Created in COMSOL Multiphysics 5.6

Rotordynamic Analysis of a Crankshaft

Introduction

A crankshaft of a 3-cylinder reciprocating engine is studied in a vibration analysis. Due to the eccentricity of the crankpin and balance masses on the crankshaft, it undergoes self-excited vibration under rotation. The crankshaft is modeled using solid elements to capture the effects of the eccentricity of the crankpin and balance masses accurately.

Model Definition

The crankshaft of a three cylinder reciprocating engine is shown in Figure 1. Four bearing locations are also highlighted.

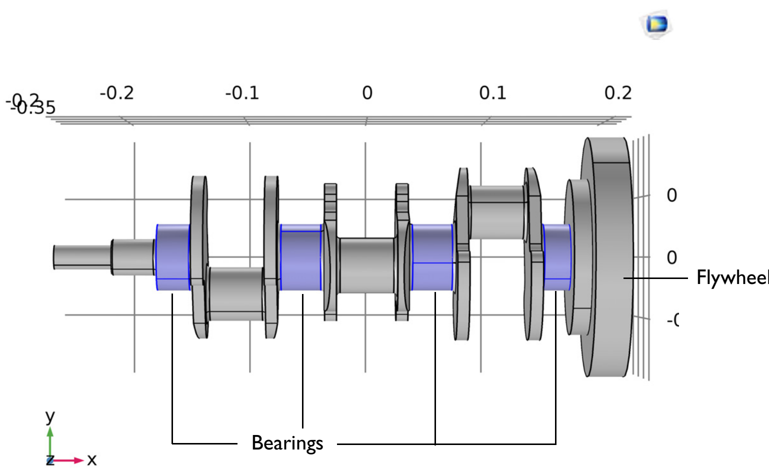

Figure 1: Crankshaft geometry.

The load on the crankpin due to the piston is neglected in the analysis, and the rotor undergoes only the self-excited vibration due to the eccentric masses. Material damping is used in the rotor to reduce high frequency vibrations. The angular speed of the crankshaft in the steady state is 3000 rpm, but it is ramped initially for a smooth startup. The duration of the ramp is chosen so that rotor completes one revolution with the linearly increasing speed from 0 to Ω and subsequently continues with the constant angular speed Ω. Assuming that the ramp duration is t_0 it then follows that

$$2\pi = \int_0^{t_0} \Omega_0 \frac{t}{t_0} dt = \frac{\Omega_0 t_0}{2}$$

Therefore,

$$t_0 = \frac{4\pi}{\Omega_0} = \frac{2}{f} = \frac{120}{N}$$

where f is the frequency corresponding to the angular speed and N is the rpm. Therefore, equation for the angular speed is

$$\Omega = \Omega_0\left\{\frac{t}{t_0}\left(\frac{t}{t_0} \le 1\right) + \left(\frac{t}{t_0} > 1\right)\right\} = \Omega_0 \mathrm{ramp}\left(\frac{t}{t_0}\right)$$

The Rayleigh coefficients for the damping are chosen such that the damping factor is close to 0.1 for the given angular speed of the rotor. The proportionality constants chosen for the analysis are

$$\alpha = 6.04 \qquad \beta = 5 \cdot 10^{-4}$$

Results and Discussion

Figure 2 shows the plot of the stress profile in the crankshaft. It can be seen that the bearing near the flywheel takes the maximum load so the stress has a maximum in the corresponding journal.

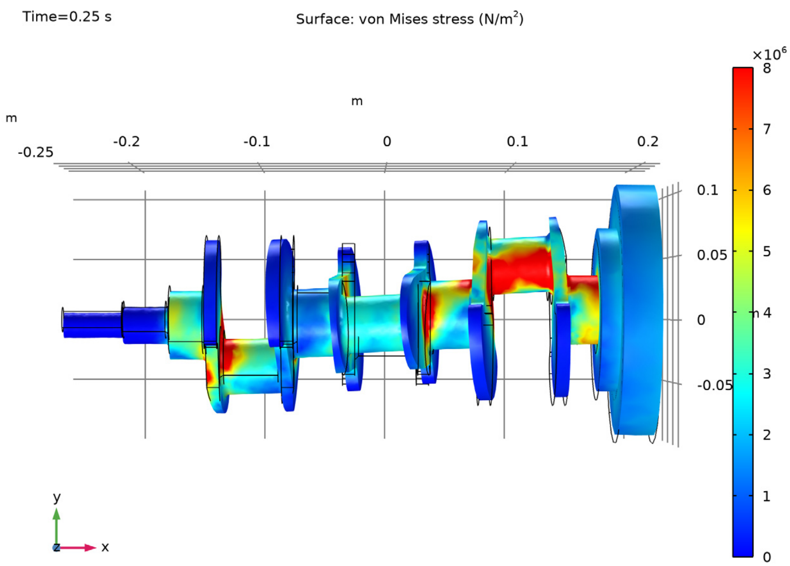

Figure 2: Stress in the crankshaft.

