Created in COMSOL Multiphysics 6.1

Vibration Analysis of a Deep Beam

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This model studies free and forced vibrations of a deep beam. With the increase of the ratio of section area per beam length, shear deformations and rotational inertia effects can no longer be neglected as it is done in the Euler-Bernoulli theory.

The solution for Eigenfrequency, frequency response and transient analysis are computed using a Timoshenko beam and compared with analytical results.

Model Definition

The model studied in this example consists of a simply supported beam with a square cross section. One end is pinned and has a constrained rotation along the beam axis. At the other end, the displacements in the plane of the beam cross section are constrained.

For the forced vibration cases, a load in the y direction is applied all along the beam.

GEOMETRY

- Beam length, $L = 10$ m
- Beam cross section dimension $l = 2$ m

MATERIAL

- Young's modulus, $E = 200$ GPa
- Poisson's ratio, $\nu = 0.3$
- Mass density, $\rho = 8000\ \mathrm{kg/m^3}$

DAMPING

The Rayleigh damping coefficients are $\alpha = 5.36\ \mathrm{s^{-1}}$, $\beta = 7.46 \cdot 10^{-5}$ m/s. The damping ratio curve is shown in Figure 1.

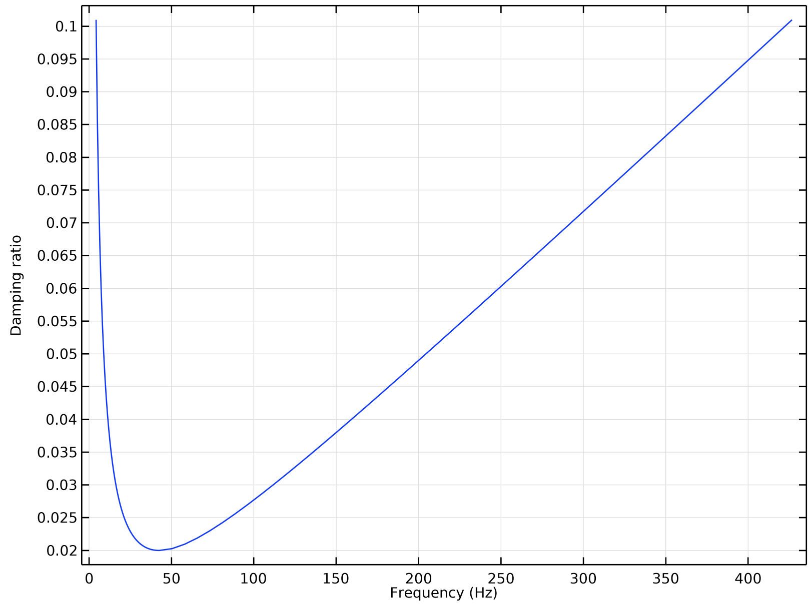

Figure 1: The damping ratio curve.

CONSTRAINTS

At $x = 0$, the displacements are $u = v = w = 0$ and the rotation around the x-axis is fixed at $\theta_x = 0$

At $x = 10$, $v = w = 0$

LOAD CASES

The load with a magnitude $F_0 = 10^6$ N/m is applied and oriented in the local positive y direction. The following forced vibration cases are studied:

- Harmonic response at a frequency of 20 Hz
- Periodic response with the force distribution $F = F_0(\sin(2\pi ft) - \sin(6\pi ft))$, where $f = 20$ Hz
- Transient response using a suddenly applied step load

MESH

To satisfy the benchmark specifications, five edge elements are used to mesh the geometry.

Results and Discussion

Free vibration

In Table 1 the computed results are compared with the analytical results for the free vibration case. The agreement is good. The accuracy decreases with increasing complexity of the mode shape because the possibility for the relatively coarse mesh to describe such a shape is limited.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES.

Mode number	Analytical frequency (Hz)	COMSOL result (Hz)	Error (%)	Type
1, 2	42.65	42.67	4.7e-2	Flexural
3	71.2	71.51	0.4	Torsional
4	125	125.5	0.4	Extensional
5, 6	148.15	150.4	1.5	Flexural
7	213.61	221.6	3.7	Torsional
8, 9	283.47	300.1	5.9	Flexural

Harmonic Forced Vibration

In Table 2 the computed results are compared with the analytical results for the harmonic forced vibration case. The agreement is good.

TABLE 2: COMPARISON BETWEEN ANALYTICAL AND COMPUTED HARMONIC RESPONSES.

