Created in COMSOL Multiphysics 6.0

Vibrating String

Introduction

In the following example you compute the natural frequencies of a pretensioned string using the 2D Truss interface. This is an example of "stress stiffening". In fact the transverse stiffness of truss elements is directly proportional to the tensile force.

Strings made of piano wire have an extremely high yield limit, thus enabling a wide range of pretension forces.

The results are compared with the analytical solution.

Model Definition

The finite element idealization consists of a single line. The diameter of the wire is irrelevant for the solution of this particular problem, but it must still be given.

GEOMETRY

- String length, $L = 0.5$ m
- Cross section diameter 1.0 mm; $A = 0.785\ \text{mm}^2$

MATERIAL

- Young's modulus, $E = 210$ GPa
- Poisson's ratio, $\nu = 0.31$
- Mass density, $\rho = 7850\ \text{kg/m}^3$

CONSTRAINTS

Both ends of the wire are fixed.

LOAD

The wire is pretensioned to $\sigma_{ni} = 1520$ MPa.

Results and Discussion

The analytical solution for the natural frequencies of the vibrating string (Ref. 1) is

$$f_k = \frac{k}{2L}\sqrt{\frac{\sigma_{ni}}{\rho}} \tag{1}$$

The pretensioning stress σ_{ni} in this example is tuned so that the first natural frequency is Concert A; 440 Hz.

In Table 1 the computed results are compared with the results from Equation 1. The agreement is very 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. The mode shapes for the first three modes are shown in Figure 1 through Figure 3.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES.

Mode number	Analytical frequency (Hz)	COMSOL result (Hz)
1	440.0	440.1
2	880.0	880.6
3	1320	1322
4	1760	1765
5	2200	2209

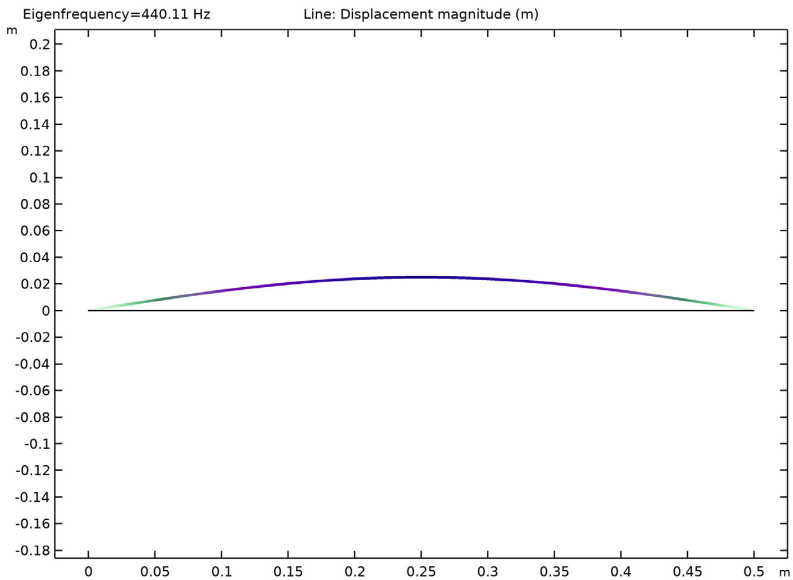

Figure 1: First eigenmode.

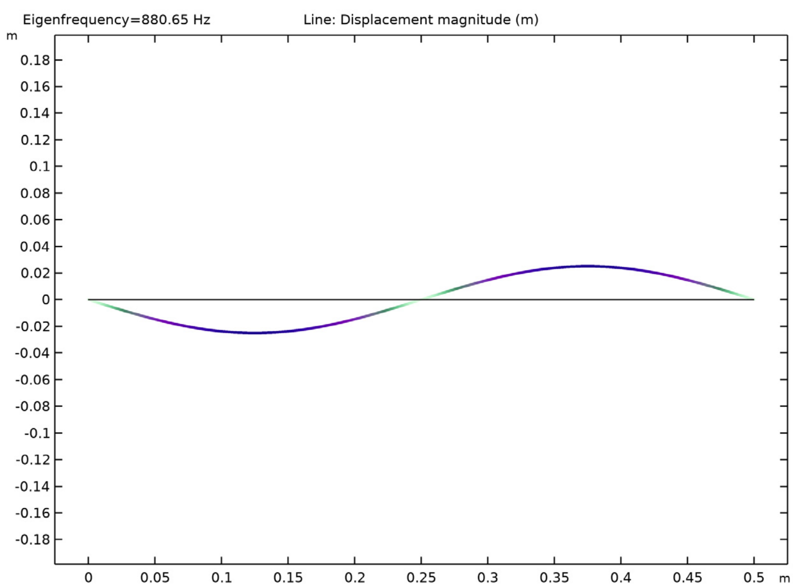

Figure 2: Second eigenmode.

