In-Plane Framework with Discrete Mass and Mass Moment of Inertia

Introduction

In the following example you build and solve a 2D beam model using the 2D Structural Mechanics Beam interface. This example describes the eigenfrequency analysis of a simple geometry. A point mass and point mass moment of inertia are used. The two first eigenfrequencies are compared with the values given by an analytical expression.

In addition, it is shown how to evaluate modal participation factors and modal masses.

Model Definition

The geometry consists of a frame with one horizontal and one vertical member. The cross section of both members has an area, A, and an area moment of inertia, I. The length of each member is L and Young's modulus is E. A point mass m is added at the middle of the horizontal member and a point mass moment of inertia J at the corner (see Figure 1 below).

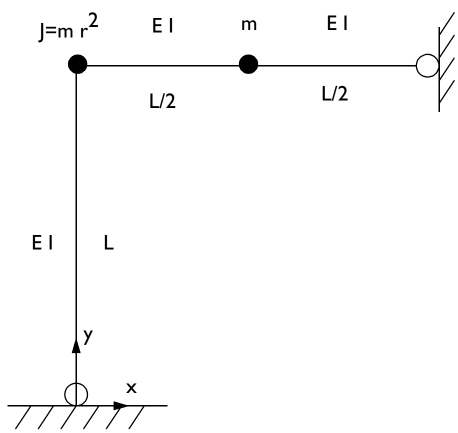


Figure 1: Definition of the problem.

GEOMETRY

- Framework member lengths, $L = 1$ m.
- The framework members has a square cross section with a side length of 0.03 m giving an area of $A = 9{\cdot}10^{-4}$ m^2 and an area moment of inertia of $I = 0.03^4/12$ m^4.

MATERIAL

Young's modulus $E = 200$ GPa.

MASS

- Point mass $m = 1000$ kg.
- Point mass moment of inertia $J = mr^2$ where r is chosen as $L/4$. This gives the value 62.5 kgm^2.

CONSTRAINTS

The beam is pinned at $x = 0, y = 0$ and $x = 1, y = 1$, meaning that the displacements are constrained whereas the rotational degrees of freedom are free.

Results and Discussion

The analytical values for the two first eigenfrequencies f_{e1} and f_{e2} are given by:

$$\omega_{e1}^2 = \frac{48EI}{mL^3}$$

$$\omega_{e2}^2 = \frac{48 \cdot 32EI}{7mL^3}$$

and

$$f_{e1} = \frac{\omega_{e1}}{2\pi}$$

$$f_{e2} = \frac{\omega_{e2}}{2\pi}$$

where ω is the angular frequency.

The following table shows a comparison between the eigenfrequencies calculated with COMSOL Multiphysics and the analytical values.

EIGENMODE	COMSOL MULTIPHYSICS	ANALYTICAL
1	4.05 Hz	4.05 Hz
2	8.65 Hz	8.66 Hz

The following two plots visualize the two eigenmodes.

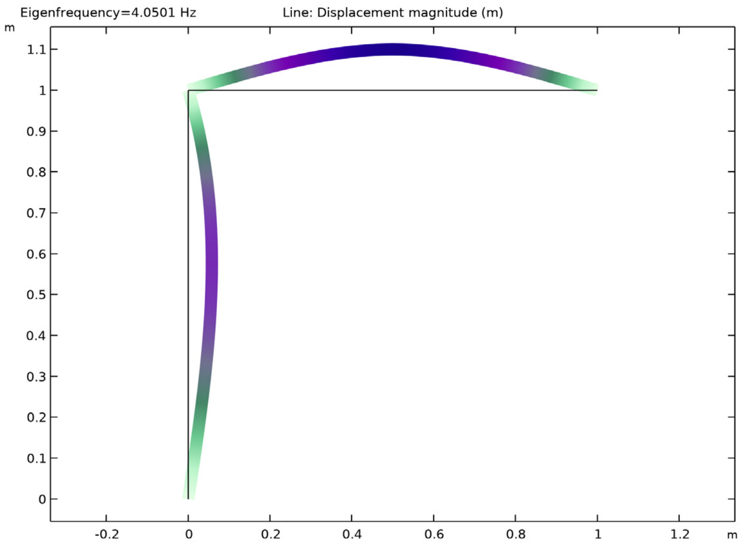


Figure 2: The first eigenmode.

