COMSOL MULTIPHYSICS®

Eigenvalue Analysis of a Crankshaft

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Eigenvalue Analysis of a Crankshaft

This model 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 Multiphysics user interface.

MATERIAL PROPERTIES AND BOUNDARY CONDITIONS

The crankshaft geometry is in millimeters, and a suitable unit system for geometries given in millimeters is the MPa system. The crankshaft is made of solid steel, and you give the material properties in the MPa unit system; that is, express the Young's modulus in MPa and the density in t/mm^3.

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 axial movement is constrained at the rear axial bearing surface.
- The crankshaft is fixed at the rear surface where the flywheel is mounted.

Results

The analysis provides the 20 first eigenfrequencies, of which several of them show the same modal shapes. This first eigenmode has an eigenfrequency at approximately 5778 Hz and it has a torsional characteristic. There are five additional eigenfrequencies in rather close vicinity to this frequency. Each of them also shows similar torsional characteristics, which suggests that they are all in fact showing the first torsional mode.

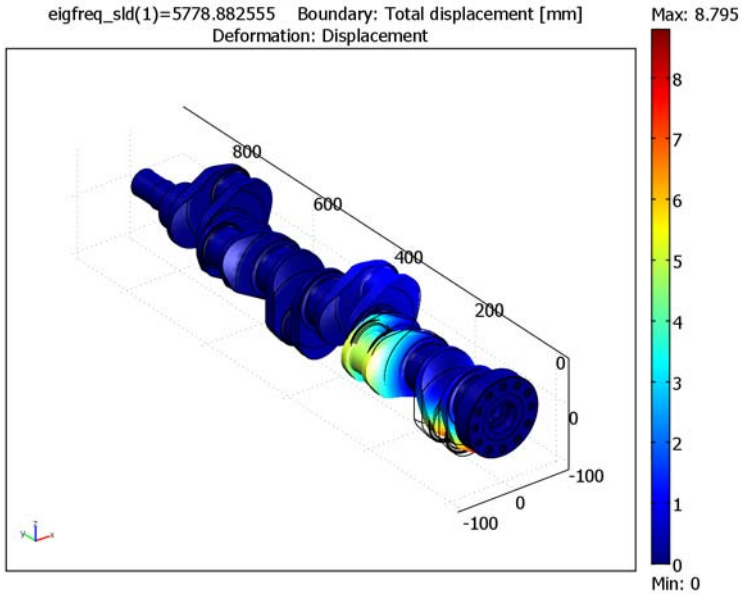


Figure 1: The first torsional mode in the crankshaft.

The next distinctly shaped eigenmode has an eigenfrequency at 8132 Hz. It can be characterized as a bending mode in the crankshaft throws. Here there are also five additional eigenfrequencies in rather close vicinity to this frequency. Each of these frequencies shows similar bending shapes as in the previously mentioned eigenmode at 8132 Hz. This suggests that they are all in fact showing the first bending mode in the crankshaft throws.