	Reference	COMSOL	Error (%)
PEAK DISPLACEMENT (MM)	13.45	13.42	0.2
PEAK STRESS (MPA)	241.9	238.6	1.4
FREQUENCY (HZ)	42.65	42.65	0

In Figure 2 the displacement at the middle of the beam is shown versus the frequency.

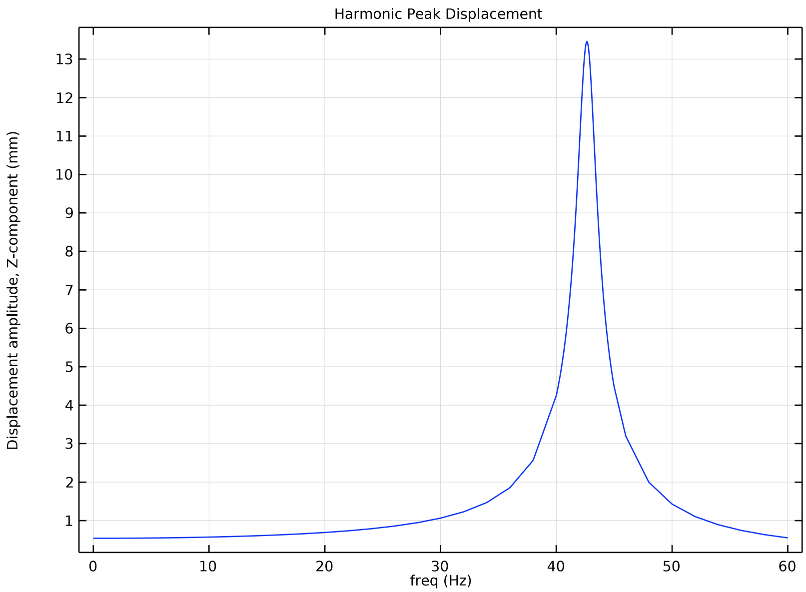

Figure 2: Displacement versus frequency, harmonic response.

In Figure 2 the bending stress at the middle of the beam is shown versus the frequency.

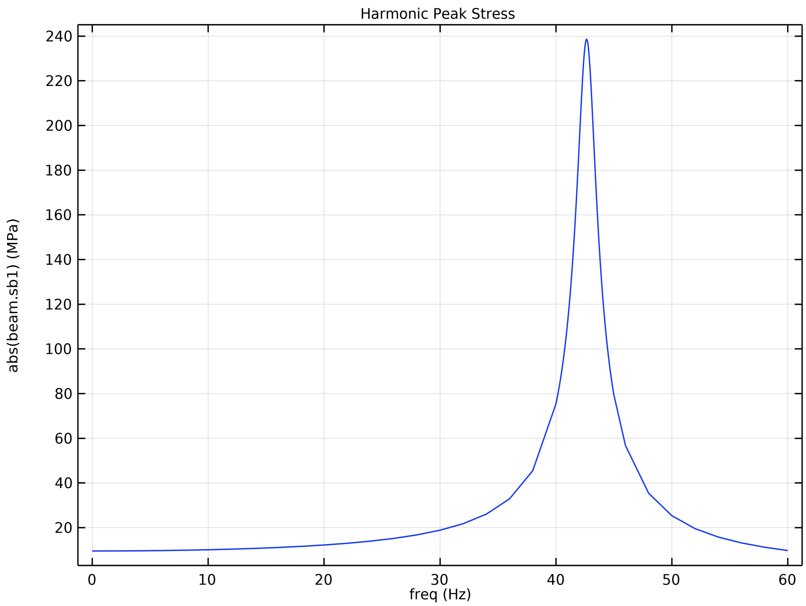

Figure 3: Bending stress versus frequency, harmonic response.

Periodic Forced Vibration

In Table 3 the computed results are compared with the analytical results. The agreement is good. The accuracy decreases with increasing complexity of the mode shape because the possibility for the relatively coarse mesh to describe such a shape is limited.

TABLE 3: COMPARISON BETWEEN ANALYTICAL AND COMPUTED PERIODIC RESPONSES.

	Reference	COMSOL	Error (%)
PEAK DISPLACEMENT (MM)	0.951	0.948	0.3
PEAK STRESS (MPA)	17.1	16.95	0.8

In Figure 4 the displacement at the middle of the beam is shown versus the time.

