Created in COMSOL Multiphysics 6.2

Bracket — General Periodic Dynamic Analysis

Introduction

The steady-state response of a system subjected to a nonharmonic periodic excitation can be computed using two different approaches:

- The most straightforward is a time-dependent analysis in which a large number of cycles is computed so that any traces of startup transients fade away. For structures with a low damping, a fairly large number of cycles may be needed so the computational cost can be high.
- An alternative, more efficient approach, is to represent the forcing function by a Fourier series. It is then possible to compute a sequence of frequency domain solutions, one for each term in the series. In order to get the results back into the time domain, the results for each harmonic has to be superimposed over the time of a full period.

In this example, you learn how to compute the Fourier series coefficients of the periodic forcing function, and how to use them to perform a frequency response analysis. The results from the frequency response analysis is used as input to an inverse FFT analysis to get the steady-state time-varying response for a complete period. In order to validate the results from this approach, a time-dependent analysis is performed.

In both cases, a full or a modal solution scheme can be used. Compared to full time-dependent or frequency-domain methods, modal methods offer advantages with a reduced problem size if a limited number of eigenmode is excited. By representing the dynamics of the system by a few significant eigenmodes, the modal method reduces the size of problem. In this example, modal solution methods are used in both cases.

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.

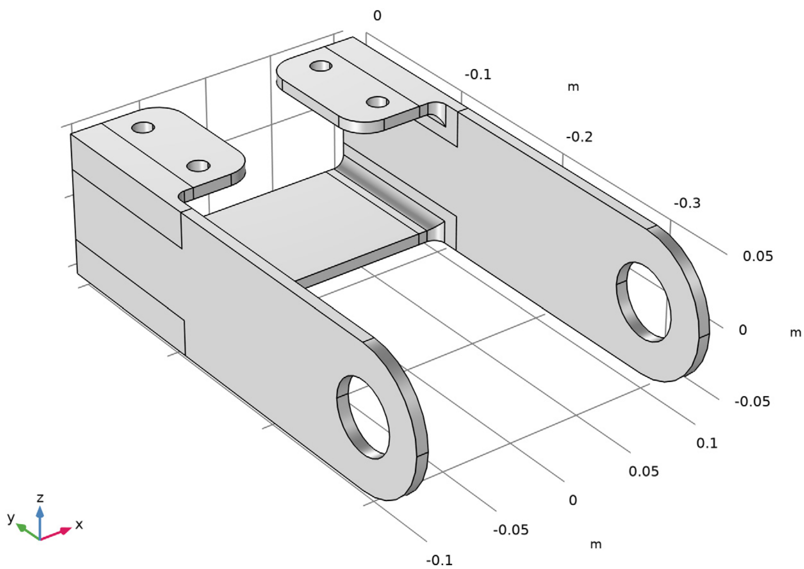

Figure 1: Bracket geometry.

The time varying periodic load is a force applied in the X direction at the bracket holes. It consists of a triangular pulse with an amplitude of 750 N amplitude and zero mean, varying with 40 Hz frequency as shown in Figure 2.

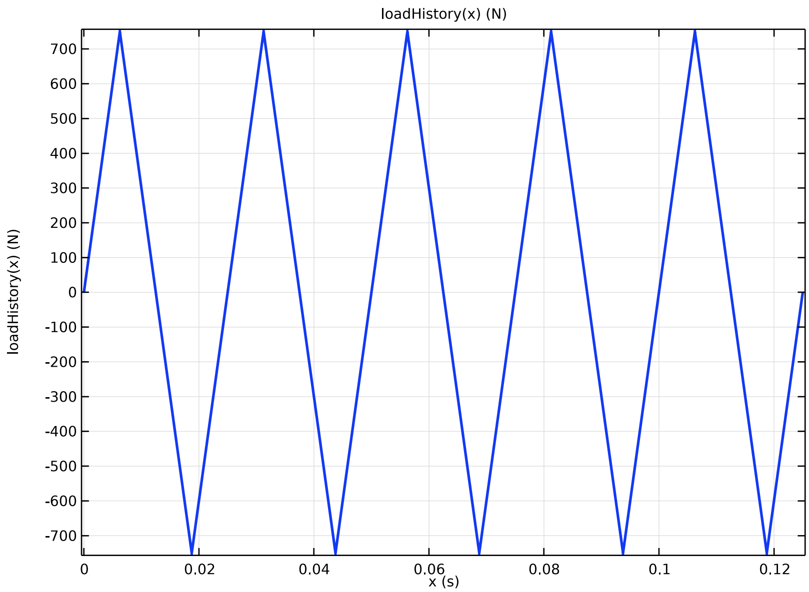

Figure 2: The load history.

