Created in COMSOL Multiphysics 6.2

Elasto-Acoustic Effect in Rail Steel

Introduction

The elasto-acoustic effect is the change in the speed of elastic waves that propagate in a structure undergoing static elastic deformations. The effect is used in many ultrasonic techniques for nondestructive testing of prestressed states within structures.

This example model studies the elasto-acoustic effect in steels typically used in railroad rails. The analysis is based on the Murnaghan material model, which represents a hyperelastic isotropic material, and is based on an expansion of the elastic potential in terms of displacement gradients keeping the terms up to the third order. This material model can be used to study various nonlinear effects in materials and structures, of which the elasto-acoustic effect is an example.

Model Definition

The geometry represents a head of a railroad rail. It is a beam with a length of $L_0 = 0.607$ m and a cross section of 0.0624 m by 0.0262 m. The rail is made of steel with the following properties (taken from Ref. 1):

- Density: $\rho = 7800$ kg/m^3
- Lamé elastic moduli: $\lambda = 11.58 \cdot 10^{10}$ Pa and $\mu = 7.99 \cdot 10^{10}$ Pa.
- Murnaghan third-order elastic constants: $l = -24.8 \cdot 10^{10}$ Pa, $m = -62.3 \cdot 10^{10}$ Pa, and $n = -71.4 \cdot 10^{10}$ Pa.

To create a prestressed state, the beam is stretched to a length $L = (1 + \varepsilon)L_0$, where $\varepsilon = 5 \cdot 10^{-4}$. The model computes the eigenfrequencies of the beam for the free and prestressed states and the relative change in the speed of propagating elastic waves. Since only the axial waves are of interest here, symmetry conditions are used along two planes (xy and xz), so that bending is suppressed. The model is shown in Figure 1.

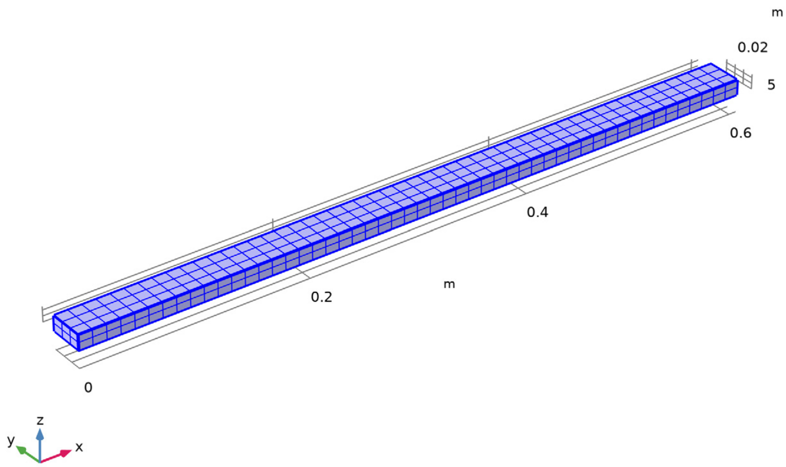

Figure 1: Geometry and mesh.

Results and Discussion

Instead of directly computing the wave speed, which can be a difficult task, the eigenfrequencies of the structure are used in order to implicitly obtain the wave speed. The relative change in the axial wave speed per unit strain can be estimated by the following formula:

$$\frac{c-c_0}{c_0\varepsilon} = \frac{f-f_0}{\varepsilon f_0} + \frac{f-f_0}{f_0} + 1 \qquad (1)$$

Here the letter c denotes axial wave speed, and f the natural frequency for an axial mode. The subscript 0 refers to the unstrained state. This equation takes the elongation of the rod into account through the last two terms.

For a stress free sample, the computed eigenfrequency is $f_0 = 4242.34$ Hz, which is shown in Figure 2. For prestressed sample, the computed eigenfrequency is $f = 4234.66$ Hz shown in Figure 3.