The pressure profile in the bearings is shown in Figure 3. One can clearly see the bearings at different locations are loaded in different directions due to the tilting of the shaft in the bearings.

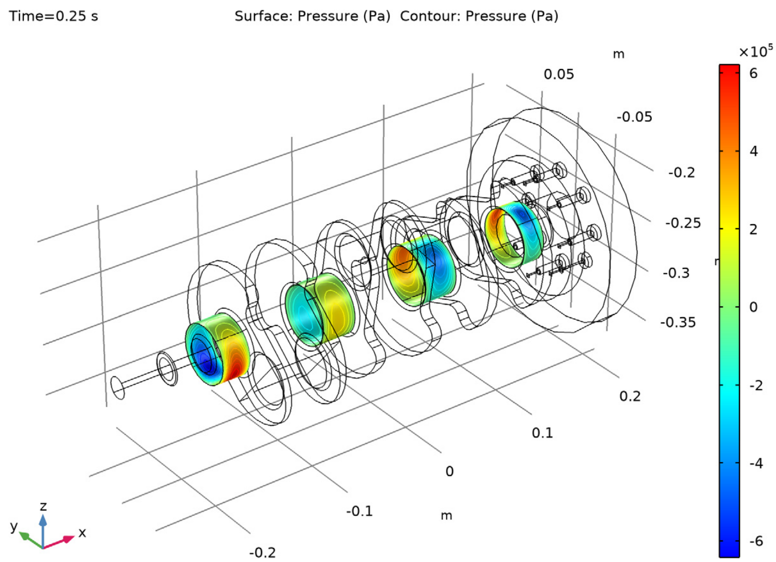

Figure 3: Pressure in the bearings.

The orbits of the center of the journals are shown in Figure 4. The orbits of all the journals are stable and the journals finally attain their respective equilibrium positions in the steady state.

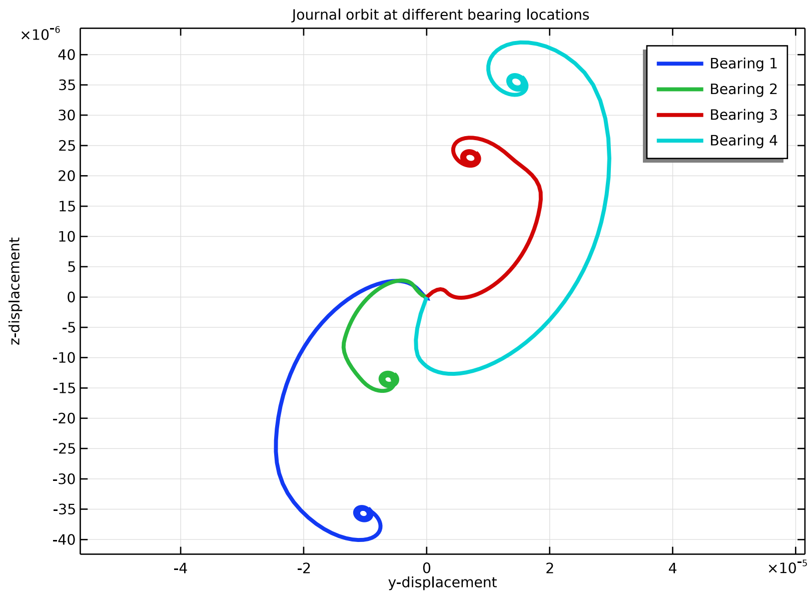

Figure 4: Journal orbits.

The lateral displacement components of the third journal are shown in Figure 5. The plot indicates that the lateral displacements of the journal undergo damped vibration and settle to an equilibrium value in the steady state as seen in the orbit plot in Figure 4.