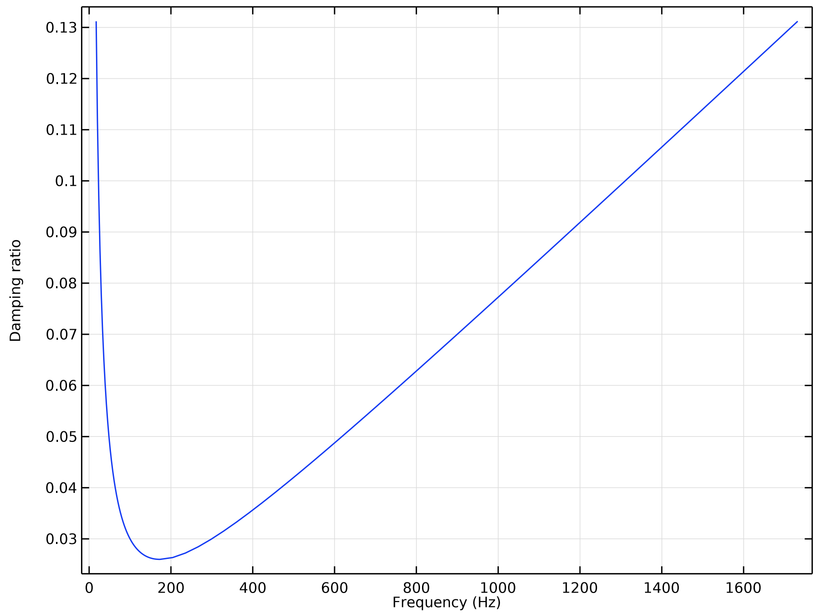

Figure 3: The damping ratio curve.

Rayleigh damping is chosen since it is applicable both in frequency and time domains. The relative damping is set to 0.03 at the frequencies 100 Hz and 300 Hz. The damping ratio curve is shown in Figure 3.

An eigenfrequency analysis of this structure is performed in the tutorial model Bracket — Eigenfrequency Analysis. It shows that the first resonance frequency is about 115 Hz. The fundamental frequency of the load, having the frequency 40 Hz, will thus mainly excite the first mode of the bracket.

FOURIER SERIES

A periodic function $F(t)$ with period T_0 (and corresponding angular frequency ω_0) can be decomposed into a discrete Fourier series of the form

$$F(t) = F_{\mathrm{a}0} + \sum_{n=1}^{\infty} (F_{\mathrm{an}} \cos(n\omega_0 t) + F_{\mathrm{bn}} \sin(n\omega_0 t)) \tag{1}$$

where

$$
\begin{aligned}
F_{\mathrm{a0}} &= \frac{1}{T}\int_0^T F(t)dt \\
F_{\mathrm{an}} &= \frac{2}{T}\int_0^T F(t)\cos(n\omega_0 t)dt \\
F_{\mathrm{bn}} &= \frac{2}{T}\int_0^T F(t)\sin(n\omega_0 t)dt
\end{aligned}
\tag{2}
$$

The periodic load in this model is a triangular function which is an odd function with zero mean. The Fourier series coefficients can be shown to be

$$
\begin{aligned}
F_{\mathrm{a0}} &= 0 \\
F_{\mathrm{an}} &= 0 \\
F_{\mathrm{bn}} &= 4A\left(\frac{1-(-1)^n}{\pi^2 n^2}\right)
\end{aligned}
\tag{3}
$$

Here A is the amplitude of the triangular function. For even values of n, the coefficients F_{bn} are zero.

Alternatively, the trigonometric functions and Fourier coefficients can be expressed on complex form:

$$
F(t) = F_{\mathrm{a0}} + \mathrm{Re}\left(\sum_{n=1}^{\infty} F_{\mathrm{n}} e^{in\omega_0 t}\right)
\tag{4}
$$

where F_{n} are complex valued. This is the notation used in COMSOL Multiphysics. The relation between the Fourier series coefficients in the two formulations are

$$
\begin{aligned}
F_{\mathrm{an}} &= \mathrm{Re}(F_{\mathrm{n}}) \\
F_{\mathrm{bn}} &= -\mathrm{Im}(F_{\mathrm{n}})
\end{aligned}
\tag{5}
$$

Results and Discussion

Since the Fourier coefficients for the given load can be determined analytically, the accuracy of the computed values can be investigated. In Figure 4, the Fourier series coefficients are plotted against the frequency.

