Created in COMSOL Multiphysics 6.0

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.

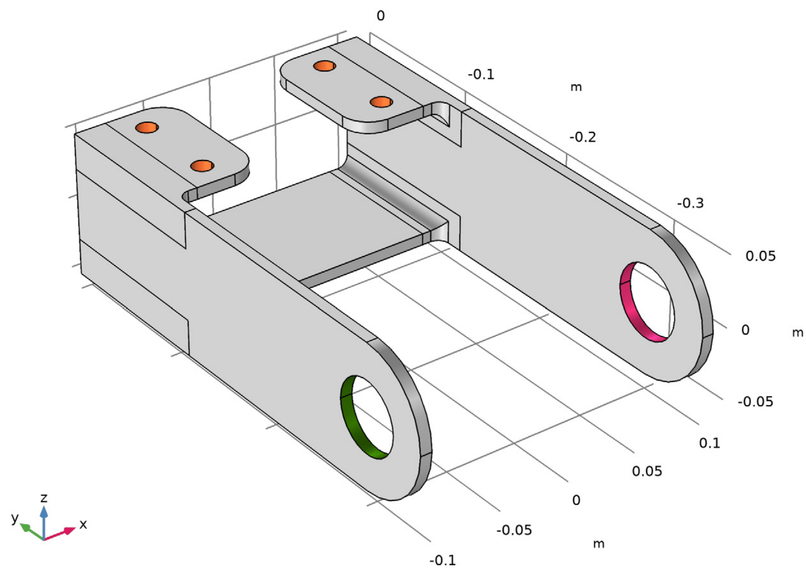

Figure 1: Bracket geometry

You study two load cases. In the first case, a harmonic load in the X-direction, with a an amplitude of 10 kPa, is applied to the boundaries of the bracket holes. 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>.mises_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, `<phys>.mises`, 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, 200 Hz is shown. More interesting is to look at the results at 114 Hz at which the first natural frequency is located. This is shown in Figure 3. Here, the peak value is 110 MPa, to be compared with 3 MPa in the previous case.

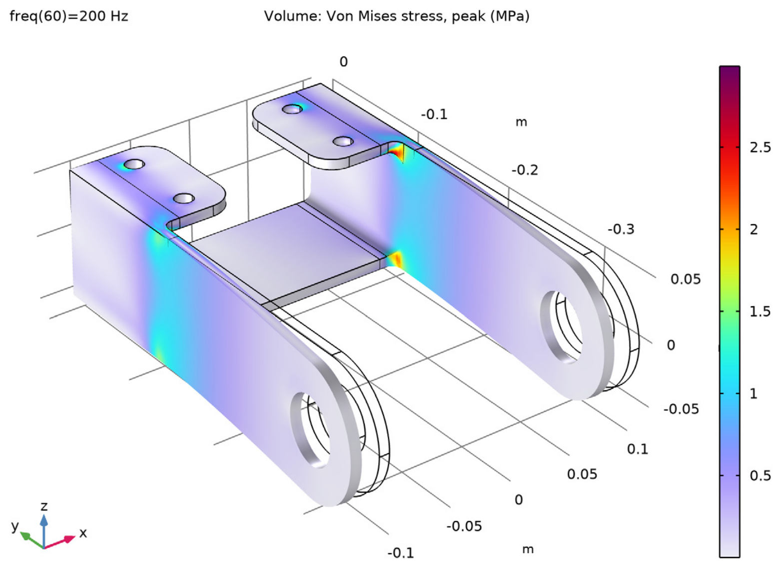

Figure 2: von Mises stress at 200 Hz.

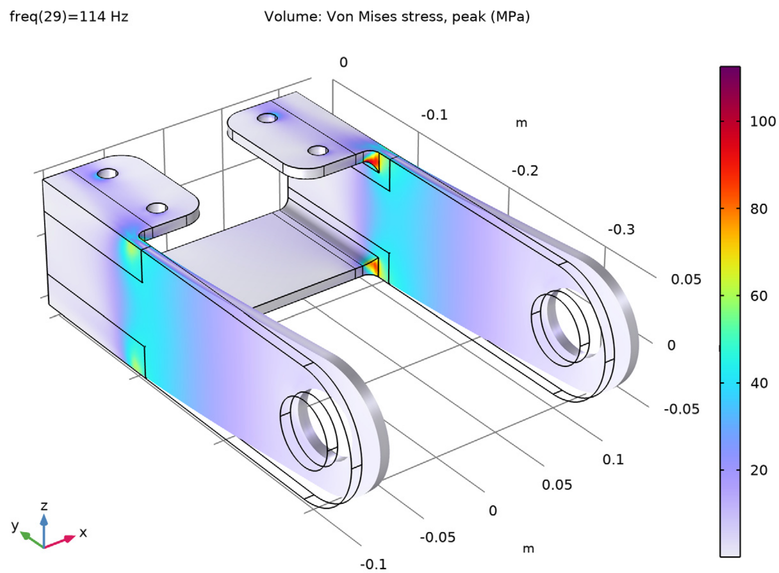

Figure 3: von Mises stress at 114 Hz.