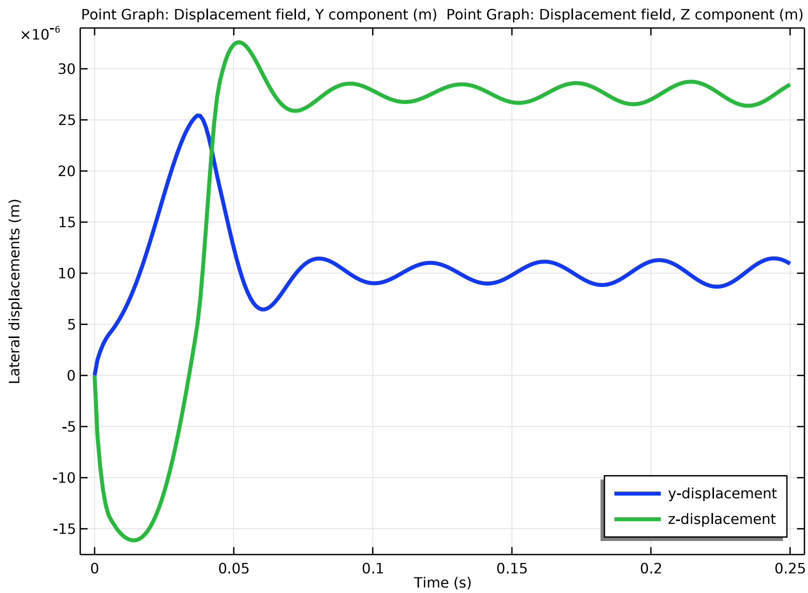

Figure 5: Lateral displacement components at journal 3.

Notes About the COMSOL Implementation

A **Solid Rotor with Hydrodynamic Bearing** multiphysics coupling is used to model the crankshaft-bearing assembly. This multiphysics coupling consists of a **Solid Rotor** physics node, a **Hydrodynamic Bearing** node, and a **Solid Rotor Bearing Coupling** multiphysics coupling node. The **Hydrodynamic Journal Bearing** feature of the **Hydrodynamic Bearing** physics interface is used to model the thin fluid-film flow in the journal bearing. You need one such node per bearing.

Application Library path: `Rotordynamics_Module/Automotive_and_Aerospace/reciprocating_engine_rotor`

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>Rotordynamics>Solid Rotor with Hydrodynamic Bearing**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

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

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

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

5 Click **Import**.

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**
Ow	3000[rpm]	50 1/s	Angular speed of the rotor
C	1e-4[m]	1E-4 m	Bearing clearance

Name	Expression	Value	Description
mu_l	0.072[Pa*s]	0.072 Pa·s	Lubricant viscosity
rho_l	864[kg/m^3]	864 kg/m³	Lubricant density

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.

DEFINITIONS

Define the ramp function for the angular speed of the rotor to get a smooth startup of the simulation.

Ramp 1 (rm1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Ramp**.

2 In the **Settings** window for **Ramp**, locate the **Parameters** section.

3 Select the **Cutoff** check box.

4 Click to expand the **Smoothing** section. Select the **Size of transition zone at start** check box.

5 In the associated text field, type `0.2`.

6 Select the **Size of transition zone at cutoff** check box.

7 In the associated text field, type `0.2`.

SOLID ROTOR (ROTSLD)

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

2 In the **Settings** window for **Solid Rotor**, locate the **Rotor Speed** section.

3 In the text field, type `Ow*rm1(Ow*t/2)`.

4 Locate the **Spin Softening** section. Clear the **Include spin softening** check box.

Set the discretization to linear for the displacement to reduce the simulation time. For more accurate results you can use the quadratic discretization.

5 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Linear Elastic Material 1

Add damping in the rotor to reduce the high frequency vibrations and stabilize the transient solver.

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor (rotsld)** click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Damping Settings** section.

3 In the α_{dM} text field, type `6.04`.

4 In the β_{dK} text field, type `0.0005`.

First Support 1

1 In the **Model Builder** window, click **First Support 1**.

2 Select Points 1 and 2 only.

Second Support 1

1 In the **Model Builder** window, click **Second Support 1**.

2 Select Points 232 and 241 only.

Fixed Axial Rotation 1

Suppress the axial rotation of the rotor at the flywheel end bearing.

1 In the **Model Builder** window, click **Fixed Axial Rotation 1**.

2 Select Boundary 128 only.

Suppress the axial displacement of the rotor using the thrust bearings.

Thrust Bearing 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thrust Bearing**.

2 Select Boundary 11 only.

Thrust Bearing 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Thrust Bearing**.

2 Select Boundary 131 only.

HYDRODYNAMIC BEARING (HDB)

Select only the surfaces corresponding to the bearing locations.

1 In the **Model Builder** window, expand the **Thrust Bearing 2** node, then click **Component 1 (comp1)>Hydrodynamic Bearing (hdb)**.

