Created in COMSOL Multiphysics 6.2

Random Vibration Analysis of a Deep Beam

Introduction

This example studies forced random vibrations of a simply-supported deep beam. The beam is loaded by a distributed force with a uniform power spectral density (PSD). The output PSD are computed for the displacement and bending stress response. The computed values are compared with analytical results (NAFEMS test 5R from Ref. 1).

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 beam cross section are constrained.

GEOMETRY

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

With such aspect ratio of the cross section size to the beam length, shear deformations and rotational inertia effects can no longer be neglected as it is done in the Euler-Bernoulli theory. Therefore, the solution is computed using a Timoshenko beam.

MATERIAL

- Young's modulus, $E = 200$ GPa
- Poisson's ratio, $\nu = 0.3$
- Mass density, $\rho = 8000\ \mathrm{kg/m^3}$
- Rayleigh damping coefficient: $\alpha = 5.36\ \mathrm{s^{-1}}$, $\beta = 7.46 \cdot 10^{-5}$ m/s

The values of the damping coefficients are chosen to give a damping ratio of 2% for the first eigenmode.

CONSTRAINTS

At x = 0, $u = v = w = 0$; $thx = 0$

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

LOAD

For a linear system, the response in the frequency domain for a single variable V to the excitation F can be written

$$V(f) = H_{VF}(f)F$$

where f is the frequency, and H is the complex valued transfer function. It can then be shown that the corresponding spectral densities have the relation

$$S_V(f) = \left|H_{VF}(f)\right|^2 S_F(f) = H^*{}_{VF}(f) H_{VF}(f) S_F(f)$$

where the asterisk denotes a complex conjugate. This type of relation is true not only for the degrees of freedom, but for any quantity that is linearly related to the input. This includes components of stress and (engineering) strain, but not nonlinear quantities such as equivalent or principal stresses.

In this example, a load of $F = 10^6$ N/m in the y direction is applied uniformly along the beam for the forced harmonic vibration study. For the random vibration analysis, the load is assumed to have a uniformly distributed PSD of 10^{12} $(\mathrm{N/m})^2/\mathrm{Hz}$. Thus, one should expect that results have the property

$$S_V(f) = |V|^2(f)$$

That is, the PSD response is simply the square of the standard harmonic response.

Results and Discussion

The plot below shows the computed PSD of the beam vertical displacement at the mid point. Note that is also matches the squared nonrandom frequency response at the same point.

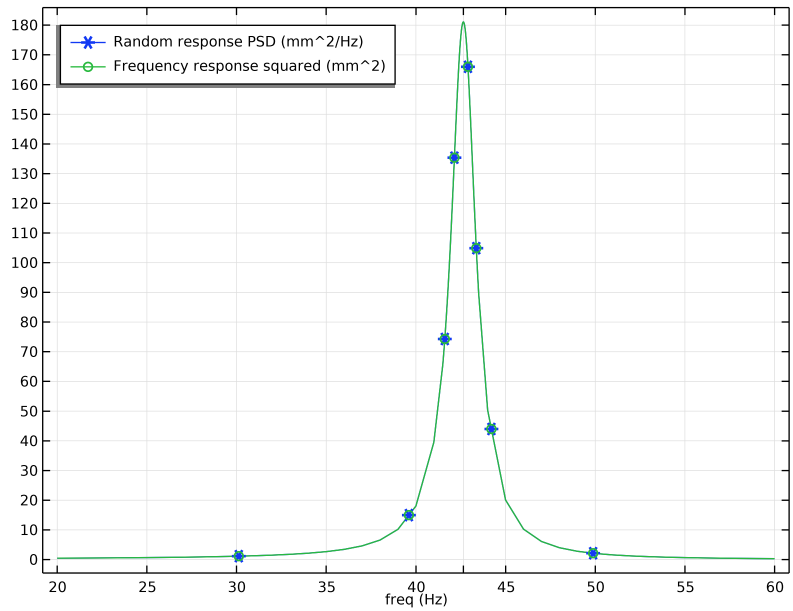

Figure 1: The PSD of the displacement response at the midpoint of the beam.

In Table 1, the computed results are compared with the analytical results from Ref. 1. The agreement is good.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED RANDOM RESPONSES.

	Peak displacement PSD mm^2/Hz	Peak stress PSD $(N/mm^2)^2/Hz$	Frequency Hz
Reference	180.90	58516	42.65
COMSOL	181.04	56922	42.66

In this benchmark, a mesh consisting of only five elements is prescribed. The stress is measured at the midpoint of the beam, that is at the midpoint of the central beam element. Since the finite element approximation in the beam elements give a linear variation of the bending moment within each element, the bending moment (and thus the stress) in the central element is constant for symmetry reasons. The true midpoint value will thus be underestimated. If six elements are used instead, there will be a node at the midpoint. The stress PSD value in that node turns out to be 60,652 $(\mathrm{N/mm^2})^2/\mathrm{Hz}$.