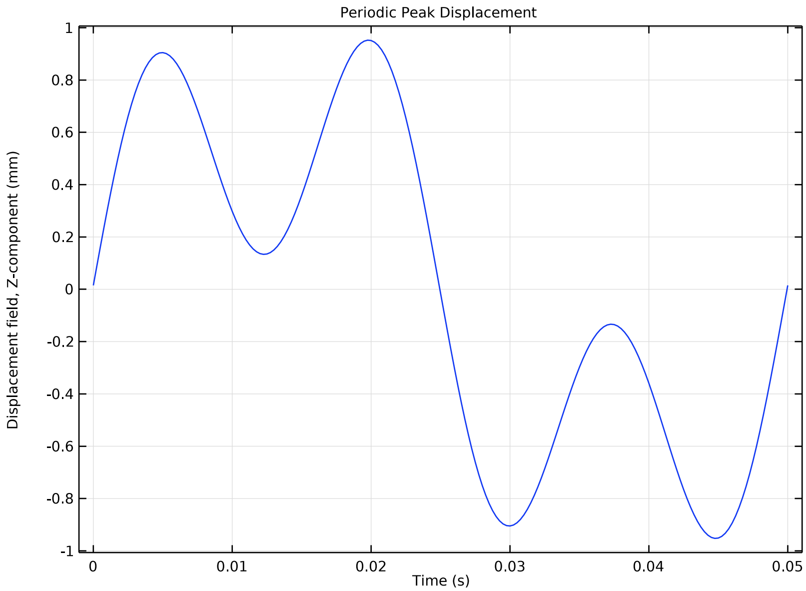

Figure 4: Displacement versus time, periodic response.

In Figure 5 the bending stress at the middle of the beam is shown versus the time.

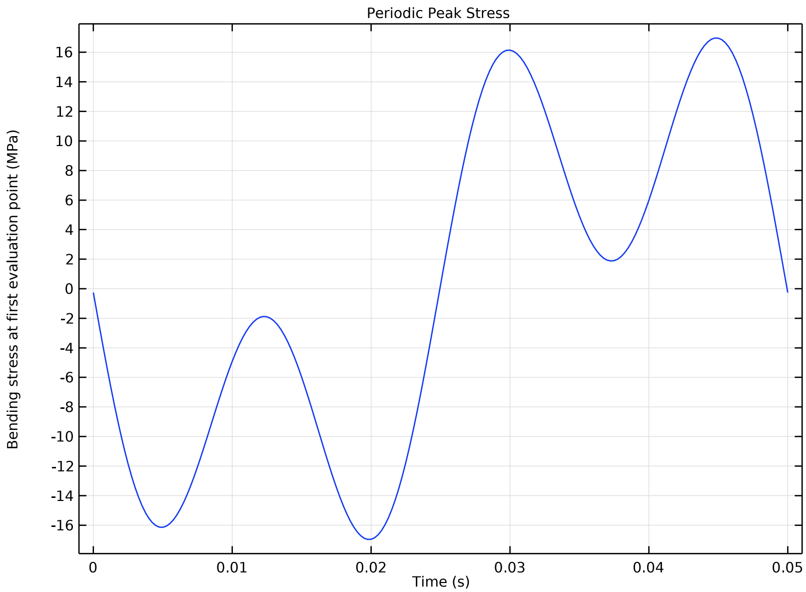

Figure 5: Bending stress versus time, periodic response.

Transient Forced Response

In Table 4 the computed results are compared with the analytical results. The agreement is good. The accuracy decreases with increasing complexity of the mode shape because the possibility for the relatively coarse mesh to describe such a shape is limited.

TABLE 4: COMPARISON BETWEEN ANALYTICAL AND COMPUTED TRANSIENT RESPONSES.

	Reference	COMSOL	Error (%)
PEAK DISPLACEMENT (MM)	1.043	1.037	0.6
PEAK STRESS (MPA)	18.76	18.14	3.3
PEAK TIME (S)	0.0117	0.0117	0
STATIC DISPLACEMENT	0.538	0.534	0.7

In Figure 6 the displacement at the middle of the beam is shown versus the time.