2 In the **Settings** window for **Hydrodynamic Bearing**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 12, 13, 50, 51, 88, 89, 126, and 127 only.

Hydrodynamic Journal Bearing 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Hydrodynamic Bearing (hdb)** click **Hydrodynamic Journal Bearing 1**.

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Bearing Properties** section.

3 In the C text field, type `C`.

4 Locate the **Fluid Properties** section. From the μ list, choose **User defined**. In the associated text field, type `mu_l`.

5 From the ρ list, choose **User defined**. In the associated text field, type `rho_l`.

Add more **Hydrodynamic Journal Bearing** nodes; one for each bearing.

Hydrodynamic Journal Bearing 2

1 Right-click **Component 1 (comp1)>Hydrodynamic Bearing (hdb)>Hydrodynamic Journal Bearing 1** and choose **Duplicate**.

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 50 and 51 only.

Hydrodynamic Journal Bearing 3

1 Right-click **Hydrodynamic Journal Bearing 2** and choose **Duplicate**.

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 88 and 89 only.

Hydrodynamic Journal Bearing 4

1 Right-click **Hydrodynamic Journal Bearing 3** and choose **Duplicate**.

2 In the **Settings** window for **Hydrodynamic Journal Bearing**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 126 and 127 only.

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 **Fine**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,1e-3,0.25)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

Set the appropriate scaling for the pressure.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Pressure (comp1.pfilm)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e5`.

Use the automatic damping in the Newton solver.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

8 From the **Nonlinear method** list, choose **Automatic (Newton)**.

9 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (rotsld)

The stress in the crankshaft, shown in Figure 2, is a default plot. Set the appropriate scale to highlight the deformation.

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

2 From the **View** list, choose **New view**.

3 In the **Stress (rotsld)** toolbar, click **Plot**.

Surface

1 In the **Model Builder** window, expand the **Stress (rotsld)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Minimum** text field, type `0`.

5 In the **Maximum** text field, type `8e6`.

6 Click to expand the **Quality** section. From the **Smoothing** list, choose **Inside material domains**.

7 Click the **Go to XY View** button in the **Graphics** toolbar.

Deformation

1 In the **Model Builder** window, expand the **Surface** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `500`.

5 In the **Stress (rotsld)** toolbar, click **Plot**.

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

The pressure in the bearing, shown in Figure 3, is a default plot.

Fluid Pressure (hdb)

1 In the **Model Builder** window, click **Fluid Pressure (hdb)**.

2 In the **Fluid Pressure (hdb)** toolbar, click **Plot**.

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

Create the cut point dataset at the center of the bearing locations. You will need this for plotting the orbit of the crankshaft at different bearing locations as shown in Figure 4.

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.

3 In the **X** text field, type `-0.16 -0.055 0.055 0.154`.

4 In the **Y** text field, type `0 0 0 0`.

5 In the **Z** text field, type `-0.28525 -0.28525 -0.28525 -0.28525`.

6 Click **Plot**.

Journal orbits

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Journal orbits` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `y-displacement`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `z-displacement`.

8 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

9 In the **Title** text area, type `Journal orbit at different bearing locations`.

Point Graph 1

1 Right-click **Journal orbits** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `w`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `v`.

6 Click to expand the **Coloring and Style** section. In the **Width** text field, type `3`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Bearing 1
Bearing 2
Bearing 3
Bearing 4

Journal orbits

1 In the **Model Builder** window, click **Journal orbits**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Preserve aspect ratio** check box.

4 In the **Journal orbits** toolbar, click **Plot**.

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

To plot the lateral displacements of a point on the third bearing, shown in Figure 5, follow the steps below.

Lateral displacements

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

2 In the **Settings** window for **1D Plot Group**, type `Lateral displacements` in the **Label** text field.

Point Graph 1

1 Right-click **Lateral displacements** and choose **Point Graph**.

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

3 Select the **Activate Selection** toggle button.

4 Select Point 158 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `v`.

6 Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`y-displacement`

10 In the **Lateral displacements** toolbar, click **Plot**.

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `w`.

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

Legends
`z-displacement`

Lateral displacements

1 In the **Model Builder** window, click **Lateral displacements**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box.

4 In the associated text field, type `Lateral displacements (m)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

6 In the **Lateral displacements** toolbar, click **Plot**.

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

To generate the animation of the crankshaft vibration, follow the steps below.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 In the **Number of frames** text field, type `100`.

4 Right-click **Animation 1** and choose **Play**.