Model created in COMSOL Multiphysics 6.4

Bracket — Harmonic Vibration Fatigue

Introduction

A frequency-sweep test, or a sine-sweep test, is sometimes used to experimentally determine the resonance frequencies (natural frequencies) of a structure. The structure is typically mounted on a shaker table, and then subjected a harmonic base excitation at variable frequency. The basic premise of the test is that the sweep is performed sufficiently slowly, so that a stationary state of harmonic vibration is obtained at each frequency. In addition to determining the resonance frequencies of the structure, a frequency-sweep test can be used to ascertain whether the structure will fail during operation, particularly if the operating conditions are such, that the excitation frequencies are in close proximity to the resonance frequencies of the structure. In this example, it is shown how to perform a fatigue analysis of a bracket that undergoes base excitation during a frequency sweep. A prestress resulting from an external load is imposed before the frequency sweep is performed.

Model Definition

This model is based on the *Bracket — Static Analysis* model, found in the Structural Mechanics Module Application Library. The bracket is made of steel, and in the original model, it is constrained at the four mounting bolt holes, with static loading applied at the two arms. The static loading is the result of a pin running between the holes, with an actuator mounted on it. Figure 1 shows the stress state in the bracket under this static loading. There are stress concentrations at the mounting bolt holes, as well as at the fillets where the arms join the remaining parts of the bracket. Note that the stress concentrations at the mounting bolt holes are somewhat artificial, as they emanate from the crude approximation of the bolted joints using fixed constraints.

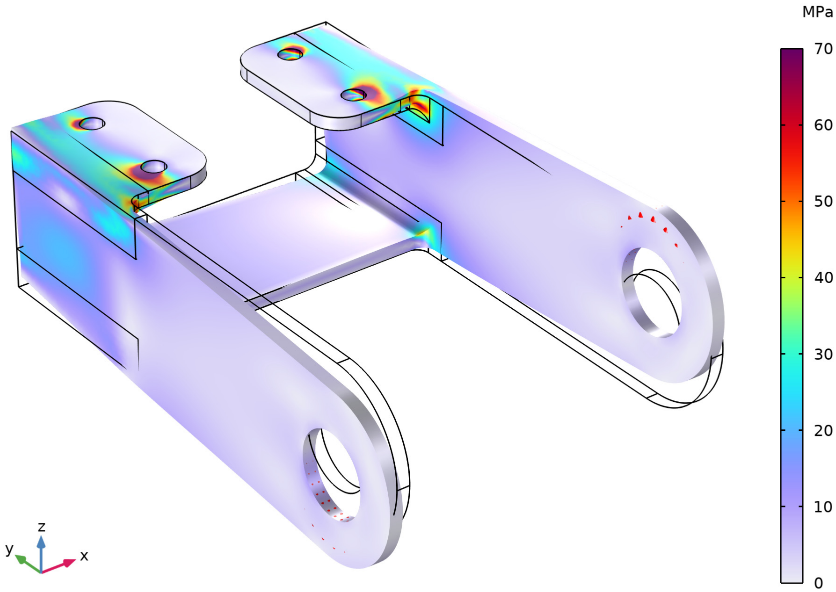

Figure 1: Stress state in the bracket when subjected to the external loading at the two arms.

In this example, we will consider the bolt mounting holes not as fixed, but instead assume that they are attached to a vibrating surrounding, such as a shaker table. We will then emulate a frequency-sweep test using a base excitation (acceleration) of 3.5g, and subsequently perform a fatigue analysis to assess the damage incurred by the frequency sweep. The effect of the prestresses will be included, which has a few consequences. First, the modal basis and the natural frequencies of the bracket are affected. Second, the inclusion of prestresses will affect the fatigue damage, as they add a mean stress state to the pure amplitude stress state resulting from harmonic excitation. Note, however, that from a dynamics point of view, replacing the pin and actuator with boundary loads may be inadequate, as the mass and inertia of these items would in fact affect the natural frequencies of the system. However, for the purpose of illustrating how to model a frequency-sweep test with a prestress, this modeling assumption is acceptable.

The frequency sweep is performed between 100 Hz and 500 Hz, with an increasing frequency, at a constant rate of 0.005 Hz/s. Of particular interest are the junctions (the fillets) between the arms and the remaining structure, as these constitute stress concentrations at which fatigue failure is thought to occur.

The discrete set of excitation frequencies that is used must be chosen carefully. The purpose of the model is to emulate a frequency-sweep test. Several resonance (natural) frequencies exist within the frequency range of the sweep test. As is well known, fatigue is highly stress dependent, so unless excitation at and around relevant resonance frequencies is included, the predicted fatigue damage will be inaccurate. In the model, we initially perform an analysis in which we consider the response at the resonance frequencies alone. This is to understand which resonance frequencies need to be resolved in greater detail during the subsequent frequency-sweep calculation.