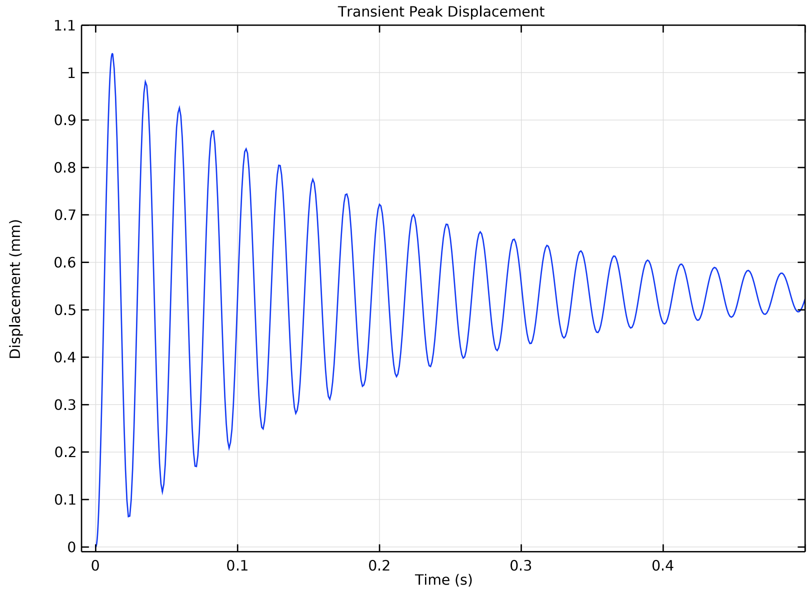

Figure 6: Displacement versus time, transient response.

Notes About the COMSOL Implementation

The **Damping Settings** section provides two action buttons for visualizing 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 under the **Results** node.

Reference

1. J. Maguire, D.J. Dawswell, and L. Gould, "Selected Benchmarks for Forced Vibration", *NAFEMS R0016*, 1989.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/vibrating_deep_beam`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Eigenfrequency**.

6 Click **Done**.

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
L	10[m]	10 m
F0	1e6[N/m]	1E6 N/m

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	0
L	0	0

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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
Young's modulus	E	2e11	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	8000	kg/m³	Basic

BEAM (BEAM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.

2 In the **Settings** window for **Beam**, locate the **Beam Formulation** section.

3 From the list, choose **Timoshenko**.

Cross-Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, locate the **Cross-Section Definition** section.

3 From the list, choose **Common sections**.

4 In the h_y text field, type 2.

5 In the h_z text field, type 2.

Section Orientation 1

1 In the **Model Builder** window, expand the **Cross-Section Data 1** node, then click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the *V* vector as

0	X
0	Y
1	Z

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 1 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 From the **Displacement in x direction** list, choose **Prescribed**.

5 From the **Displacement in y direction** list, choose **Prescribed**.

6 From the **Displacement in z direction** list, choose **Prescribed**.

7 Locate the **Prescribed Rotation** section. From the list, choose **Rotation**.

8 Select the **Free rotation around y direction** check box.

9 Select the **Free rotation around z direction** check box.

Prescribed Displacement/Rotation 2

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 2 only.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 From the **Displacement in y direction** list, choose **Prescribed**.

5 From the **Displacement in z direction** list, choose **Prescribed**.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

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

3 From the **Geometric entity level** list, choose **Entire geometry**.

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.

2 Right-click **Distribution 1** and choose **Build All**.

FREE VIBRATION

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Free Vibration` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Free Vibration** 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 `10`.

4 In the **Search for eigenfrequencies around** text field, type `40`.

To get all eigenfrequencies, including the duplicates at higher frequency, set a tighter tolerance in the solver sequence by following the steps below.

Solution 1 (sol1)

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

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

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

4 In the **Relative tolerance** text field, type `1.0E-15`.

5 Click **Compute to Selected**.

Change the **Radius scale factor** to 0.1 for better visualization.

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Mode Shape (beam)** node, then click **Line 1**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 In the **Radius scale factor** text field, type `0.1`.

BEAM (BEAM)

Edge Load 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 In the **Settings** window for **Edge Load**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **All edges**.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{L}$ vector as

0	X
0	Y
F0	Z

5 Right-click **Edge Load 1** and choose **Harmonic Perturbation**.

Linear Elastic Material 1

In the **Model Builder** window, 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 In the $\alpha_{\mathrm{d}M}$ text field, type `5.36`.

4 In the $\beta_{\mathrm{d}K}$ text field, type `7.46e-5`.

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

5 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**.

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.

HARMONIC FORCED VIBRATION

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Harmonic Forced Vibration` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 2: Frequency Domain, Modal

1 In the **Model Builder** window, under **Harmonic Forced Vibration** 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(0,2,38) range(40,5e-2,45) range(46,2,60)`.