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/random_vibration_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 **Preset Studies for Selected Physics Interfaces>Random Vibration (PSD)**.

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**	**Description**
F	1e6[N/m]	1E6 N/m	Edge load
PSD	F^2/1[Hz]	1E12 kg²/s³	Random edge load, power spectral density

GEOMETRY 1

Line Segment 1 (ls1)

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

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **x** text field, type 10.

6 Click **Build All Objects**.

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 **Section type** list, choose **Rectangle**.
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, 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

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** 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 α_{dM} text field, type `5.36`.
4 In the β_{dK} text field, type `7.46e-5`.

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 **More Generators** and choose **Edge**.

2 Select Edge 1 only.

Distribution 1

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

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

DEFINITIONS

Set up an operator to evaluate variables at the beam midpoint.

General Extrusion 1 (genext1)

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

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

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

4 Select Edge 1 only.

5 Locate the **Destination Map** section. In the **x-expression** text field, type `5`.

6 In the **y-expression** text field, type `0`.

7 In the **z-expression** text field, type `0`.

8 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
V	genext1(v)	m	Displacement, y-component
Sb	genext1(beam.sb1)	N/m^2	Bending stress

GLOBAL DEFINITIONS

Set up a control parameter to be used as the edge load.

Global Reduced-Model Inputs 1

1 In the **Model Builder** window, expand the **Global Definitions>Reduced-Order Modeling** node, then click **Global Reduced-Model Inputs 1**.

2 In the **Settings** window for **Global Reduced-Model Inputs**, locate the **Reduced-Model Inputs** section.

3 In the table, enter the following settings:

Control name	Expression
Fy	F

BEAM (BEAM)

Edge Load 1

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 $\mathbf{F}_L$ vector as

0	x
Fy	y
0	z

STUDY 1

Step 1: Eigenfrequency

Set the search position close to the target value of the first natural frequency.

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

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

3 In the **Search for eigenfrequencies around shift** text field, type 40.

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)>Beam (beam)>Linear Elastic Material 1>Damping 1**.

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

The eigenmode computation should be always performed for the undamped system. The damping will be used however in the consequent modal frequency response and random response analysis.

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

STUDY 2

Step 1: Model Reduction

The computation of the solution for Study 2 will find the eigenfrequencies and build a modal reduced-order model (ROM) based on the computed eigenmodes.

1 Click **Compute**.

You can see all computed eigenfrequencies in the automatically generated evaluation group.

GLOBAL DEFINITIONS

Next, set up the input PSD for the random edge load.

Random Vibration 1 (rvib1)

1 In the **Model Builder** window, under **Global Definitions>Reduced-Order Modeling** click **Random Vibration 1 (rvib1)**.

2 In the **Settings** window for **Random Vibration**, locate the **Power Spectrum** section.

3 In the table, enter the following settings:

Control name	Power spectral density
Fy	PSD

Update the study to make the input change available for the solution.

STUDY 2

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

The random response computations can be performed as postprocessing steps using the updated solution.

RESULTS

Add a plot of the PSD for the displacement responses at the midpoint. For verification, you can also plot the nonrandom frequency response result computed using ROM.

Global Evaluation Sweep 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Other> Global Evaluation Sweep**.

Use the frequency range to resolve well the values close to the target first natural frequency.

2 In the **Settings** window for **Global Evaluation Sweep**, locate the **Parameters** section.

3 In the table, enter the following settings:

Parameter name	Parameter value list
freq	range(20,1,41) range(41.5,0.01,43.5) range(44,1,60) [Hz]

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

Expression	Unit	Description
rvib1.psd(V)	mm^2/Hz	Random response PSD
abs(rom1.eval(V))^2	mm^2	Frequency response squared

5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

6 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, expand the **Results>Tables** node, then click **Results> 1D Plot Group 2>Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 From the **Positioning** list, choose **Interpolated**.

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

1D Plot Group 2

Indicate the target peak frequency.

1 In the **Model Builder** window, click **1D Plot Group 2**.

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

3 In the **Extra x** text field, type `42.65`.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 In the **1D Plot Group 2** toolbar, click **Plot**.

The actually computed peak frequency is close to 42.66 (Hz).

Finally, calculate the maximum PSD values in the computed frequency range for both the displacement and bending stress responses.

Global Evaluation Sweep 1

In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation Sweep 1** and choose **Duplicate**.

Global Evaluation Sweep 2

1 In the **Model Builder** window, click **Global Evaluation Sweep 2**.

2 In the **Settings** window for **Global Evaluation Sweep**, locate the **Parameters** section.

3 In the table, enter the following settings:

Parameter name	**Parameter value list**
`freq`	`42.66 [Hz]`

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

Expression	**Unit**	**Description**
`rvib1.psd(V)`	`mm^2/Hz`	`Displacement, y-component, maximum PSD`
`rvib1.psd(Sb)`	`(N/mm^2)^2/Hz`	`Bending stress, maximum PSD`

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

Compare the results with the target values.