Created in COMSOL Multiphysics 5.6

Shaft Vibration due to Gear Rattle and Bearing Misalignment

Introduction

In a gearbox, vibrations due to gear rattling and bearing misalignment are well known sources of noise. In this example, two shafts connected through a pair of gears are considered. The shafts are supported on roller bearings at their ends. Initially, the driven shaft is unloaded and the driver shaft rotates with a varying speed. Due to backlash, intermittent tooth meshing causes vibrations in the shafts. After some time, a resisting torque is applied to the driven shaft, making the tooth meshing smooth. In order to analyze the effect of bearing misalignment on rotor vibrations, a time-dependent analysis is performed for two cases. In the first case, all bearings are aligned with the shafts, and in the second, one of the bearings (number 2) has a small angular misalignment.

Note: This model requires the Multibody Dynamics Module and the Rotordynamics Module.

Model Definition

The model consists of two shafts connected through a pair of spur gears. The spur gear of the first (driver) shaft transfers rotation to the larger spur gear of the second (driven) shaft. Both shafts are supported at their ends using roller bearings.

The geometry is shown in Figure 1 below.

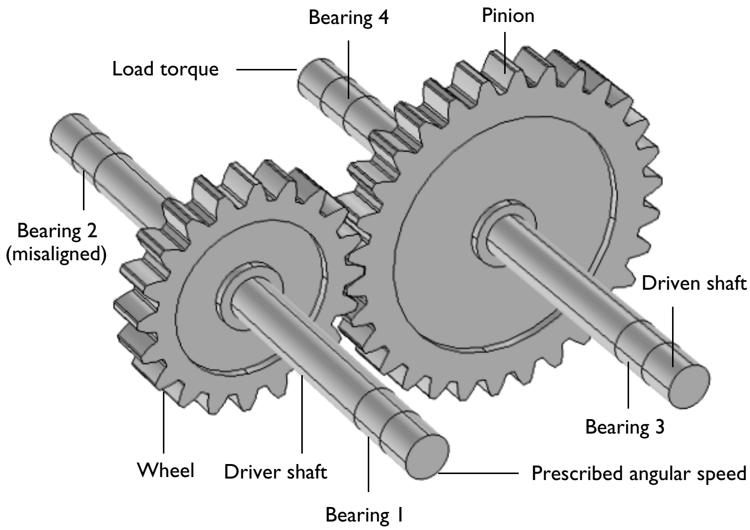

Figure 1: System geometry.

SHAFTS

The shafts are made of structural steel, with diameter and length of 20 mm and 300 mm, respectively. The mean angular speed of the driver shaft is assumed to be ω_0 = 20 rad/s. It fluctuates about this mean speed as:

$$\omega = \omega_0(1 + 0.05\sin(10\omega_0 t))$$

BEARINGS

Each shaft is supported by deep groove ball bearings at the ends. The bearings have the same dimensions and material properties. The properties are given in Table 1.

TABLE 1: BEARING PROPERTIES.

PROPERTY	VALUE
Number of balls	20
Ball diameter	1.33 mm
Pitch diameter	21.33 mm
Contour radius, inner race	0.7049 mm

TABLE 1: BEARING PROPERTIES.

PROPERTY	VALUE
Contour radius, outer race	0.7049 mm
Young's modulus	200 GPa
Poisson's ratio	0.3

GEARS

The properties of the spur gears are given in Table 2.

TABLE 2: GEAR PROPERTIES.

PROPERTY	VALUE
Number of teeth (Wheel)	20
Pitch diameter (Wheel)	100 mm
Number of teeth (Pinion)	30
Pitch diameter (Pinion)	150 mm
Pressure angle	25°
Gear ratio	1.5
Backlash	1 mm

The density of the gears, used to compute inertial properties, is equal to the shaft density.

CONSTRAINTS AND LOADS

- The axial rotation is prescribed at one end of the driver shaft.
- A resisting load torque of 100 Nm is applied at the opposite end of the driven shaft. The torque is activated only after the driver shaft has completed a 45° rotation.

