RMS** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.disp_rms`.

5 From the **Unit** list, choose **mm**.

6 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Blue**.

7 From the **Width** list, choose **2**.

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

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Right arm, no load`

Displacement, RMS

1 In the **Model Builder** window, click **Displacement, RMS**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Point Graph 2

1 Right-click **Displacement, RMS** and choose **Point Graph**.

2 Select Point 109 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.disp_rms`.

5 From the **Unit** list, choose **mm**.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 From the **Width** list, choose **2**.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Left arm, no load`

11 In the **Displacement, RMS** toolbar, click **Plot**.

Add a marker showing the peak value, and at which frequency it occurs.

Graph Marker 1

1. In the **Model Builder** window, 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. In the **Display precision** text field, type `3`.
5. Select the **Show x-coordinate** check box.
6. Select the **Include unit** check box.
7. In the **Displacement, RMS** toolbar, click **Plot**.

Generate an 1/3 octave band plot of the RMS of the x-component of the velocity at the tip of the left arm of the bracket.

Velocity, X-component RMS 1/3 Octave

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Velocity, X-component RMS 1/3 Octave` in the **Label** text field.
3. Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `1/3 Octave Band: RMS Velocity (re 1[m/s])`.
5. Locate the **Axis** section. Select the **x-axis log scale** check box.
6. Select the **y-axis log scale** check box.
7. Click the **Show More Options** button in the **Model Builder** toolbar.
8. In the **Show More Options** dialog box, in the tree, select the check box for the node **Results>All Plot Types**.
9. Click **OK**.

Octave Band 1

1. In the **Velocity, X-component RMS 1/3 Octave** toolbar, click **More Plots** and choose **Octave Band**.
2. Select Point 2 only.

3 In the **Settings** window for **Octave Band**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `abs(solid.u_tX)/sqrt(2)`.

5 From the **Expression type** list, choose **General (non-dB)**.

6 In the **Reference expression** text field, type `1[m/s]`.

7 Locate the **Plot** section. From the **Band type** list, choose **1/3 octave**.

8 In the **Velocity, X-component RMS 1/3 Octave** toolbar, click **Plot**.

You will now consider a static load applied to the bracket and perform a prestressed frequency domain analysis.

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 Click **Add Study** in the window toolbar.

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

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
P0	30[MPa]	3E7 Pa	Peak load intensity

DEFINITIONS

Analytic 1 (an1)

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

2 In the **Settings** window for **Analytic**, type `load` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

4 In the **Arguments** text field, type `F, py, px`.

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

Argument	Unit
F	Pa
py	m
px	m

6 In the **Function** text field, type `Pa`.

SOLID MECHANICS (SOLID)

Boundary Load, Prestress

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

Apply a boundary load to the bracket holes.

2 In the **Settings** window for **Boundary Load**, type `Boundary Load, Prestress` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pin Holes**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Z,Y-PinHoleY)*(sign(X)*(Y-PinHoleY)<0)`	n

The default boundary system is in the deformed configuration. This would make the load behave as a follower load when used in a geometrically nonlinear context. Change to a fixed coordinate system.

DEFINITIONS

Boundary System 1 (sys1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Boundary System 1 (sys1)**.

2 In the **Settings** window for **Boundary System**, locate the **Settings** section.

3 From the **Frame** list, choose **Reference configuration**.

STUDY 2

Step 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 2** 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 `range(50,10,90) range(91,1,139) range(140,10,200)`.

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

RESULTS

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

You have previously created a point graph plot for the unloaded case. Add a new point graph plot to the same figure but use the dataset of the second load case.

Point Graph 1, Point Graph 2

1 In the **Model Builder** window, under **Results>Displacement, RMS**, Ctrl-click to select **Point Graph 1** and **Point Graph 2**.

2 Right-click and choose **Duplicate**.

Point Graph 3

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

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

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

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

Legends
`Right arm, prestressed`

Estimate the damping from the response using the full width at half maximum (FWHM) method. Since this method is based on a power definition, the corresponding level for an amplitude variable is the peak value divided by $\sqrt{2}$. Note that a table containing the damping values is automatically generated.

Graph Marker 1

1 In the **Model Builder** window, expand the **Point Graph 3** node, then click **Graph Marker 1**.

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

3 From the **Display mode** list, choose **Bandwidth**.

4 From the **Cutoff mode** list, choose **Relative to peak**.

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

Point Graph 4

1 In the **Model Builder** window, under **Results>Displacement, RMS** click **Point Graph 4**.

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

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

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

Legends
`Left arm, prestressed`

6 In the **Displacement, RMS** toolbar, click **Plot**.

Volume 1

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

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

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

4 From the **Unit** list, choose **MPa**.

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

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

7 Click **OK**.

What is shown here is the deviation from the prestress value. This is why the von Mises stress can be negative.

Stress (solid), Prestressed

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

2 In the **Settings** window for **3D Plot Group**, type `Stress (solid), Prestressed` in the **Label** text field.

Exactly at a resonance, there will be a phase shift of 90 degrees between load and displacement. In order to see the highest stresses, it is necessary to plot the results at another phase angle.

3 Locate the **Phase** section. From the **Solution at angle (phase)** list, choose **Manual**.

4 In the **Phase** text field, type `90`.

Investigate the stress distribution around the resonances.

5 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **107**.

6 Click the **Transparency** button in the **Graphics** toolbar.

7 Click the **Show Grid** button in the **Graphics** toolbar.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

9 From the **Parameter value (freq (Hz))** list, choose **128**.

10 In the **Stress (solid), Prestressed** toolbar, click **Plot**.

Now plot the phase shift with respect to the applied load phase.

Displacement, RMS

In the **Model Builder** window, right-click **Displacement, RMS** and choose **Duplicate**.

Displacement Phase, X Component

1 In the **Model Builder** window, expand the **Results>Displacement, RMS 1** node, then click **Displacement, RMS 1**.

2 In the **Settings** window for **1D Plot Group**, type `Displacement Phase, X Component` in the **Label** text field.

3 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

Point Graph 1

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

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

3 In the **Expression** text field, type `solid.uPhaseX-2*pi`.

4 From the **Unit** list, choose **°**.

Graph Marker 1

1 In the **Model Builder** window, expand the **Point Graph 1** node.

2 Right-click **Graph Marker 1** and choose **Delete**.

Point Graph 2

1 In the **Model Builder** window, under **Results>Displacement Phase, X Component** click **Point Graph 2**.

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

3 In the **Expression** text field, type `solid.uPhaseX-2*pi`.

4 From the **Unit** list, choose **°**.

Point Graph 3

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

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

3 In the **Expression** text field, type `solid.uPhaseX-2*pi`.

4 From the **Unit** list, choose **°**.

Graph Marker 1

1 In the **Model Builder** window, expand the **Point Graph 3** node.

2 Right-click **Graph Marker 1** and choose **Delete**.

Point Graph 4

1 In the **Model Builder** window, under **Results>Displacement Phase, X Component** click **Point Graph 4**.

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

3 In the **Expression** text field, type `solid.uPhaseX-2*pi`.

4 From the **Unit** list, choose **°**.