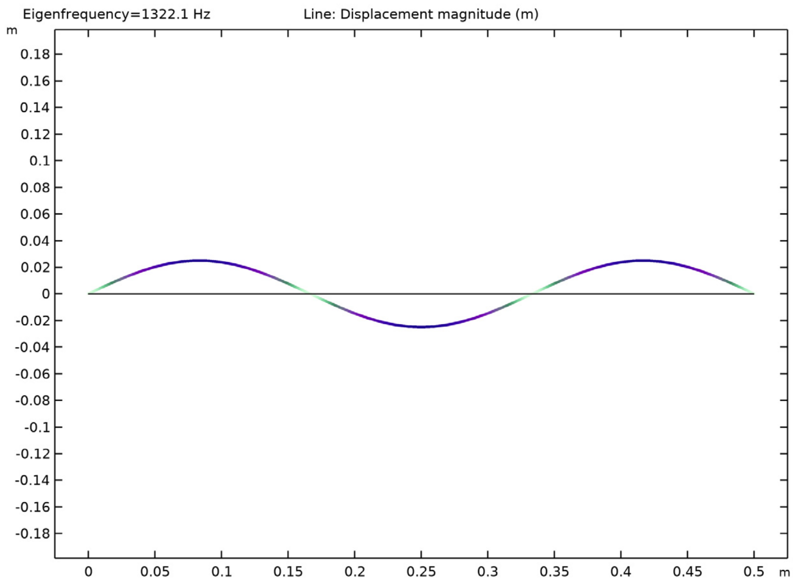

Figure 3: Third eigenmode.

Notes About the COMSOL Implementation

In this example, the stresses are known in advance, so it is possible to use an initial stress condition. This is shown in the first study.

In a general case, the prestress is given by some external loading. The structural response of to this loading needs to be calculated and incorporated into the structure before the eigenfrequency can be computed. Such a study therefore consists of two steps: One stationary step for computing the prestressed state, and one step for the eigenfrequency. The special study type Prestressed Analysis, Eigenfrequency can be used to set up such a sequence. This is shown in the second study in this example.

Since an unstressed string has no stiffness in the transverse direction, it is generally difficult to get an analysis to converge without taking special measures. One such method is shown in the second study: A spring foundation is added during initial loading, and is then removed.

You must switch on geometrical nonlinearity in the study in order to capture effects of prestress. This is done automatically when a study of the type Prestressed Analysis, Eigenfrequency is used.

Reference

1. R. Knobel, *An Introduction to the Mathematical Theory of Waves*, The American Mathematical Society, 2000.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/vibrating_string`

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>Truss (truss)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

x (m)	y (m)
0	0
0.5	0

4 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	210e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.31	1	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m³	Basic

TRUSS (TRUSS)

Cross-Section Data 1

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

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

3 In the A text field, type `pi/4*0.001^2`.

Pinned 1

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

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

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

The straight edge constraint must be removed because the vibration gives the string a curved shape.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the σ_{n0} text field, type `1520e6`.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Edge**.
2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.01`.

   This setting results in a mesh with 50 elements, which COMSOL Multiphysics generates when you solve the model.

The stiffness caused by the prestress is a nonlinear effect, so geometric nonlinearity must be switched on.

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 Find the **Values of linearization point** subsection. Select the **Include geometric nonlinearity** check box.
4 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (truss)

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

   The default plot shows the displacement for the first eigenmode.

Line 1

1 In the **Model Builder** window, expand the **Mode Shape (truss)** 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 `2`.

Mode Shape (truss)

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

2 In the **Model Builder** window, click **Mode Shape (truss)**.

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

4 From the **Eigenfrequency (Hz)** list, choose **880.65**.

   This corresponds to the second eigenmode.

5 In the **Mode Shape (truss)** toolbar, click **Plot**.

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

7 From the **Eigenfrequency (Hz)** list, choose **1322.1**.

   This is the third eigenmode.

8 In the **Mode Shape (truss)** toolbar, click **Plot**.

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

Now, prepare a second study where the prestress is instead computed from an external load. The pinned condition at the right end must then be replaced by a force.

TRUSS (TRUSS)

Pinned 2

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

2 Select Point 1 only.

Prescribed Displacement 1

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

2 Select Point 2 only.

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

4 Select the **Prescribed in y direction** check box.

Point Load 1

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

2 Select Point 2 only.

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

4 Specify the $\mathbf{F}_P$ vector as

`1520[MPa]*truss.area`	x
0	y

Add a spring with an arbitrary, small stiffness in order to suppress the out-of-plane singularity of the unstressed wire.

Spring Foundation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the list, choose **Diagonal**.

5 In the $\mathbf{k}_L$ table, enter the following settings:

0	0
0	10

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency, Prestressed**.

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

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

STUDY 2

Step 1: Stationary

Switch off the initial stress and double-sided pinned condition, which should not be part of the second study. In the eigenfrequency step, the stabilizing spring support must also be removed.

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

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

3 In the tree, select **Component 1 (Comp1)>Truss (Truss)>Linear Elastic Material 1>Initial Stress and Strain 1** and **Component 1 (Comp1)>Truss (Truss)>Pinned 1**.

4 Click **Disable**.

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: 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)>Truss (Truss)>Linear Elastic Material 1>Initial Stress and Strain 1**, **Component 1 (Comp1)>Truss (Truss)>Pinned 1**, and **Component 1 (Comp1)>Truss (Truss)>Spring Foundation 1**.

5 Click **Disable**.

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

RESULTS

Mode Shape (truss) 1

The eigenfrequencies computed using this more general approach are close to those computed in the previous step.

Line 1

1 In the **Model Builder** window, expand the **Mode Shape (truss) 1** 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 2.

To make **Study 1** behave as when it was first created, the features added for **Study 2** must be disabled.

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 **Physics and Variables Selection** section.

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

4 In the tree, select **Component 1 (Comp1)>Truss (Truss)>Pinned 2**, **Component 1 (Comp1)>Truss (Truss)>Prescribed Displacement 1**, **Component 1 (Comp1)>Truss (Truss)>Point Load 1**, and **Component 1 (Comp1)>Truss (Truss)>Spring Foundation 1**.

5 Click **Disable**.