The assessment of fatigue is made using the stresses computed in the respective frequency sweeps. The model uses a fatigue-stress measure based on a signed von Mises stress and a load-ratio-dependent S-N curve, $f_{\mathrm{SN}}(R, N)$; this measure is then used to compute damage according to the Palmgren–Miner theory.

Results and Discussion

Figure 2 shows the peak von Mises stress in the bracket as it is subjected to harmonic vibration at its six lowest (constrained) resonance frequencies. As mentioned earlier, the main concern is the junctions between the arms and the remaining structure, and the two fillets are of particular interest as they constitute stress concentrations. Figure 3 shows the peak von Mises stress specifically in these areas. We note that the stress levels at the two lowest resonance frequencies are about four times higher, or more, than the stress levels at the higher resonance frequencies. Bearing in mind that fatigue is highly stress dependent, this suggests that we need to resolve the frequency sweep around the two lowest resonance frequencies finely, but in contrast that we do not need to do this for the higher resonance frequencies. The first and second natural frequencies are both about 115 Hz, and the frequency-sweep calculation was therefore performed for the frequency range 100 Hz to 600 Hz, with closely spaced frequencies around 115 Hz.

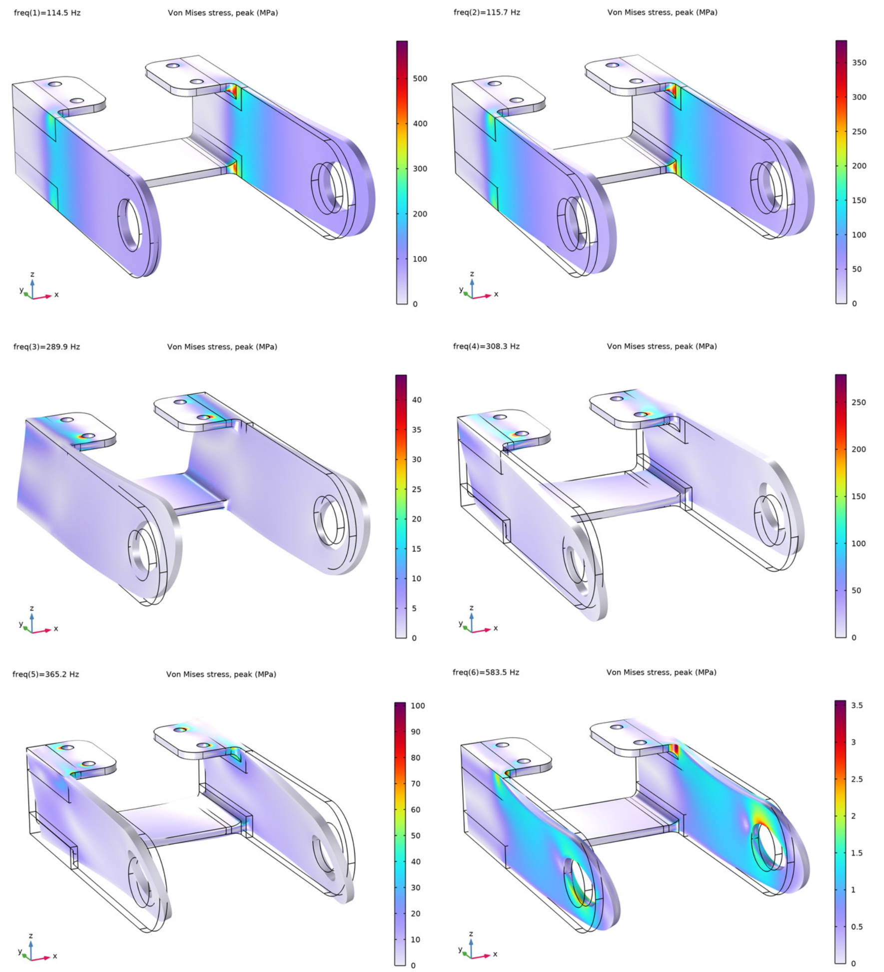

Figure 2: The peak von Mises stress when the bracket is subjected to harmonic vibration at its first six resonance frequencies.