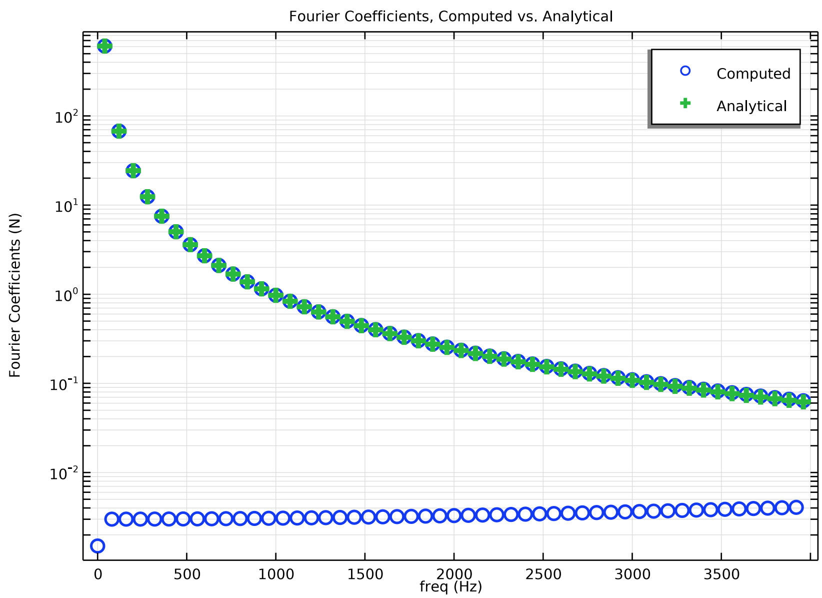

Figure 4: Numerical and analytical values of Fourier series coefficients.

As can be seen, the computed values match the values given by Equation 3 very well.

Another way of investigating the accuracy of the computed Fourier series coefficients is to compare the real and imaginary values. Since, for this loading function, all coefficients are purely imaginary, the real parts should be small. This comparison is shown in Figure 5. It can be seen that the error increases with the order of the term, but the accuracy is still good up to 2000 – 3000 Hz. The accuracy can be improved by computing a higher number of terms, that is the setting **Maximum output frequency** in the **Time to Frequency FFT** study step.

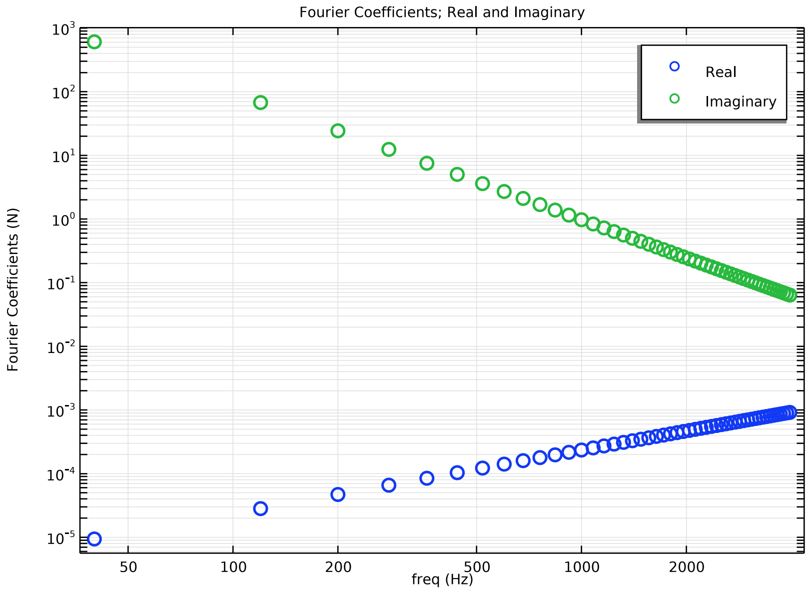

Figure 5: Real and imaginary parts of the Fourier series coefficients.

In Figure 6 and Figure 7, the von Mises stress at a certain time in the period is plotted for the two different solution methods. The results are very similar. There are two different possible sources of inaccuracy:

- The accuracy in the computation of Fourier series coefficients.
- Errors in the time integration, in particular the number of periods used for letting startup transients fade away. In this case, 21 periods are computed.

In most cases, the time-stepping algorithm will be the larger source of error.

.

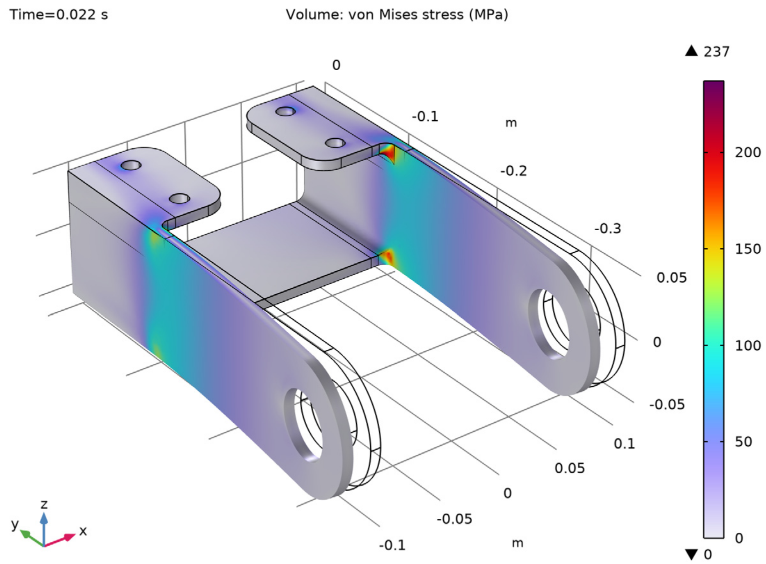

Figure 6: Equivalent stress at t = 0.022 s using the general periodic approach.