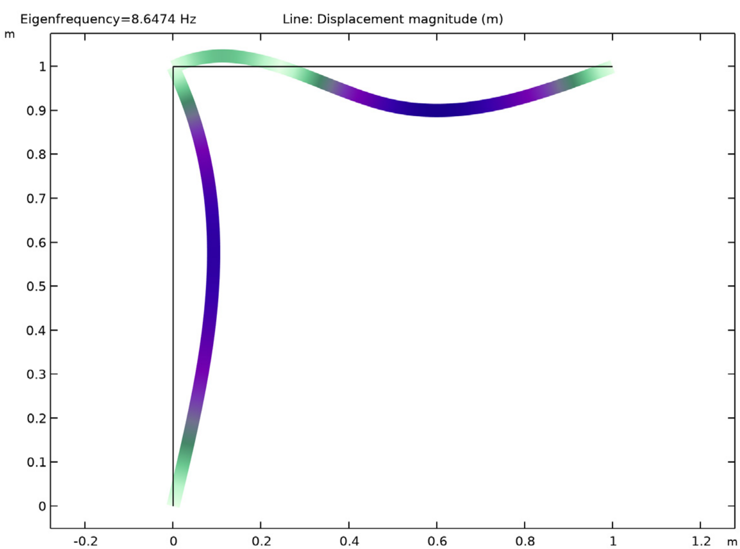
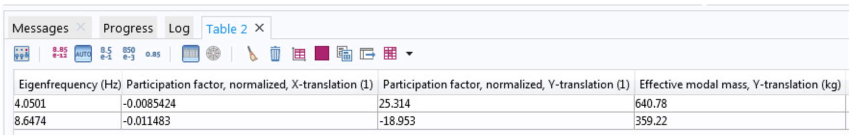


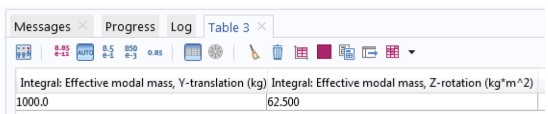
Figure 3: The second eigenmode.

Because the beams have no density in this example, the total mass is the 1000 kg supplied by the point mass. The mass moment of inertia is also a point contribution, and has the value 62.5 kgm^2. The mass represented by the computed eigenmodes can be evaluated using the modal participation factors, see Figure 4 and Figure 5. In this case, it can be seen that in the y direction, the correspondence is perfect, while almost none of the mass in the x direction is represented. The axial deformation mode for the horizontal member has a higher frequency, and was not computed. Similarly, all rotational inertia is captured by the first two modes.



Eigenfrequency (Hz)	Participation factor, normalized, X-translation (1)	Participation factor, normalized, Y-translation (1)	Effective modal mass, Y-translation (kg)
4.0501	-0.0085424	25.314	640.78
8.6474	-0.011483	-18.953	359.22

Figure 4: Participation factors for each eigenfrequency.



Integral: Effective modal mass, Y-translation (kg)	Integral: Effective modal mass, Z-rotation (kg*m^2)
1000.0	62.500

Figure 5: Summed modal masses.

Notes About the COMSOL Implementation

The variables for evaluation of participation factors are created in the **Participation Factors** node under **Definitions**. This node is created automatically when an **Eigenfrequency** study is added.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/inplane_framework_freq`

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 **2D**.

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 Click **Load from File**.

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

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

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

3 From the **Type** list, choose **Open curve**.

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

x (m)	y (m)
0	0
0	L
L/2	L
L	L

5 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	Emod	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0	1	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic

BEAM (BEAM)

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 `a`.

5 In the h_z text field, type `a`.

Pinned 1

1 In the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1 and 4 only.

Point Mass 1

1 In the **Physics** toolbar, click **Points** and choose **Point Mass**.

2 Select Point 3 only.

3 In the **Settings** window for **Point Mass**, locate the **Point Mass** section.

4 In the m text field, type `m`.

Point Mass 2

1 In the **Physics** toolbar, click **Points** and choose **Point Mass**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Mass**, locate the **Point Mass** section.

4 In the J_z text field, type `J`.

STUDY 1

Step 1: Eigenfrequency

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 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `2`.

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

RESULTS

Line 1

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

2 In the **Mode Shape (beam)** toolbar, click **Plot**.

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

Mode Shape (beam)

1 In the **Model Builder** window, click **Mode Shape (beam)**.

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

3 From the **Eigenfrequency (Hz)** list, choose **8.6474**.

4 In the **Mode Shape (beam)** toolbar, click **Plot**.

Compare the computed eigenfrequencies to the analytical values.

Eigenfrequency Comparison

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Eigenfrequency Comparison` in the **Label** text field.

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

Expression	Unit	Description
`f1`	`1/s`	`Eigenfrequency 1, analytical`
`f2`	`1/s`	`Eigenfrequency 2, analytical`

4 Click **Evaluate**.

Participation Factors (Study 1)

1 In the **Model Builder** window, under **Results** click **Participation Factors (Study 1)**.

2 In the **Participation Factors (Study 1)** toolbar, click **Evaluate**.

Examine the modal participation factors.

Finally, compute the total effective mass accounted for in the computed eigenmodes.

Summed Modal Masses

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Summed Modal Masses` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Participation Factors 1>Effective modal mass>mpf1.mEffLY - Effective modal mass, Y-translation - kg**.

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

Expression	Unit	Description
mpf1.mEffLY	kg	Effective modal mass, Y-translation
mpf1.mEffRZ	kg*m^2	Effective modal mass, Z-rotation

5 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Integral**.

6 From the **Method** list, choose **Summation**.

7 Click **Evaluate**.