Created in COMSOL Multiphysics 6.1

Eigenvalue Analysis of a Crankshaft

Introduction

This application describes a modal analysis of a crankshaft. The pistons' reciprocating movement is transferred to the crankshaft through connecting rods by means of crankshaft throws. The forces, torques, and bending moments, which are highly variable both in time and space, subject the crankshaft to very high and complex loading. The crankshaft design must therefore incorporate careful and precise calculations of the vibrational characteristics.

Model Definition

GEOMETRY

The geometry comes from a NASTRAN mesh, which you import into the COMSOL Desktop.

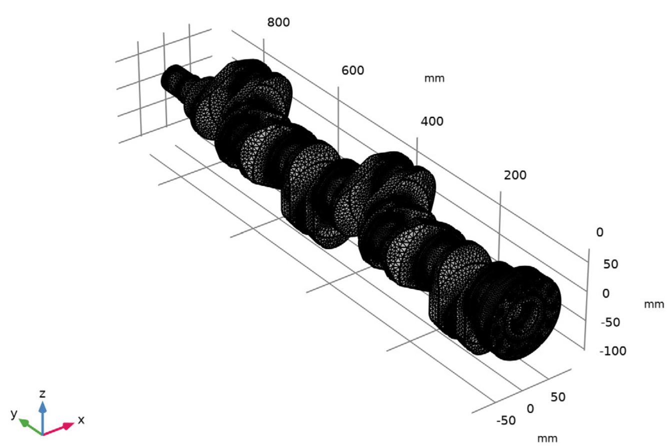

Figure 1: The NASTRAN mesh.

MATERIAL PROPERTIES AND BOUNDARY CONDITIONS

The crankshaft geometry is in millimeters.The crankshaft is made of structural steel. The boundary conditions are as follows:

- The crankshaft's main bearing surfaces are constrained from moving in the normal directions; that is, the crankshaft is allowed to rotate and slide at the bearing surfaces.

- The crankshaft is fixed at the rear surface where the flywheel is mounted.
- All the other boundaries are free.

Results

The analysis provides the lowest six eigenfrequencies. The first eigenmode is shown in Figure 2. It has an eigenfrequency of approximately 467.1 Hz, and the mode shape shows a mixture of axial and torsional motion, with the axial one dominating. Note that the eigenmode solution is scaled, so that the maximum value shown in the plot does not reflect the actual magnitude of any corresponding vibration. The type of scaling used can be changed under the setting for the eigenvalue solver.

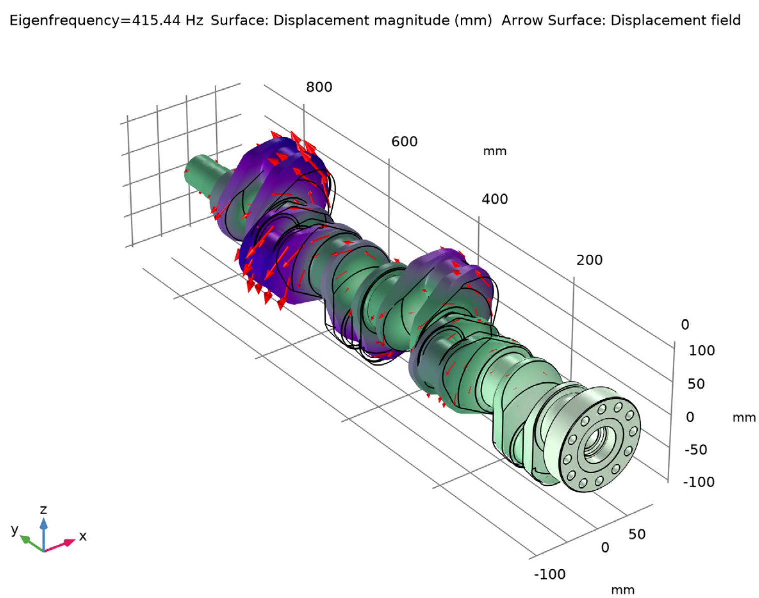

Figure 2: The first axial mode in the crankshaft.

The second distinctly shaped eigenmode has an eigenfrequency at 512.8 Hz. It is also a superposition of torsional and axial motion. But in contrast to the first one, in this case the torsion dominates Figure 3.