.

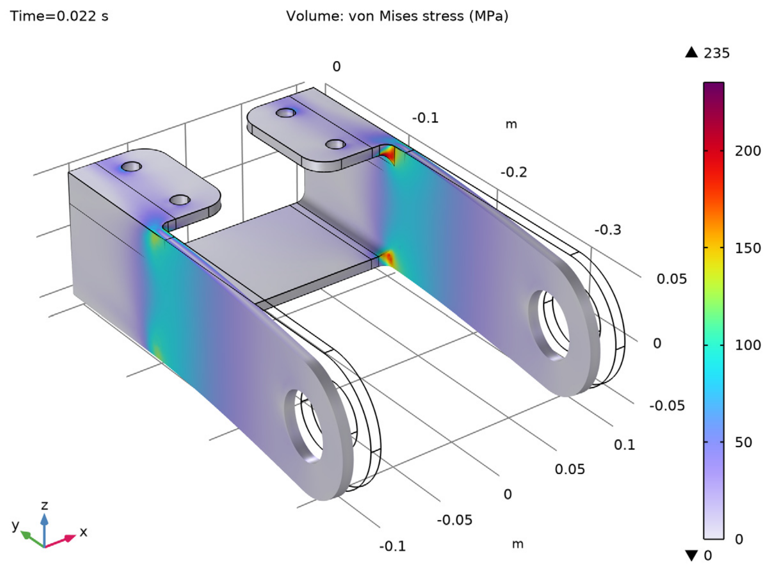

Figure 7: Equivalent stress at t = 0.022 s using a time-dependent modal approach.

In Figure 8, the contributions to the displacement from each harmonic is plotted. This is the amplitude for each frequency in the frequency sweep. It can be seen that it is mainly the Fourier terms at 40 Hz and 120 Hz that contribute. Thus, it is only some of the few terms with the very best accuracy that are dominating.

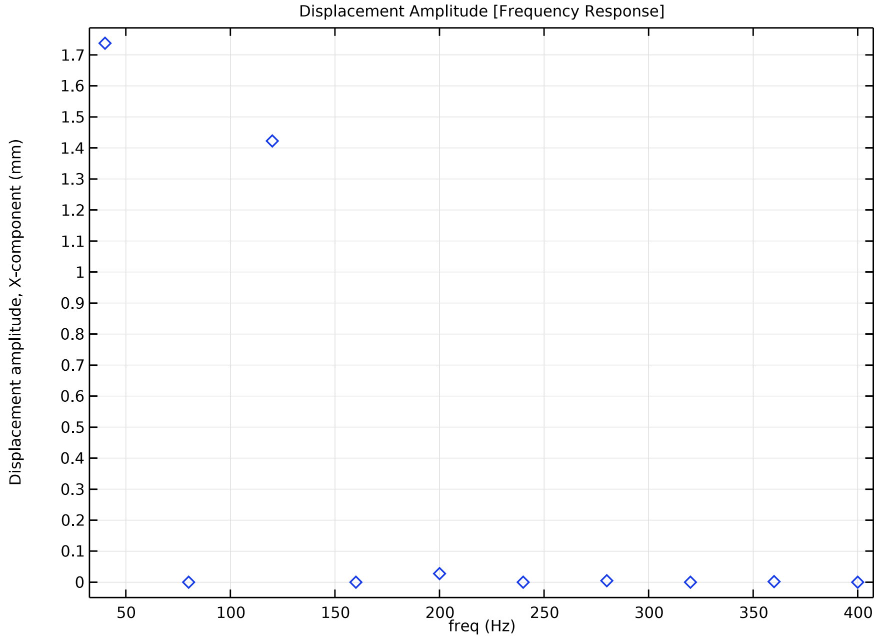

Figure 8: Amplitude contribution from each harmonic to the displacement in the X direction at the tip of the bracket.

In Figure 9, the displacement at the tip of the bracket is shown for one period. The two different solution methods are compared, and the results are almost indistinguishable.

In Figure 10, a similar graph for the σ_{xx} stress component at a point in the highly stressed fillet is shown.

In both graphs the effect of the two dominating frequencies 40 Hz and 120 Hz is clearly visible. The frequency content of the excitation is filtered by the dynamic properties of the structure, giving a response that bears little similarity to the excitation.

.