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

RESULTS

Cut Point 3D 1

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

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

3 From the **Dataset** list, choose **Harmonic Forced Vibration/Solution 2 (sol2)**.

4 Locate the **Point Data** section. In the **X** text field, type `L/2`.

5 In the **Y** text field, type `0`.

6 In the **Z** text field, type `0`.

Harmonic Peak Displacement

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

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

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

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

Point Graph 1

1 Right-click **Harmonic Peak Displacement** and choose **Point Graph**.

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

3 In the **Expression** text field, type `beam.uAmpZ`.

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

5 In the **Harmonic Peak Displacement** toolbar, click **Plot**.

Harmonic Peak Stress

1 In the **Model Builder** window, right-click **Harmonic Peak Displacement** and choose **Duplicate**.

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

Point Graph 1

1 In the **Model Builder** window, expand the **Harmonic Peak Stress** node, then 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 `abs(beam.sb1)`.

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

5 In the **Harmonic Peak Stress** toolbar, click **Plot**.

Harmonic Peak Displacement

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Maximum>Line Maximum**.

2 In the **Settings** window for **Line Maximum**, type `Harmonic Peak Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Harmonic Forced Vibration/Solution 2 (sol2)**.

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

5 In the **Parameter values (freq (Hz))** list, select **42.65**.

6 Select Edge 1 only.

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

Expression	Unit	Description
`beam.uAmpZ`	`mm`	`Displacement amplitude, Z-component`

8 Click **Evaluate**.

Harmonic Peak Stress

1 Right-click **Harmonic Peak Displacement** and choose **Duplicate**.

2 In the **Settings** window for **Line Maximum**, type `Harmonic Peak Stress` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit
abs(beam.sb1)	MPa

4 Click ▼ next to **Evaluate**, then choose **New Table**.

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 In the **Function name** text field, type phase.

4 In the table, enter the following settings:

t	f(t)
20	-pi/2
60	pi/2

5 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Nearest neighbor**.

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

Argument	Unit
t	Hz

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

Function	Unit
phase	rad

BEAM (BEAM)

Edge Load 2

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 Select Edge 1 only.

3 In the **Settings** window for **Edge Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_{\mathrm{L}}$ vector as

0	X
0	Y
F0	Z

Phase 1

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

2 In the **Settings** window for **Phase**, locate the **Load Phase** section.

3 Specify the ϕ vector as

0	X
0	Y
`phase(freq)`	Z

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>Frequency Domain**.

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

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

STUDY 3

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `20 60`.

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)>Beam (beam)>Edge Load 1**.

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

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(0.0,1/(200*20),1/20)`.

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 1 (comp1)>Beam (beam)>Edge Load 1**.

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 **FFT Solver 1**.

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

4 From the **Defined by study step** list, choose **User defined**.

5 In the **Model Builder** window, click **Study 3**.

6 In the **Settings** window for **Study**, type `Periodic Forced Vibration` in the **Label** text field.

7 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

RESULTS

Cut Point 3D 2

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

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

3 From the **Dataset** list, choose **Periodic Forced Vibration/Solution 4 (sol4)**.

4 Locate the **Point Data** section. In the **X** text field, type `L/2`.

5 In the **Y** text field, type `0`.

6 In the **Z** text field, type `0`.

Periodic Peak Displacement

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

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

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

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

Point Graph 1

1 Right-click **Periodic Peak Displacement** and choose **Point Graph**.

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

3 In the **Expression** text field, type `w`.

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

5 In the **Periodic Peak Displacement** toolbar, click **Plot**.

Periodic Peak Stress

1 In the **Model Builder** window, right-click **Periodic Peak Displacement** and choose **Duplicate**.

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

Point Graph 1

1 In the **Model Builder** window, expand the **Periodic Peak Stress** node, then 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 `beam.sb1`.

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

5 In the **Periodic Peak Stress** toolbar, click **Plot**.

Periodic Peak Displacement

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Maximum>Line Maximum**.

2 In the **Settings** window for **Line Maximum**, type `Periodic Peak Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Periodic Forced Vibration/Solution 4 (sol4)**.

4 Select Edge 1 only.

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

Expression	Unit
`abs(w)`	`mm`

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

7 Click **Evaluate**.

Periodic Peak Stress

1 Right-click **Periodic Peak Displacement** and choose **Duplicate**.

2 In the **Settings** window for **Line Maximum**, type `Periodic Peak Stress` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit
`abs(beam.sb1)`	`MPa`

4 Click next to **Evaluate**, then choose **New Table**.

BEAM (BEAM)

Edge Load 3

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 In the **Settings** window for **Edge Load**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **All edges**.

