



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](#).

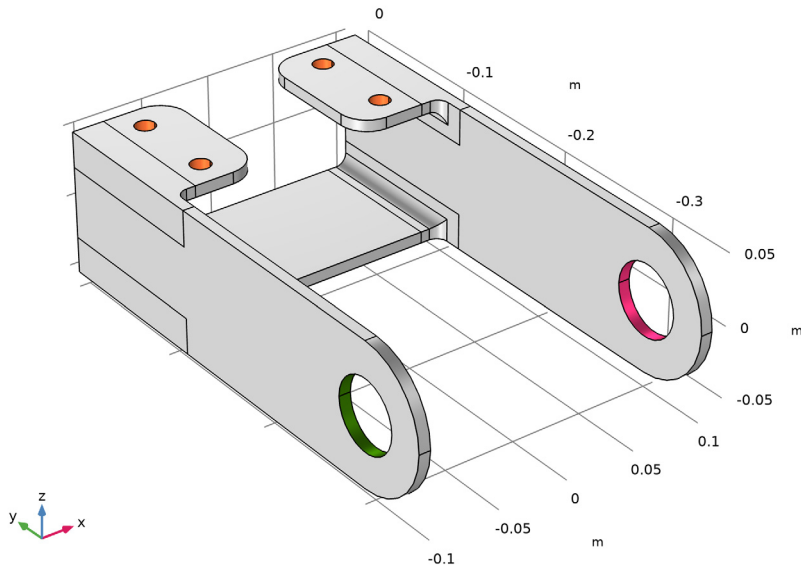


Figure 1: Bracket geometry

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 115 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

Figure 2 shows the root mean square of the displacement at the tip of the left arm of the bracket for both the pure harmonic load case and the combined harmonic and static load cases.

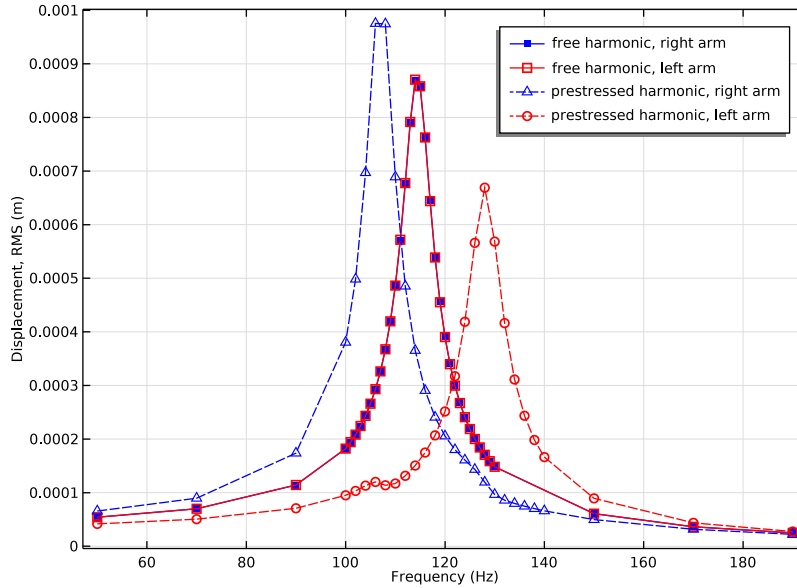


Figure 2: Root mean square of the displacement at the tip of the left (red) and right (blue) arms for both pure harmonic loaded case (solid) and a combined static and harmonic loaded case (dashed).

The curves show resonance peaks around 115 Hz for the unloaded structure in both bracket arms and the frequency shift for the loaded structure. These results are in agreement with the values predicted by solving with the eigenfrequency solver.

You can also verify that the deformation remains small even around the resonance frequency.

Figure 3 shows the phase of the x-displacement at the tip of the bracket right arm.

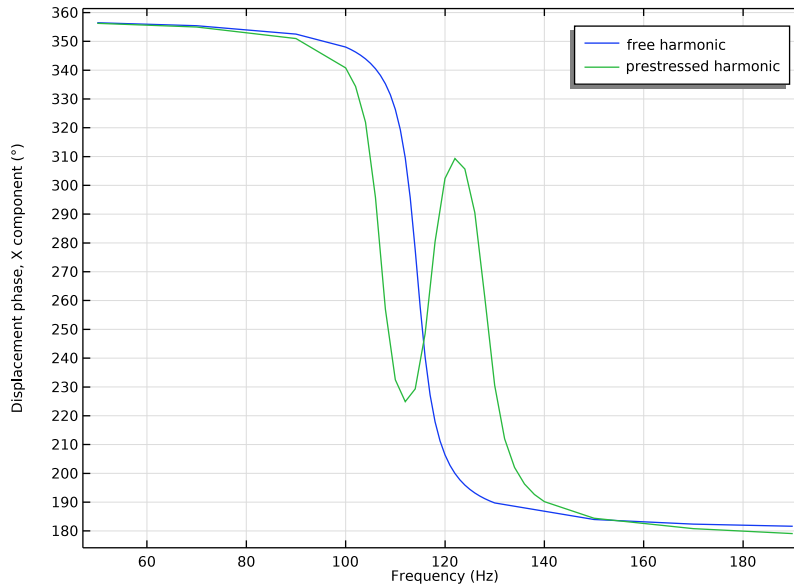


Figure 3: Phase of x -displacement at the tip of the bracket right arm.

Note the smooth transition around the resonance frequency which results from the damping using a 5% loss factor. The prestressed load case solution shows a double phase change corresponding to the two vibration modes.

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**, and **Frequency Domain Prestressed** and **Frequency Domain Prestressed, Modal**.

The modal analysis uses the modal solver to compute the frequency response. This analysis type significantly speeds up the computation when compared to the regular frequency domain analysis if the number of frequencies is large.

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

From the **File** menu, choose **Open**.


Browse to the model's Application Libraries folder and double-click the file `bracket_basic.mph`.

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 **ID Plot Group**.
- 2 In the **Settings** window for **ID 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 **ID 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 Select the **Snap to closest boundary** check box.

Displacement phase, X component

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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 $^{\circ}$.
- 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 I (anI)

- 1 In the **Home** toolbar, click  **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 $F \cdot \cos(\text{atan2}(py, \text{abs}(px)))$.
- 4 In the **Arguments** text field, type F, py, px .
- 5 Locate the **Units** section. In the **Arguments** text field, type Pa, m, 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 I (sys1)**.
- 5 Locate the **Force** section. Specify the \mathbf{F}_A vector as

0	t1
0	t2
$\text{load}(-P0, Z, Y-YC) \cdot (\text{sign}(X) \cdot (Y-YC) < 0)$	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

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

Point Graph 3

- 1 Right-click and choose **Duplicate**.
- 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 **Triangle**.
- 6 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.