Results and Discussion

Figure 2 shows the axial stress variation in the shafts. In addition to the torque, there are mainly two forces acting on the shafts. One, a gear mesh force acting in the pressure angle direction which bends the shafts in opposite direction and other the reaction forces from the bearing. Reaction forces in the bearing mainly support the shaft against the gear mesh force. Moreover, due to the angular misalignment in one of the bearings, an additional reaction moment is also present to overcome the misalignment in that bearing. Net axial stress in the shaft will be a combination of the bending of the shaft about two axes, one perpendicular to the pressure angle direction and the other parallel to the misalignment axis. From the stress distribution in the driving shaft it is clear that the bending in the shaft due to misalignment is larger as compared to the bending due to the gear mesh force. The bearing force direction on the driven shaft confirms that gear meshing forces are directly transmitted to the bearings in the pressure angle direction. Reaction in the bearings on the driving shaft are again combination of the gear mesh force and the reaction due to misalignment.

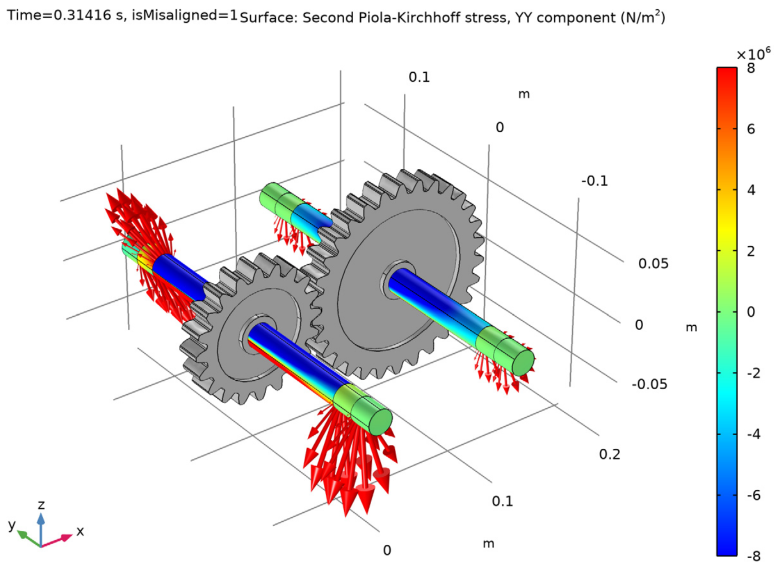

Figure 2: Stresses and bearing forces.

Figure 3 shows a comparison of angular speeds of the wheel and the pinion, for the cases of an aligned and a misaligned bearing. There is a higher torsional vibration for the case of a misaligned bearing, just after the driven shaft is loaded.

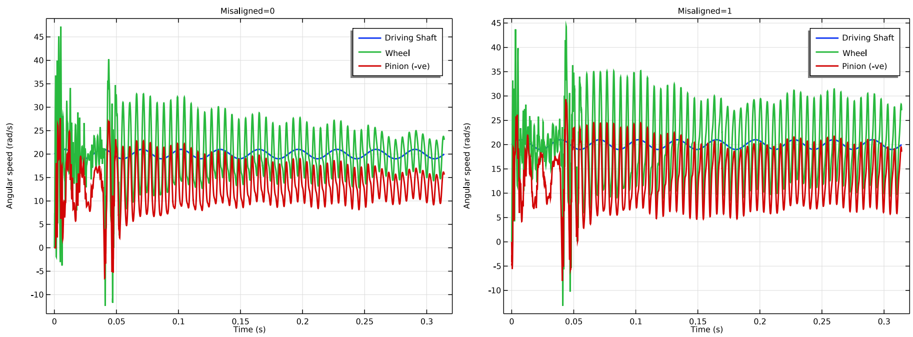

Figure 3: Angular velocity of the shafts for aligned bearings (left), and for a misaligned bearing (right).

The axial vibration at the wheel is shown in Figure 4. In the case of a misaligned bearing, axial vibrations (accelerations) are mainly due to the misalignment in the bearing. In the case of aligned bearings, axial vibrations appear when the torque is applied to the driven shaft. The vibrations reach their maximum immediately after the torque is applied, and they decay to lower levels after this initial stage. Figure 4 shows that the accelerations for the case of a misaligned bearing are significantly higher than that with aligned bearings.

Due to the angular misalignment in the bearings during operation, the force transmitted through the roller to the respective races has an axial component. This is the reason for the significant axial vibration of the shafts with misaligned bearing.