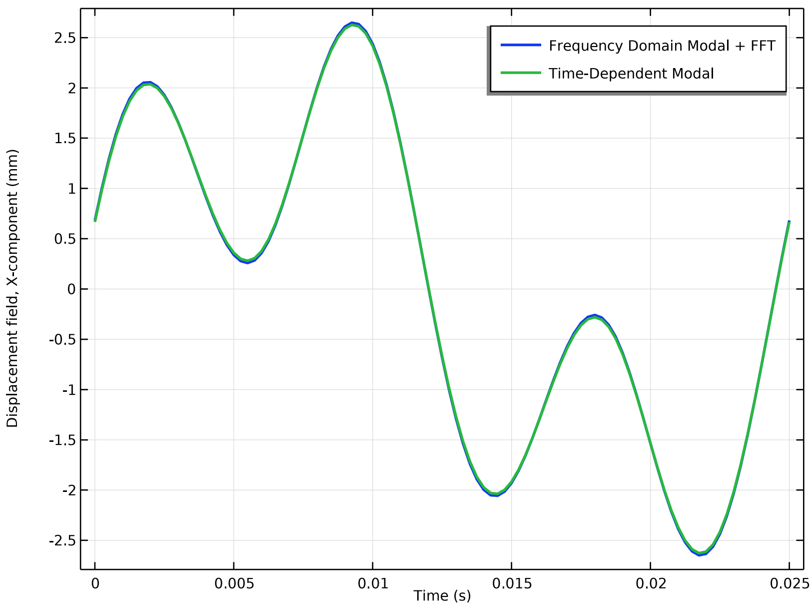

Figure 9: Displacement in the X direction at the tip of the bracket.

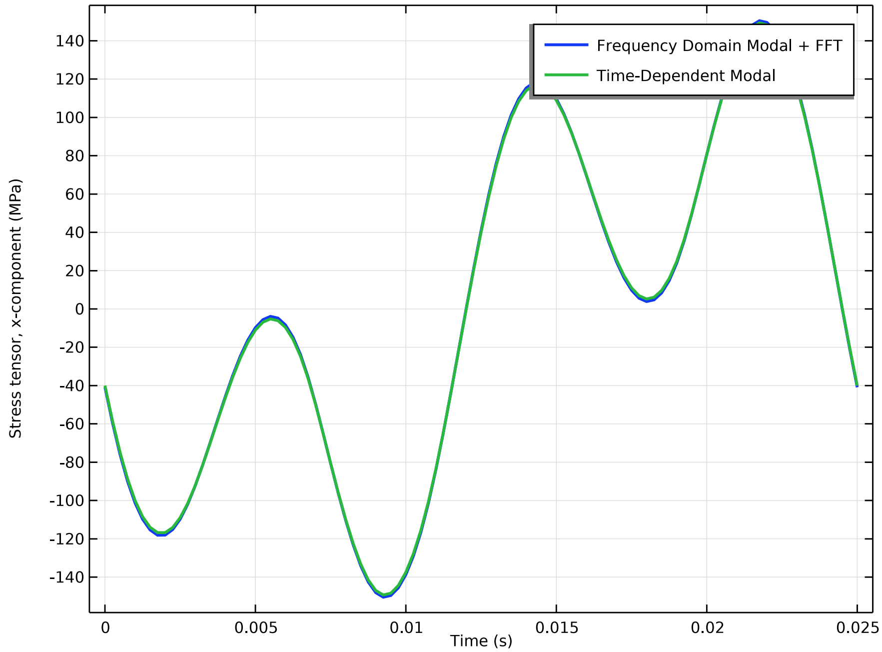

Figure 10: Stress in the critical region in the fillet.

Notes About the COMSOL Implementation

Frequency-domain analysis in COMSOL Multiphysics is performed using a complex-valued representation. The complex-valued Fourier series coefficients can then be directly used as loads. In order to assign the correct coefficient to the corresponding frequency, a `withsol()` operator is used.

As the load is periodic, either a discrete Fourier transform (DFT) or continuous Fourier transform (CFT) can be used. In the CFT, the Fourier series coefficients are scaled in a more natural way, so it is used to compute the Fourier series coefficients. When transforming back into the time domain, a DFT is used. The unscaled version is suitable for performing a direct superposition of the computed frequency response.

In the modal time-dependent procedure, all loads must have the same variation in time, specified in the study step. This means that you should not enter any time-dependent loads (that is, loads with an explicit dependency on the time variable `t`). In this example, the amplitude of the periodic load is entered in the **Boundary Load** node, while the periodic part of the load is entered at the **Modal Solver** node.

Two action buttons are provided in the **Damping Settings** section in order to visualize the damping ratio with respect to frequency. The first button shows a dynamic preview plot of the damping ratio, while the second button generates a plot in the **Results** node.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_general_periodic`

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**.

Describe the periodic load using interpolation and analytical functions.