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

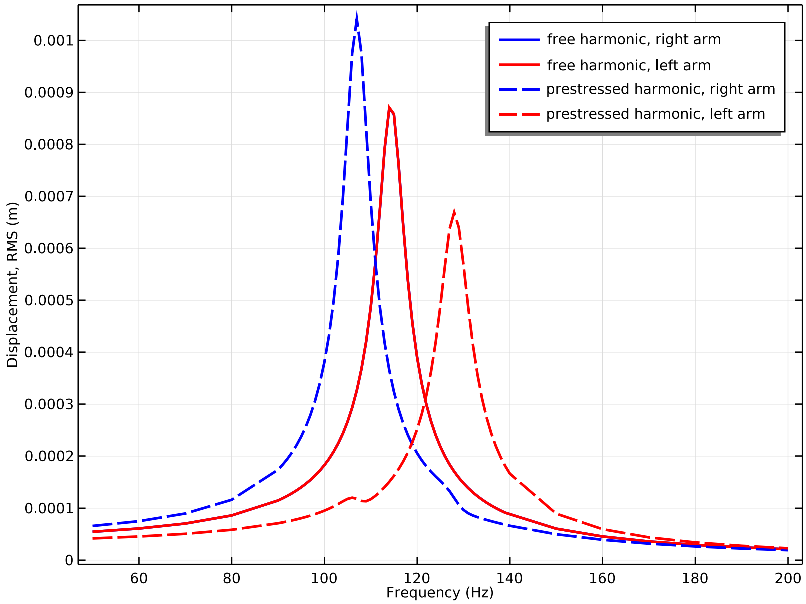

Figure 4: Root mean square of the displacement at the tip of the left (red) and right (blue) arms for both pure harmonic loaded case (solid) and a combined static and harmonic loaded case (dashed).

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 solving with the eigenfrequency solver. 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.

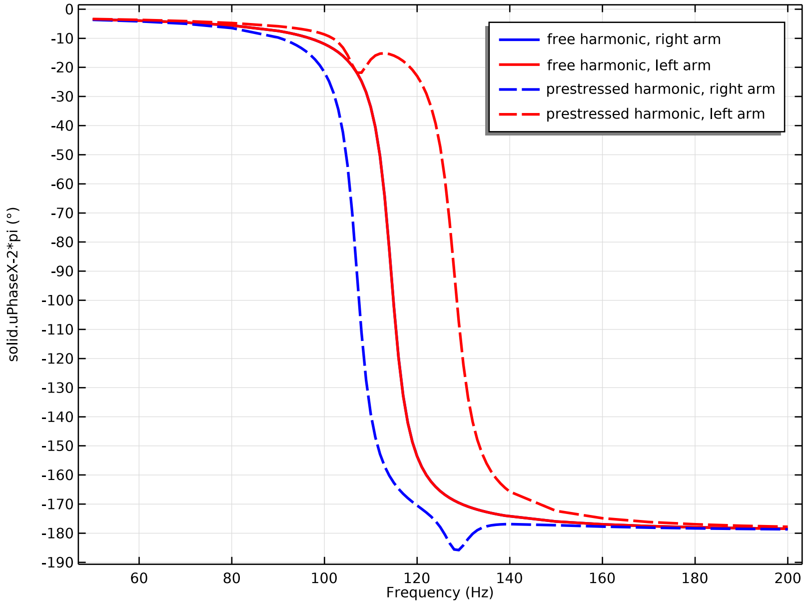

Figure 5: Phase of x-displacement at the tip of the bracket right arm.

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 using a 5% loss factor. 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 an d 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.

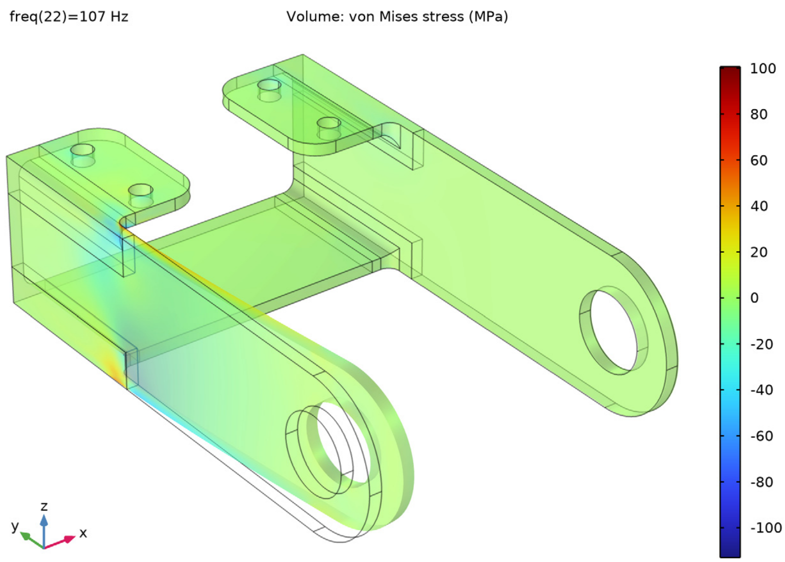

Figure 6: Perturbation in von Mises stress at first eigenfrequency, 108 Hz.