4 Locate the **Force** section. Specify the $\mathbf{F}_L$ vector as

0	X
0	Y
F0	Z

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

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

STUDY 4

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,1e-4,1.2e-2) range(1.3e-2,1e-3,1.2)`.

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)>Beam (beam)>Edge Load 1**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Beam (beam)>Edge Load 2**.

7 Click **Disable**.

8 In the **Model Builder** window, click **Study 4**.

9 In the **Settings** window for **Study**, type `Transient Forced Vibration` in the **Label** text field.

10 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 6 (sol6)

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

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node.

3 In the **Model Builder** window, expand the **Transient Forced Vibration> Solver Configurations>Solution 6 (sol6)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e-4`.

6 In the **Model Builder** window, under **Transient Forced Vibration>Solver Configurations> Solution 6 (sol6)** click **Time-Dependent Solver 1**.

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

8 From the **Steps taken by solver** list, choose **Intermediate**.

9 Click **Compute**.

RESULTS

Cut Point 3D 3

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

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

3 From the **Dataset** list, choose **Transient Forced Vibration/Solution 6 (sol6)**.

4 Locate the **Point Data** section. In the **X** text field, type `L/2`.

5 In the **Y** text field, type `0`.

6 In the **Z** text field, type `0`.

Transient Peak Displacement

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 3**.

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

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Time (s)`.

7 Select the **y-axis label** check box. In the associated text field, type `Displacement (mm)`.

8 Locate the **Axis** section. Select the **Manual axis limits** check box.

9 In the **x minimum** text field, type `-1e-2`.

10 In the **x maximum** text field, type `0.5`.

11 In the **y minimum** text field, type `-1e-2`.

12 In the **y maximum** text field, type `1.1`.

Point Graph 1

1 Right-click **Transient Peak Displacement** and choose **Point Graph**.

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

3 In the **Expression** text field, type `w`.

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

5 In the **Transient Peak Displacement** toolbar, click **Plot**.

6 Click **Plot**.

Transient Forced Vibration - Max Displacement

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Maximum>Line Maximum**.

2 In the **Settings** window for **Line Maximum**, type `Transient Forced Vibration - Max Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Transient Forced Vibration/Solution 6 (sol6)**.

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

5 In the **Times (s)** list, select **0.0117**.

6 Select Edge 1 only.

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

Expression	Unit	Description
`w`	`mm`	`Displacement field, Z-component`

8 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

9 Click **Evaluate**.

Transient Forced Vibration - Max Stress

1 Right-click **Transient Forced Vibration - Max Displacement** and choose **Duplicate**.

2 In the **Settings** window for **Line Maximum**, type `Transient Forced Vibration - Max Stress` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit
`abs(beam.sb1)`	`MPa`

4 Click next to **Evaluate**, then choose **New Table**.

Transient Forced Vibration - Static Displacement

1 In the **Model Builder** window, right-click **Transient Forced Vibration - Max Displacement** and choose **Duplicate**.

2 In the **Settings** window for **Line Maximum**, type `Transient Forced Vibration - Static Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Time selection** list, choose **Interpolated**.

4 In the **Times (s)** text field, type `range(1.1,1e-3,1.2)`.

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

Expression	Unit	Description
`w`	`mm`	`Displacement field, Z-component`

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

7 Click next to **Evaluate**, then choose **New Table**.

TABLE

1 Go to the **Table** window.

The vibration analysis is now finished. If you want to save the model and run the study with the configuration set in the previous steps, you need to disable some of the nodes added after you run that particular study. If you do not want to save the model you do not need to follow the steps below.

FREE VIBRATION

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Free Vibration** click **Step 1: Eigenfrequency**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Beam (beam)>Linear Elastic Material 1> Damping 1**.

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

HARMONIC FORCED VIBRATION

Step 2: Frequency Domain, Modal

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

2 In the **Settings** window for **Frequency Domain, Modal**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Beam (beam)>Edge Load 2**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Beam (beam)>Edge Load 3**.

7 Click **Disable**.

PERIODIC FORCED VIBRATION

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Periodic Forced Vibration** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)>Beam (beam)>Edge Load 3**.

4 Click **Disable**.

Step 2: Frequency to Time FFT

1. In the **Model Builder** window, click **Step 2: 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)>Beam (beam)>Edge Load 3**.
4. Click **Disable**.