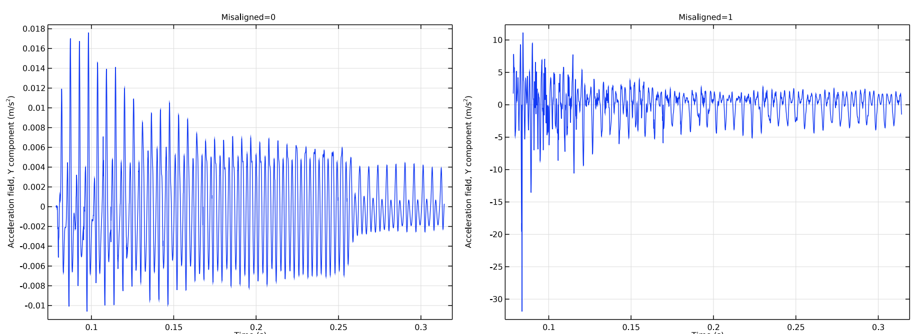

Figure 4: Axial acceleration at the wheel, for aligned bearings (left), and for a misaligned bearing (right).

Frequency spectra for axial vibrations are shown in Figure 5. Compared to the case of aligned bearings, the case of a misaligned bearing shows participation from a broader range of frequencies. More importantly acceleration levels are significantly higher in the misaligned case.

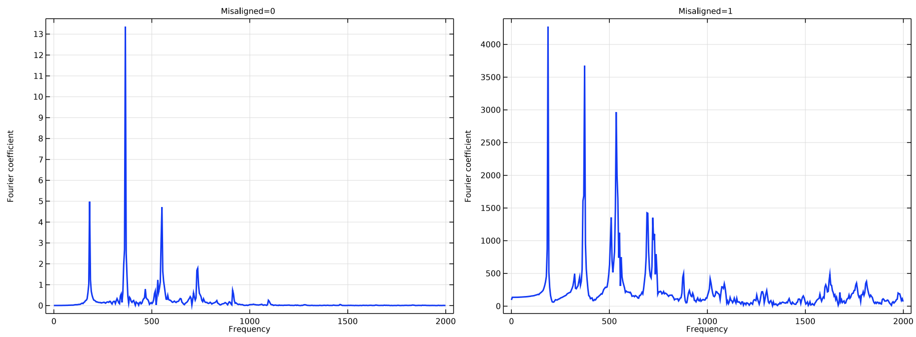

Figure 5: Frequency spectrum of the axial acceleration at the wheel, for the aligned (left) and misaligned (right) bearing.

Figure 6 shows the bearing force components for bearing 2. The case where the bearing is aligned is shown on the left, and the case where it is misaligned, on the right. In the misaligned case the force variations are larger..

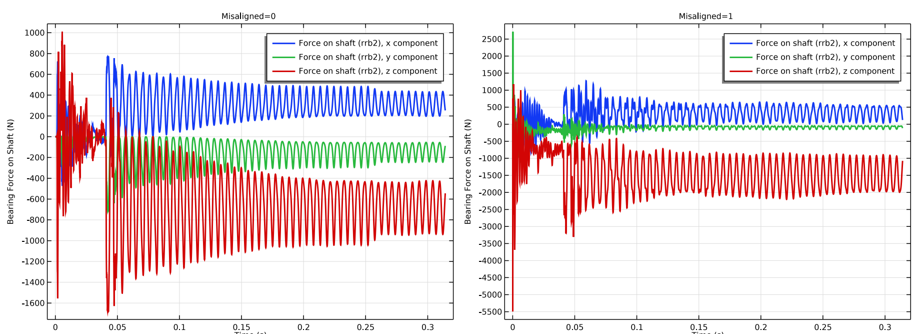

Figure 6: Force in bearing 2, for the aligned (left) and misaligned (right) bearing.

Figure 7 compares the moments in bearing 2. The result for the aligned bearings is presented on the left, and for the misaligned bearing on the right. One can clearly see a large moment about the x-axis in the misaligned case. The moment about z-axis also has larger variation for the case of a misaligned bearing. When the bearing is misaligned, there is a net moment reaction, because the roller loads are not passing from the center of the bearing.

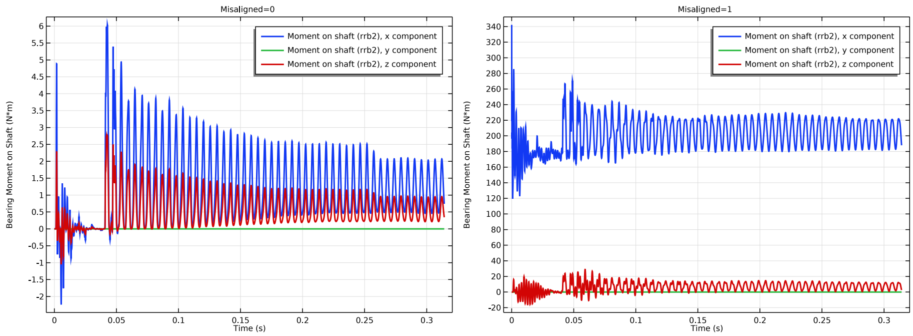

Figure 7: Moments in bearing 2, for the aligned (left) and misaligned (right) bearing.