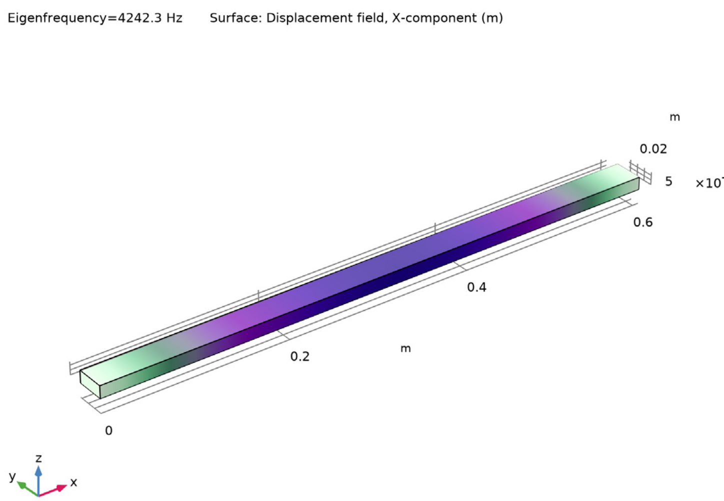

Figure 2: Eigenfrequency and normalized eigenfunction for a stress-free rail.

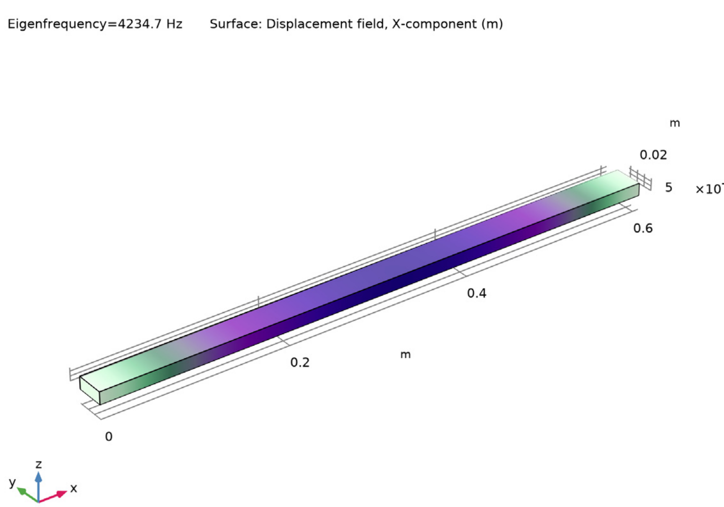

Figure 3: Eigenfrequency and normalized eigenfunction for a prestressed rail.

The deformation in the prestressed state is shown in Figure 4 below.

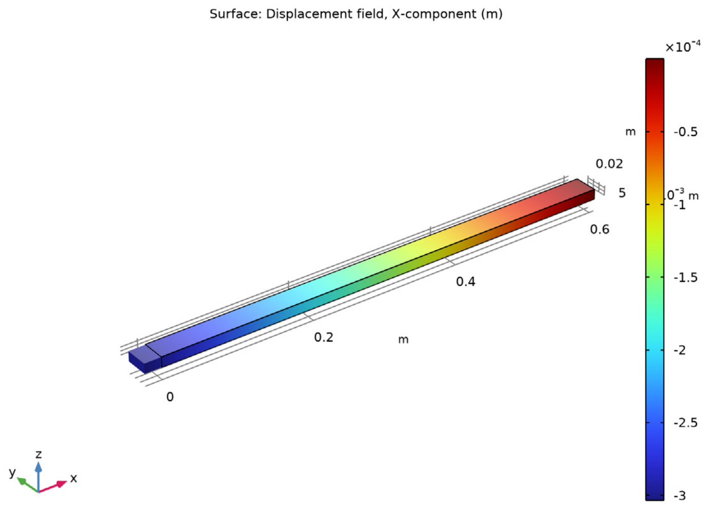

Figure 4: The static displacement field in the prestressed beam.