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
A0	750[N]	750 N	Peak load intensity
M0	0[N]	0 N	Mean load intensity
f0	40[Hz]	40 Hz	Fundamental frequency
T0	1/f0	0.025 s	Base period

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 In the table, enter the following settings:

t	f(t)
0	M0
T0/4	M0+A0
3/4*T0	M0-A0
T0	M0

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

Argument	Unit
t	s

5 In the **Function** table, enter the following settings:

Function	Unit
int1	N

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `int1(x)`.

4 Click to expand the **Periodic Extension** section. Select the **Make periodic** check box.

5 In the **Upper limit** text field, type `T0`.

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

Argument	Unit
x	s

7 In the **Function** text field, type `N`.

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	0	5*T0	0	s

9 Click **Plot**.

10 Right-click **Analytic 1 (loadHistory)** and choose **Duplicate**.

Analytic 2 (loadHistory2)

1 In the **Model Builder** window, under **Global Definitions** click **Analytic 2 (loadHistory2)**.

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

3 Locate the **Definition** section. In the **Expression** text field, type `int1(x)/A0`.

4 Locate the **Units** section. In the **Function** text field, type `1`.

In order to find the Fourier series coefficients of the periodic function, add a zero-dimensional component with a Global ODEs and DAEs mathematical interface.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>0D**.

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 **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

4 Click **Add to Component 2** in the window toolbar.

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

GLOBAL ODES AND DAES (GE)

Global Equations 1 (ODE1)

1 In the **Model Builder** window, under **Component 2 (comp2)>Global ODEs and DAEs (ge)** click **Global Equations 1 (ODE1)**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
P	P-loadHistory(t)	loadHistory(0)	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `force` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Force (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, select **General>Force (N)** in the tree.

12 Click **OK**.

Add **Time Dependent** and **Time to Frequency FFT** study steps to generate Fourier coefficients.

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 **General Studies> Time Dependent**.
4. Right-click and choose **Add Study**.
5. In the **Model Builder** window, click the root node.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1: FOURIER COEFFICIENT GENERATION

1. In the **Settings** window for **Study**, type `Study 1: Fourier Coefficient Generation` in the **Label** text field.
2. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1: Fourier Coefficient Generation** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Output times** text field, type `range(0,T0/1000,T0)`.
4. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

Step 2: Time to Frequency FFT

1. In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Time to Frequency FFT**.
2. In the **Settings** window for **Time to Frequency FFT**, locate the **Study Settings** section.
3. In the **End time** text field, type `T0`.

   By setting a high upper frequency, the accuracy of the Fourier coefficients for the lower frequencies will be improved.
4. In the **Maximum output frequency** text field, type `10000`.
5. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, under **Study 1: Fourier Coefficient Generation>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

5 From the **Steps taken by solver** list, choose **Strict**.

6 Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

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

RESULTS

Fourier Coefficients, Computed vs. Analytical

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** and choose **1D Plot Group**.

3 In the **Settings** window for **1D Plot Group**, type `Fourier Coefficients, Computed vs. Analytical` in the **Label** text field.

4 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **Manual**.

5 In the **Parameter indices (1-251)** text field, type `range(1,1,100)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

7 Locate the **Plot Settings** section.

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

9 Select the **y-axis label** check box. In the associated text field, type `Fourier Coefficients (N)`.

10 Locate the **Axis** section. Select the **y-axis log scale** check box.

Global: Computed

1 Right-click **Fourier Coefficients, Computed vs. Analytical** and choose **Global**.

2 In the **Settings** window for **Global**, type `Global: Computed` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
abs(comp2.P)	N	

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 **Circle**.
6. From the **Width** list, choose **3**.
7. Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
8. In the table, enter the following settings:

Legends
`Computed`

9. Right-click **Global: Computed** and choose **Duplicate**.

Global: Analytical

1. In the **Model Builder** window, under **Results>Fourier Coefficients, Computed vs. Analytical** click **Global: Computed 1**.
2. In the **Settings** window for **Global**, type `Global: Analytical` in the **Label** text field.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`8*A0/(pi^2*(freq/f0)^2)`	`N`	

4. Locate the **Data** section. From the **Dataset** list, choose **Study 1: Fourier Coefficient Generation/Solution 1 (sol1)**.
5. From the **Parameter selection (freq)** list, choose **Manual**.
6. In the **Parameter indices (1-251)** text field, type `range(2,2,100)`.
7. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.
8. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Analytical`

9. In the **Fourier Coefficients, Computed vs. Analytical** toolbar, click **Plot**.

Fourier Coefficients; Real and Imaginary

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Fourier Coefficients; Real and Imaginary` in the **Label** text field.

3. Locate the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Data** section. From the **Parameter selection (freq)** list, choose **Manual**.
5. In the **Parameter indices (1-251)** text field, type `range(2,2,100)`.
6. Locate the **Axis** section. Select the **x-axis log scale** check box.
7. Select the **y-axis log scale** check box.
8. Locate the **Plot Settings** section.
9. Select the **x-axis label** check box. In the associated text field, type `freq (Hz)`.
10. Select the **y-axis label** check box. In the associated text field, type `Fourier Coefficients (N)`.