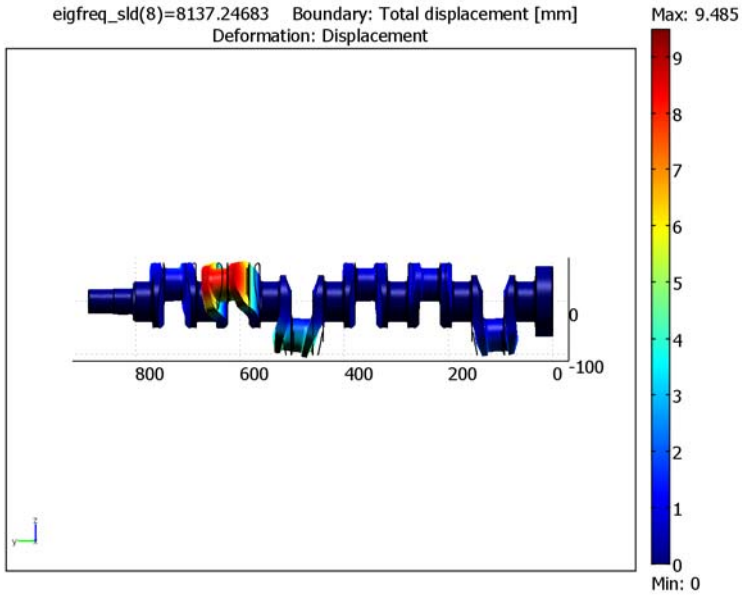


Figure 2: The bending mode of the crankshaft throws.

The third distinctly shaped eigenmode has an eigenfrequency at 8891 Hz. This modal shape can be characterized as a twisting mode in the crankshaft throws. There are, in the same manner as earlier, five additional eigenfrequencies in rather close vicinity to this frequency. Each of these frequencies shows similar twisting shapes.

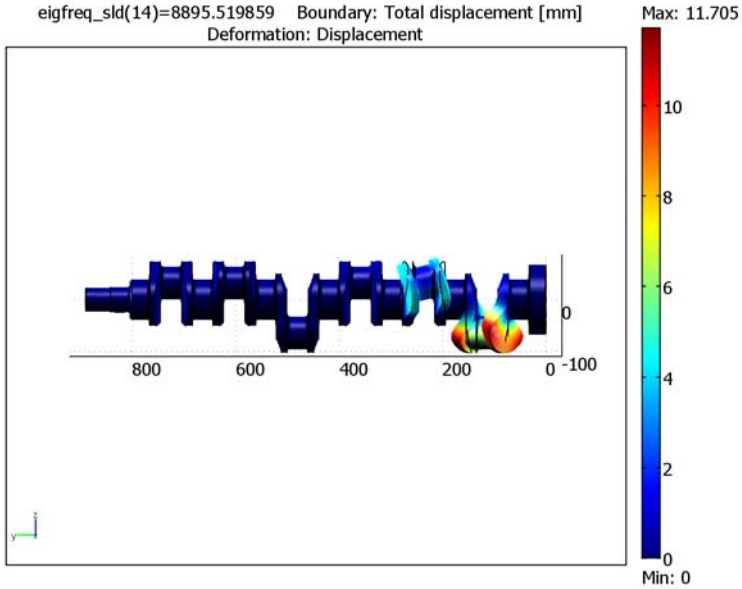


Figure 3: The twisting mode of the crankshaft throws.

Model Library path: `COMSOL_Multiphysics/Structural_Mechanics/crankshaft`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**, and in the **Space dimension** list select **3D**.

2 In the list of application modes select **COMSOL Multiphysics>Structural Mechanics**>**Solid, Stress-Strain**>**Eigenfrequency analysis**.

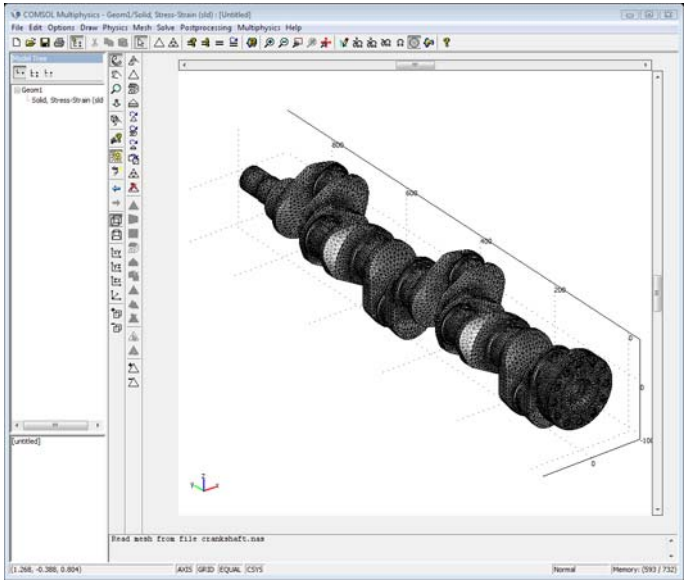
3 Click **OK**.