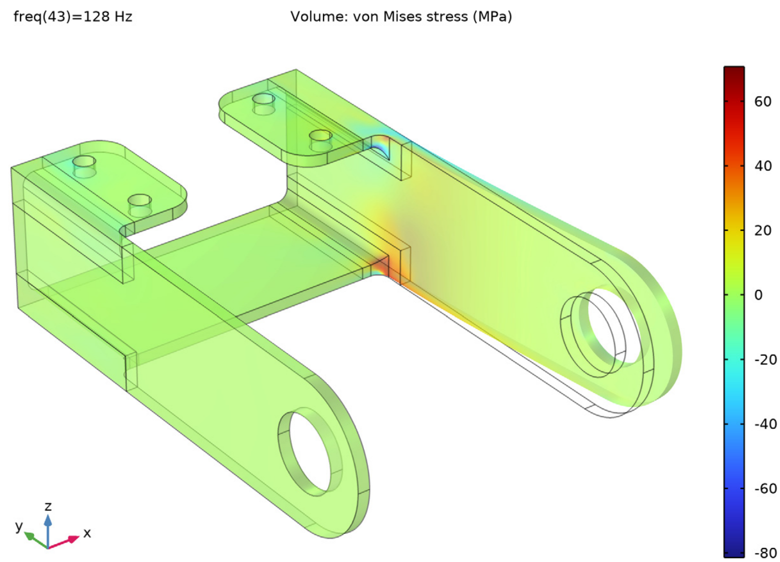

Figure 7: Perturbation in von Mises stress at first eigenfrequency, 108 Hz.

Notes About the COMSOL Implementation

For a structural mechanics physics interface in COMSOL Multiphysics, there are four predefined study types available for frequency-response analysis: **Frequency Domain**; **Frequency-Domain Modal**; **Frequency Domain Prestressed**; and **Frequency Domain Prestressed, Modal**.

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 wold 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. Select Boundaries 4, 5, 75, and 76 only.
4. Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

10[kPa]	x
0	y
0	z

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

5. 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.
3. In the associated text field, type 2.

Step 2: Frequency Domain, Modal

The frequency range will be 50 Hz–190 Hz with a refined frequency sweep step between 100 Hz and 130 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 `50 70 90 range(100,1,130) 150 170 190`.

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. You can change the frequency for the plot evaluation in the **Parameter value** list in the settings for the plot group.

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. Select the **x-axis label** check box.

4 In the associated text field, type `Frequency (Hz)`.

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

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

7 From the **Positioning** list, choose **In data points**.

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
`free harmonic, right arm`

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

6 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

7 From the **Positioning** list, choose **In data points**.

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
`free harmonic, left arm`

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

Now plot the phase shift with respect to the applied load at a specified point location.

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.

3 In the **X** text field, type `0`.

4 In the **Y** text field, type `-50e-3`.

5 In the **Z** text field, type `-50e-3`.

6 From the **Snapping** list, choose **Snap to closest boundary**.

Displacement phase, X component

1 In the **Results** toolbar, click **1D Plot Group**.

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

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

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Frequency (Hz)`.

Point Graph 1

1 Right-click **Displacement phase, X component** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 3D 1**.

4 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 phase (material and geometry frames) - rad>solid.uPhaseX - Displacement phase, X component**.

5 Locate the **y-Axis Data** section. From the **Unit** list, choose **°**.

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

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

8 In the table, enter the following settings:

Legends
`free harmonic`

9 In the **Displacement phase, X component** toolbar, click **Plot**.

In the solution dataset node, you can change the phase used to display the solution.

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
YC	-300[mm]	-0.3 m	Y coordinate of hole center

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-YC)*(sign(X)*(Y-YC)<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 `50 70 90 range(100,2,140) 150 170 190`.

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

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

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

4 Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.

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

Legends
`prestressed harmonic, right arm`

Point Graph 4

1 In the **Model Builder** window, 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 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

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

Legends
`prestressed harmonic, left arm`

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

Cut Point 3D 2

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 3D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.

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

Point Graph 2

1 In the **Model Builder** window, under **Results>Displacement phase, X component** right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Point 3D 2**.

4 In the **Displacement phase, X component** toolbar, click **Plot**.

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

Legends
`prestressed harmonic`

6 In the **Displacement phase, X component** toolbar, click **Plot**.

Stress (solid), prestressed

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

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