Global 1

1. Right-click **Fourier Coefficients; Real and Imaginary** 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
`abs(real(comp2.P))`	N	
`abs(imag(comp2.P))`	N	

4. Locate 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 **Circle**.
6. From the **Width** list, choose **3**.
7. Locate the **Legends** section. From the **Legends** list, choose **Manual**.
8. In the table, enter the following settings:

Legends
`Real`
`Imaginary`

9. In the **Fourier Coefficients; Real and Imaginary** toolbar, click **Plot**.

Now, set up the Solid Mechanics interface in order to find the response of the bracket to the periodic load.

DEFINITIONS (COMP1)

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

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**.

   Rayleigh damping is used for this example since it is applicable both in frequency and time domain analysis.

2 In the **Settings** window for **Damping**, locate the **Damping Settings** section.

3 From the **Input parameters** list, choose **Damping ratios**.

4 In the f_1 text field, type `100`.

5 In the ζ_1 text field, type `0.03`.

6 In the f_2 text field, type `300`.

7 In the ζ_2 text field, type `0.03`.

   In order to visualize the damping ratio curve, create the **Damping Ratio** plot through an action button from the **Damping Settings** section.

8 Click **Damping Ratio Preview** in the upper-right corner of the **Damping Settings** section. From the menu, choose **Create Damping Ratio Plot**.

RESULTS

Damping Ratio Plot

1 In the **Model Builder** window, under **Results** click **Damping Ratio Plot**.

2 In the **Damping Ratio Plot** toolbar, click **Plot**.

You can now apply an external harmonic load in terms of the Fourier Series coefficients 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

`withsol('sol1',comp2.P,setval(freq,freq))`	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.

6 Right-click **Boundary Load, Harmonic** and choose **Harmonic Perturbation**.

Add the **Frequency Domain, Modal** study along with the **Time to Frequency FFT** study step.

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>Solid Mechanics>Frequency Domain, Modal**.

4 Right-click and choose **Add Study**.

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

STUDY 2: FREQUENCY DOMAIN MODAL + FFT

In the **Settings** window for **Study**, type `Study 2: Frequency Domain Modal + FFT` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2: Frequency Domain Modal + FFT** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `12`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1>Damping 1**.

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

7 In the tree, select **Component 2 (comp2)>Global ODEs and DAEs (ge)**.

8 Right-click and choose **Disable in Solvers**.

Step 2: Frequency Domain, Modal

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(f0,f0,40*f0)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 2 (comp2)>Global ODEs and DAEs (ge)**.

6 Right-click and choose **Disable in Solvers**.

Step 3: Frequency to Time FFT

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Frequency to Time FFT**.

2 In the **Settings** window for **Frequency to Time FFT**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,T0/100,T0)`.

The purpose of the inverse FFT study step is just to superimpose the results from the different harmonics, so the unscaled discrete FFT is suitable.

4 From the **Scaling** list, choose **Discrete Fourier transform**.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the tree, select **Component 2 (comp2)>Global ODEs and DAEs (ge)**.

7 Right-click and choose **Disable in Solvers**.

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

RESULTS

Stress: Frequency Domain Modal + FFT

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

2 From the **Time (s)** list, choose **0.022**.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

4 In the **Label** text field, type `Stress: Frequency Domain Modal + FFT`.

Volume 1

1 In the **Model Builder** window, expand the **Stress: Frequency Domain Modal + FFT** 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: Frequency Domain Modal + FFT** toolbar, click **Plot**.

Displacement Amplitude [Frequency Response]

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 Amplitude [Frequency Response]` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Frequency Domain Modal + FFT/Solution Store 3 (sol5)**.

5 From the **Parameter selection (freq)** list, choose **Manual**.

6 In the **Parameter indices (1-40)** text field, type `range(1,1,10)`.

Point Graph 1

1 Right-click **Displacement Amplitude [Frequency Response]** and choose **Point Graph**.

2 Select Point 1 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.uAmpX - Displacement amplitude, X-component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.

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

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

7 In the **Displacement Amplitude [Frequency Response]** toolbar, click **Plot**.

For verification, solve the problem also in time domain. To provide the time-domain load, enter its amplitude in a **Boundary Load** node, and its time dependency in the solver settings for the **Time Dependent, Modal** study step.

SOLID MECHANICS (SOLID)

Boundary Load, Time Domain Amplitude

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 In the **Settings** window for **Boundary Load**, type `Boundary Load, Time Domain Amplitude` 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

A0	x
0	y
0	z

Disable the transient boundary load in the second study. While not necessary, since it is a harmonic perturbation load, it makes the modeling easier to follow.