GEOMETRY MODELING

1 From the **File** menu choose **Import** and then **Mesh From File**.

2 Browse to the NASTRAN mesh file located at `models/COMSOL_Multiphysics/Structural_Mechanics/crankshaft.nas` in the COMSOL installation directory.

3 Click the **Import** button.



PHYSICS SETTINGS

Model Settings

1 From the **Physics** menu choose **Model Settings**.

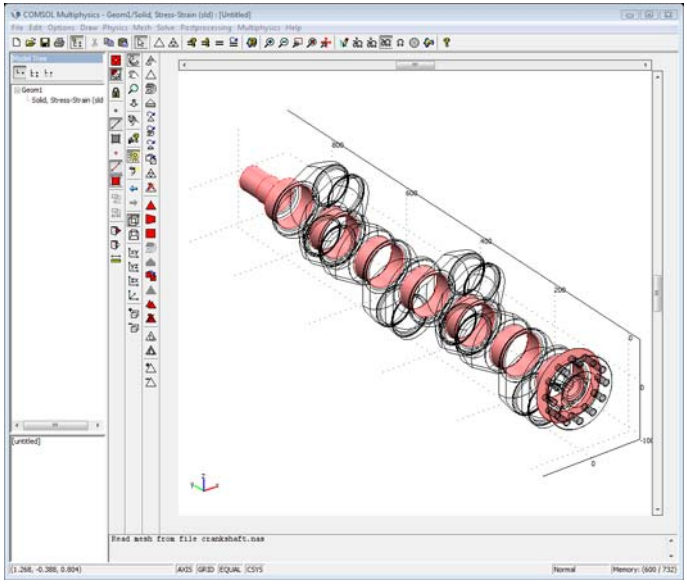
2 In the **Base unit system** list select **MPa**.

3 Click **OK**.

Boundary Conditions

1 From the **Physics** menu choose **Boundary Settings**.

2 In the **Boundary selection** list choose Boundaries 27, 55, 65, and 68–74.



3 Click the **General notation, Hu=R** button, then click the **Edit** button associated with the **H Matrix**. In the matrix position (1, 1) enter `nx`, in the (1, 2) position enter `ny`, and in the (1, 3) position enter `nz`. Click **OK**.

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

4 In the **Boundary selection** list choose **26**.

5 Click the **Standard notation** button, then select the $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$ check boxes to fix this surface.

6 Click **OK**.

Subdomain Settings

The crankshaft geometry is in millimeters. This means that you must enter the material properties in the MPa unit system to avoid having to scale the geometry; that is, you specify the Young's modulus in MPa and the density in t/mm^3.

1 From the **Physics** menu choose **Subdomain Settings**.

2 Select Subdomain 1, then enter the following material properties:

PROPERTY	VALUE
E	2.0e5
ν	0.33
ρ	7850e-12

3 Click the **Element** tab.

4 Select **Lagrange - Linear** in the **Predefined elements** list. The model uses linear elements to conform with the NASTRAN mesh.

5 Click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu choose **Solver Parameters**.

2 In the **General** page go to the **Desired number of eigenfrequencies** edit field and enter 20. Click **OK**.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

COMSOL Multiphysics plots the first eigenmode using a slice plot. To change the plot to show the deformed shape and other eigenmodes, follow these steps:

1 From the **Postprocessing** menu choose **Plot Parameters**.

2 In the **Plot type** area on the **General** page, clear the **Slice** check box, then select the **Boundary** and **Deformed shape** check boxes.

3 Select the second eigenfrequency in the **Eigenfrequency** list and click **Apply** to plot it.

Select the last eigenfrequency in the **Eigenfrequency** list and click **OK** to plot the twentieth eigenmode and close the **Plot Parameters** dialog box.