A comparison of the rotor tilting in bearing 2 is shown in Figure 8 with results for the aligned case on the left and for the misaligned case on the right. Tilting of the rotor about x-axis is reversed in the misaligned case due to net reaction moment from the bearing. For aligned bearings case, the rotor bends due to gear mesh force in the pressure angle direction giving effective tilting about the x and z-axes. In the presence of the misalignment bearing reaction significantly changes the tilting about x-axis.

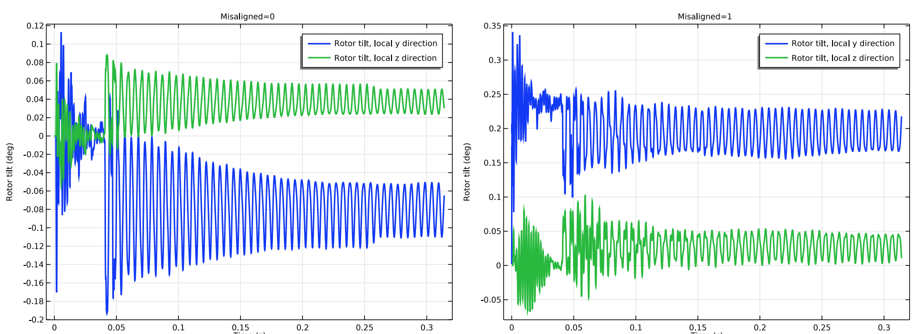

Figure 8: Rotor tilting in bearing 2, for the aligned (left) and misaligned (right) bearing.

The gear mesh contact force for the cases of aligned and misaligned bearings are shown in Figure 9. During the rattling vibration, intermittent contact in the gear meshing is clearly visible. The contact force variation is only lightly influenced by the misalignment in the bearing.

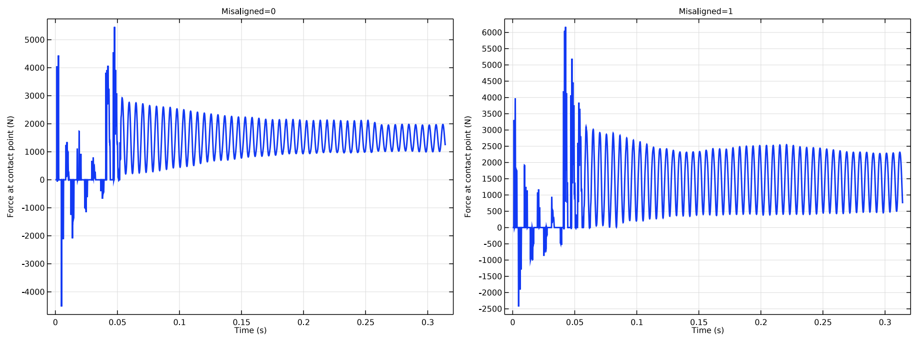

Figure 9: Gear mesh contact forces, for aligned bearings (left), and for a misaligned bearing (right).

Application Library path: `Multibody_Dynamics_Module/Tutorials, _Transmission/gear_rattle_with_bearing_misalignment`

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>Multibody Dynamics (mbd)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

Create a list of parameters for the speed and the loading torque.

GLOBAL DEFINITIONS

Parameters: General

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Parameters: General` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
omega0	20[rad/s]	20 rad/s	Mean angular speed
T0	100[N*m]	100 N·m	Loading torque
T	2*pi/omega0	0.31416 s	Time period
isMisaligned	0	0	Is misaligned

Create a list of parameters for the gears.

Parameters: Gears

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Parameters: Gears` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
N1	20	20	No of teeth on gear 1
N2	30	30	No of teeth on gear 2
dp1	100[mm]	0.1 m	Pitch diameter of gear 1
dp2	150[mm]	0.15 m	Pitch diameter of gear 2
gr	N2/N1	1.5	Gear ratio
rc	0.5*(dp1+dp2)	0.125 m	Center to center distance
bl	1e-3[m]	0.001 m	Backlash

Create a list of parameters for the bearings.