STUDY 2: FREQUENCY DOMAIN MODAL + FFT

Step 2: Frequency Domain, Modal

1 In the **Model Builder** window, under **Study 2: Frequency Domain Modal + FFT** click **Step 2: Frequency Domain, Modal**.
2 In the **Settings** window for **Frequency Domain, Modal**, locate the **Physics and Variables Selection** section.
3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load, Time Domain Amplitude**.
4 Right-click and choose **Disable**.

Step 3: Frequency to Time FFT

1 In the **Model Builder** window, click **Step 3: Frequency to Time FFT**.
2 In the **Settings** window for **Frequency to Time FFT**, locate the **Physics and Variables Selection** section.
3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load, Time Domain Amplitude**.
4 Right-click and choose **Disable**.

Add the **Time-Dependent, Modal** study.

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>Solid Mechanics>Time Dependent, Modal**.

4 Right-click and choose **Add Study**.

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

STUDY 3: TIME-DEPENDENT MODAL [VERIFICATION]

In the **Settings** window for **Study**, type `Study 3: Time-Dependent Modal [Verification]` in the **Label** text field.

Since the eigenfrequencies are already computed in a previous study, it is possible to remove the eigenfrequency study step here.

Step 1: Eigenfrequency

In the **Model Builder** window, under **Study 3: Time-Dependent Modal [Verification]** right-click **Step 1: Eigenfrequency** and choose **Delete**.

Step 1: Time Dependent, Modal

In order for any startup transients to fade away, a significant number of cycles need to be analyzed, but only the results from the last cycle need to be stored. Here, 20 periods will pass before results are stored.

1 In the **Model Builder** window, under **Study 3: Time-Dependent Modal [Verification]** click **Step 1: Time Dependent, Modal**.

2 In the **Settings** window for **Time Dependent, Modal**, locate the **Study Settings** section.

3 In the **Output times** text field, type `-20*T0 range(0,T0/100,T0)`.

For a **Time Dependent, Modal** study step, the time-dependent part of the load needs to be entered in the study settings.

4 In the **Load factor** text field, type `Periodic(t)`.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load, Harmonic**.

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

8 In the tree, select **Component 2 (comp2)>Global ODEs and DAEs (ge)**.

9 Right-click and choose **Disable in Solvers**.

Solution 6 (sol6)

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

Enforce small time steps.

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node, then click **Modal Solver 1**.

3 In the **Settings** window for **Modal Solver**, locate the **General** section.

4 From the **Maximum step constraint** list, choose **Constant**.

5 In the **Maximum step** text field, type `5e-5`.

Since the eigenfrequency study step has been removed from this study, it is necessary to point to the study step that provides the eigenvalue solution.

6 Locate the **Eigenpairs** section. From the **Solution** list, choose **Solution 3 (sol3)**.

7 From the **Use** list, choose **Solution Store 2 (sol4)**.

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

RESULTS

Stress: Time-Dependent Modal

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

2 From the **Time (s)** list, choose **0.022**.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

4 In the **Label** text field, type `Stress: Time-Dependent Modal`.

Volume 1

1 In the **Model Builder** window, expand the **Stress: Time-Dependent Modal** 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: Time-Dependent Modal** toolbar, click **Plot**.

ADD PREDEFINED PLOT

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

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 3: Time-Dependent Modal [Verification]/Solution 6 (sol6)>Solid Mechanics>Applied Loads (solid)>Boundary Loads (solid)**.

4 Click **Add Plot** in the window toolbar.

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

RESULTS

Boundary Loads (solid)

In the **Boundary Loads (solid)** toolbar, click **Plot**.

The maximum displacement occurs at the tip of the bracket, and the maximum stress occurs in the fillet near the bolt holes. Generate 1D plots of displacement and stress in order to visualize the transient response of the bracket. Plot the response from **Study 3** along with **Study 2** in order to validate the results.

Tip Displacement

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

3 From the **Dataset** list, choose **Study 2: Frequency Domain Modal + FFT/Solution 3 (sol3)**.

4 In the **Label** text field, type `Tip Displacement`.

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

Point Graph 1

1 Right-click **Tip Displacement** 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 `u`.

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

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

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

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

9 In the table, enter the following settings:

Legends
`Frequency Domain Modal + FFT`

10 Right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 From the **Dataset** list, choose **Study 3: Time-Dependent Modal [Verification]/ Solution 6 (sol6)**.

4 From the **Time selection** list, choose **Manual**.

5 In the **Time indices (1-102)** text field, type `range(2,102)`.

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

Legends
`Time-Dependent Modal`

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

Tip Displacement

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

Fillet Stress

1 In the **Model Builder** window, under **Results** click **Tip Displacement 1**.

2 In the **Settings** window for **1D Plot Group**, type `Fillet Stress` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Fillet Stress** node, then click **Point Graph 1**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 34 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.sx`.

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 Click to select the **Activate Selection** toggle button.

4 Click **Clear Selection**.

5 Select Point 34 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.sx`.

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

8 In the **Fillet Stress** toolbar, click **Plot**.