When the piece of rail is stretched as shown in Figure 4, the length of the rail changes, but most importantly, we get a prestressed state, which changes the wave speed in the material. Therefore, the eigenfrequency changes to f = 4234.66 Hz. The resulting value of the relative change in wave speed using Equation 1 is –2.62, which is in a good agreement with the experimental value of –2.52 reported in Ref. 1 (Table III, specimen 1). It is noted that if a linearly elastic material model is used, the predicted change in wave speed instead shows an increase.

Notes About the COMSOL Implementation

The eigenfrequency computation is performed using boundary conditions in the axial direction in both ends. One of them is implemented using a symmetry condition, which is just a simple way of prescribing the displacement in the normal direction to zero. In the other end, the displacement is prescribed to the value that gives the intended axial strain. In the first analysis, where the unstrained eigenfrequency is studied, this boundary condition acts as if the prescribed value is zero, since no static analysis precedes the eigenfrequency calculation.

Reference

1. D.M. Egle and D.E. Bray, “Measurement of Acoustoelastic and Third-order Elastic Constants for Rail Steel,” *J. Acoust. Soc. Am.*, vol. 60, no. 3, p. 741, 1976.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/rail_steel`

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>Solid Mechanics (solid)**.

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	Description
eps0	5e-4	5E-4	Prescribed strain
L0	0.607[m]	0.607 m	Length of the beam

GEOMETRY 1

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `L0`.

4 In the **Depth** text field, type `0.0624*0.5`.

5 In the **Height** text field, type `0.0262*0.5`.

6 Click **Build All Objects**.

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

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**.

SOLID MECHANICS (SOLID)

Hyperelastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, locate the **Domain Selection** section.

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

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Murnaghan**.

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
Lamé parameter λ	lambLame	11.58e10	N/m²	Lamé parameters
Lamé parameter μ	muLame	7.99e10	N/m²	Lamé parameters
Murnaghan third-order elastic moduli	l	-24.8e10	N/m²	Murnaghan

Property	Variable	Value	Unit	Property group
Murnaghan third-order elastic moduli	m	-62.3e10	N/m²	Murnaghan
Murnaghan third-order elastic moduli	n	-71.4e10	N/m²	Murnaghan
Density	rho	7800	kg/m³	Basic

SOLID MECHANICS (SOLID)

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundaries 2, 3, and 6 only.

Prescribed Displacement 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4. From the **Displacement in x direction** list, choose **Prescribed**.
5. In the u_{0x} text field, type `-eps0*L0`.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. From the **Element size** list, choose **Extremely fine**.

Mapped 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Mapped**, click **Build All**.

Swept 1

1. In the **Mesh** toolbar, click **Swept**.
2. In the **Settings** window for **Swept**, click **Build All**.

STUDY 1

Solve for the natural frequencies in the undeformed case.

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

RESULTS

Mode Shape (Stress-Free)

In the **Settings** window for **3D Plot Group**, type `Mode Shape (Stress-Free)` in the **Label** text field.

Reproduce the plot in Figure 2 as follows.

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape (Stress-Free)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X-component**.
3 In the **Mode Shape (Stress-Free)** toolbar, click **Plot**.

ROOT

Add a new study to solve for the natural frequencies in the prestressed case.

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

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

RESULTS

Mode Shape (Prestressed)

In the **Settings** window for **3D Plot Group**, type `Mode Shape (Prestressed)` in the **Label** text field.

To reproduce the plot in Figure 3, follow these steps:

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X-component**.

3 In the **Mode Shape (Prestressed)** toolbar, click **Plot**.

Finally, reproduce the plot in Figure 4.

Displacement

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution Store 1 (sol3)**.

Surface 1

1 Right-click **Displacement** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X-component**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 Click the **Go to Default View** button in the **Graphics** toolbar.

3 In the **Displacement** toolbar, click **Plot**.