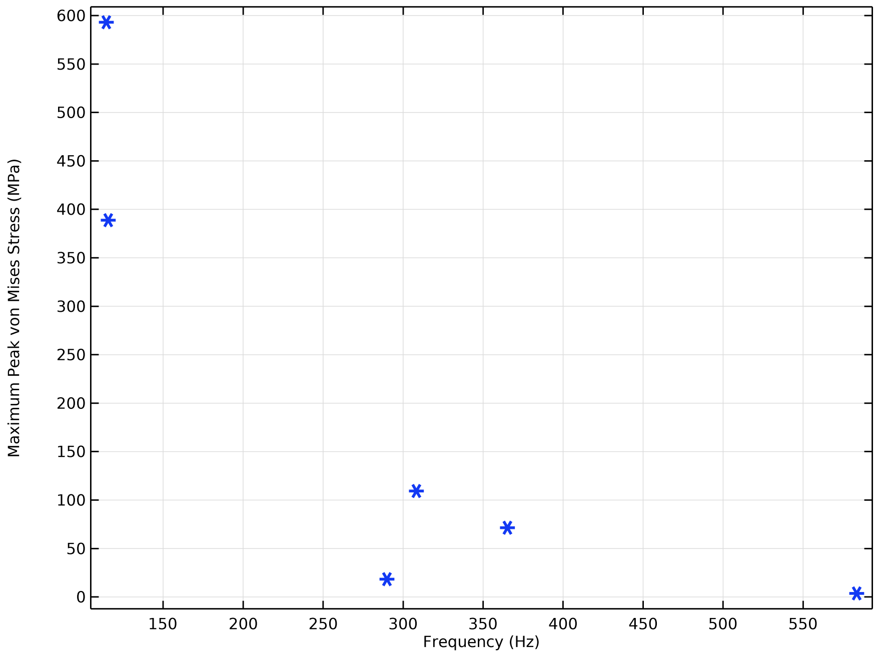

Figure 3: The peak von Mises stress in the critical regions (fillets) when the bracket is subjected to harmonic vibration at its first six resonance frequencies.

Figure 4 shows the fatigue usage factor resulting from the frequency sweep. The highest (most critical) value is about 0.29, and it represents the fraction of the fatigue life of the bracket spent during the sweep.

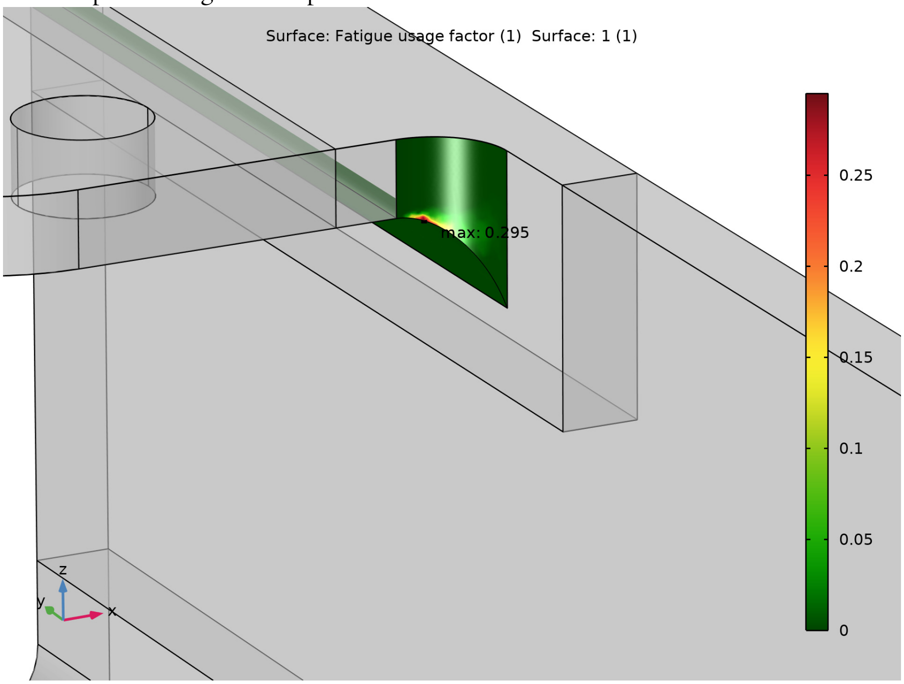

Figure 4: The calculated fatigue usage factor resulting from the frequency sweep.

Application Library path: `Fatigue_Module/Harmonic_Vibration/bracket_fatigue_harmonic_vibration`

Modeling Instructions

ROOT

In this example you will start from an existing model from the Structural Mechanics Module.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module** > **Tutorials** > **bracket_static** in the tree.

3 Click **Open**.

RESULTS

Stress (solid)

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

SOLID MECHANICS (SOLID)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Solid Mechanics (solid)** node, then 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**.