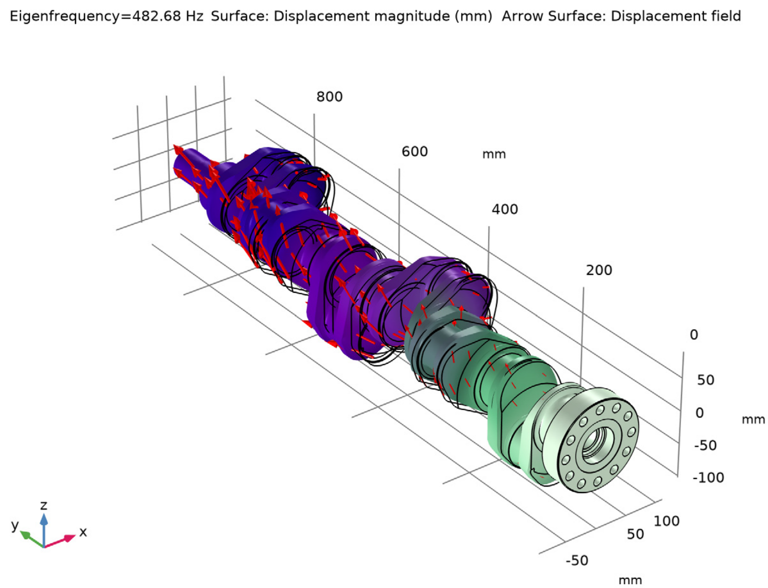

Figure 3: The torsional mode of the crankshaft throws.

The other higher modes show a similar behavior. Such mode shapes are caused by the nature of the constraints due to the main bearings, which allow certain axial sliding and torsion but prevent any significant bending of the shaft. Furthermore, the torsion implies either extension or contraction in the axial direction, so that the modes appear as superposition of both axial and torsional motion.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/crankshaft`

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**.

GEOMETRY 1

Import a NASTRAN mesh file under the mesh node which provides the mesh as well as domains, boundaries, and so on of the model. Before importing the file, select mm as the length unit to get the correct size for the geometry.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

MESH 1

Import 1

1 In the **Mesh** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 From the **Source** list, choose **NASTRAN file**.
4 From the **Data to import** list, choose **Only mesh**.
5 Click **Browse**.
6 Browse to the model's Application Libraries folder and double-click the file `crankshaft.nas`.
7 Click **Import**.
8 Click the **Zoom Extents** button in the **Graphics** toolbar.

MATERIALS

Now, define the material properties using structural steel from the built-in materials.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Structural steel**.

4 Click **Add to Component** in the window toolbar.

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

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

Change the element order to 1 so as to conform to the NASTRAN mesh.

3 From the **Displacement field** list, choose **Linear**.

Add a cylindrical coordinate system to be used when prescribing the constraints.

DEFINITIONS

Cylindrical System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

2 In the **Settings** window for **Cylindrical System**, locate the **Settings** section.

3 Find the **Longitudinal axis** subsection. In the table, enter the following settings:

x	y	z
0	1	0

SOLID MECHANICS (SOLID)

Prescribed Displacement 1

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

2 Select Boundaries 65, 67, 69, 71, 73, 75, 77, 112, and 113 only.

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

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Prescribed Displacement** section. Select the **Prescribed in r direction** check box.

These settings constrain the normal direction movements of the main bearing surfaces.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. Select Boundary 24 only.

STUDY 1

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

RESULTS

Mode Shape (solid)

The default plot shows the displacement magnitude and the deformed shape for the first mode in the crankshaft. Add the displacement arrow plot for better visualization of the mode type.

Arrow Surface 1

1. Right-click **Mode Shape (solid)** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.
3. Select the **Scale factor** check box. In the associated text field, type `7e4`.
4. In the **Mode Shape (solid)** toolbar, click **Plot**.
5. Click the **Go to Default View** button in the **Graphics** toolbar.

   The resulting plot should be similar to that shown in Figure 2. The mode presents a mixture of axial and torsional motion, but the axial one is dominating.

To get a plot showing the second eigenmode, follow these steps:

Mode Shape (solid)

1. In the **Model Builder** window, click **Mode Shape (solid)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Eigenfrequency (Hz)** list, choose **482.68**.
4. In the **Mode Shape (solid)** toolbar, click **Plot**.

   The resulting plot should look similar to that shown in Figure 3. It shows that the second mode is also a superposition of torsional and axial motion. In this case, the torsion dominates.