Boundary Load 1

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.

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Boundary Load 1**.

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

3 Click **Go to Source** for **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**.

SOLID MECHANICS (SOLID)

Base Excitation 1

1 In the **Physics** toolbar, click **Global** and choose **Base Excitation**.

2 Right-click **Base Excitation 1** and choose **Harmonic Perturbation**.

3 In the **Settings** window for **Base Excitation**, locate the **Base Excitation** section.

4 Specify the $\mathbf{a}_b$ vector as

3.5*g_const	x

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** > **Eigenfrequency, Prestressed**.

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

STUDY 2

Step 2: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, 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** > **Linear Elastic Material 1** > **Damping 1**.

4 Right-click and choose **Disable**.

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

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **More Studies** > **Frequency Domain, Prestressed, Modal**.

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

STUDY 3

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 Right-click and choose **Disable**.

First perform a frequency response analysis at the natural frequencies. This is to understand which frequencies must be resolved adequately in the subsequent frequency sweep computation.

Step 3: Frequency Domain, Modal

1 In the **Model Builder** window, click **Step 3: Frequency Domain, Modal**.

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

3 In the **Frequencies** text field, type `114.5 115.7 289.9 308.3 365.2 583.5`.

Solution 4 (sol4)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Modal Solver 1**.

3 In the **Settings** window for **Modal Solver**, click to expand the **Values of Linearization Point** section.

4 Select the **Store linearization point and deviation in output** checkbox.

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

DEFINITIONS

Fillets

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** > **Selections** node.

2 Right-click **Component 1 (comp1)** > **Definitions** > **Selections** and choose **Explicit**.

3 In the **Settings** window for **Explicit**, type `Fillets` in the **Label** text field.

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 24, 25, 70, and 71 only.

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Fillets**.

STUDY 3

In the **Study** toolbar, click **Update Solution**.

RESULTS

Maximum Peak von Mises Stress in Fillets

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

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

3 From the **Dataset** list, choose **Study 3/Solution 4 (sol4)**.

4 In the **Label** text field, type `Maximum Peak von Mises Stress in Fillets`.

Global 1

1 Right-click **Maximum Peak von Mises Stress in Fillets** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`maxop1(solid.misesGp_peak)`	`MPa`	`Maximum 1`

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

7 In the **Maximum Peak von Mises Stress in Fillets** toolbar, click **Plot**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **More Studies** > **Frequency Domain, Prestressed, Modal**.

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

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

STUDY 4

Step 2: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 Right-click and choose **Disable**.

An investigation of the peak von Mises stress suggests the need to resolve the frequency response around 115 Hz.

Step 3: Frequency Domain, Modal

1 In the **Model Builder** window, click **Step 3: 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(100,5,600) range(0.95,0.0025,1.05)*115`.

Solution 7 (sol7)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 7 (sol7)** node, then click **Modal Solver 1**.

3 In the **Settings** window for **Modal Solver**, locate the **Values of Linearization Point** section.

4 Select the **Store linearization point and deviation in output** checkbox.

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

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics > Fatigue (ftg)**.

4 Click the **Add to Component 1** button in the window toolbar.

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

FATIGUE (FTG)

Harmonic Vibration 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Harmonic Vibration**.

2 In the **Settings** window for **Harmonic Vibration**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Fillets**.

4 Locate the **Solution Field** section. From the **Physics interface** list, choose **Solid Mechanics (solid)**.

5 Locate the **Load History Definition** section. From the **Frequency history** list, choose **Linear frequency sweep**.

6 Locate the **Fatigue Evaluation Parameters** section. Find the **Direction** subsection. From the σ list, choose **Signed von Mises**.

GLOBAL DEFINITIONS

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 **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `bracket_fatigue_harmonic_vibration_sn_curve.txt`.

6 Click **null**.

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

Function	Unit
int1	Pa

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

Argument	Unit
Argument 1	1
Argument 2	1

FATIGUE (FTG)

Harmonic Vibration 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Fatigue (ftg)** click **Harmonic Vibration 1**.

2 In the **Settings** window for **Harmonic Vibration**, locate the **Fatigue Evaluation Parameters** section.

3 Find the **Material** subsection. From the $f_{\mathrm{SN}}(R,N)$ list, choose **int1**.

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** > **Fatigue** > **Fatigue**.

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

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

STUDY 5

Step 1: Fatigue

1 In the **Settings** window for **Fatigue**, locate the **Values of Dependent Variables** section.

2 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

3 From the **Method** list, choose **Solution**.

4 From the **Study** list, choose **Study 4, Frequency Domain, Modal**.

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