Parameters: Roller Bearings

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Parameters: Roller Bearings` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
db	1.33[mm]	0.00133 m	Ball diameter
dp	21.33[mm]	0.02133 m	Pitch diameter
rin	0.53*db	7.049E-4 m	Inner race radius
rout	0.53*db	7.049E-4 m	Outer race radius

Define a variable for the varying angular speed.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Omega	omega0*(1+0.05*sin(10*omega0*t))	rad/s	Angular speed

Start by creating the gear system geometry using the geometry from the Part Libraries.

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

3 In the **Part Libraries** window, select **Multibody Dynamics Module>3D>External Gears> spur_gear** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Spur Gear 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Spur Gear 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
n	N1	20	Number of teeth
dp	dp1	0.1 m	Pitch diameter
lsr	3	3	Shaft length to pitch diameter ratio (Set 0 for no shaft)
egy	1	1	Gear axis, y component
egz	0	0	Gear axis, z component

4 Click **Build Selected**.

Spur Gear 2 (pi2)

1 In the **Geometry** toolbar, click **Parts** and choose **Spur Gear**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
n	N2	30	Number of teeth
dp	dp2	0.15 m	Pitch diameter
dhr	0.2/gr	0.13333	Hole diameter to pitch diameter ratio (Set 0 for no hole)
wgr	0.2/gr	0.13333	Gear width to pitch diameter ratio
lsr	3/gr	2	Shaft length to pitch diameter ratio (Set 0 for no shaft)
xc	rc	0.125 m	Gear center, x coordinate
egy	1	1	Gear axis, y component
egz	0	0	Gear axis, z component
th	360[deg]/2/N2	6 °	Mesh alignment angle

4 Click **Build Selected**.

Create the workplanes to partition the shafts at the bearing locations.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

4 In the **y-coordinate** text field, type `-0.13`.

5 Click **Build Selected**.

Work Plane 2 (wp2)

1 Right-click **Work Plane 1 (wp1)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **y-coordinate** text field, type `0.13`.

Work Plane 3 (wp3)

1 Right-click **Work Plane 2 (wp2)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **y-coordinate** text field, type `-0.11`.

Work Plane 4 (wp4)

1 Right-click **Work Plane 3 (wp3)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **y-coordinate** text field, type `0.11`.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 From the **Work plane** list, choose **Work Plane 1 (wp1)**.

Partition Objects 2 (par2)

1 Right-click **Partition Objects 1 (par1)** and choose **Duplicate**.

2 Select the object **pi1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Find the **Objects to partition** subsection. Select the **Activate Selection** toggle button.

5 Click in the **Graphics** window and then press Ctrl+A to select both objects.

6 From the **Work plane** list, choose **Work Plane 2 (wp2)**.

7 Click **Build Selected**.

Partition Objects 3 (par3)

1 Right-click **Partition Objects 2 (par2)** and choose **Duplicate**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Work plane** list, choose **Work Plane 3 (wp3)**.

5 Click **Build Selected**.

Partition Objects 4 (par4)

1 Right-click **Partition Objects 3 (par3)** and choose **Duplicate**.

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

3 From the **Work plane** list, choose **Work Plane 4 (wp4)**.

4 Click in the **Graphics** window and then press Ctrl+A to select both objects.

5 Click **Build Selected**.

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.

Create a **Hinge Joint** to specify the angular speed for the driver shaft.

MULTIBODY DYNAMICS (MBD)

Attachment 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multibody Dynamics (mbd)** and choose **Attachment**.

2 Select Boundary 86 only.

Hinge Joint 1

1 In the **Physics** toolbar, click **Global** and choose **Hinge Joint**.

2 In the **Settings** window for **Hinge Joint**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Fixed**.

4 From the **Destination** list, choose **Attachment 1**.

5 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

6 Locate the **Joint Elasticity** section. From the list, choose **Elastic joint**.

You create the elastic joint to allow the lateral and tilting motion of the shaft. You will constrain this motion later by adding the **Roller Bearing** support to the shafts.

Prescribed Motion 1

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

2 In the **Settings** window for **Prescribed Motion**, locate the **Prescribed Rotational Motion** section.

3 From the **Prescribed motion through** list, choose **Angular velocity**.

4 In the ω_p text field, type `Omega`.

Radial Roller Bearing 1

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

2 Select Boundaries 87, 88, 118, and 119 only.

3 In the **Settings** window for **Radial Roller Bearing**, locate the **Bearing Orientation** section.

4 From the **Bearing axis** list, choose **y-axis**.

5 From the **Local y direction** list, choose **User defined**.

6 Specify the **Orientation vector defining local y direction** vector as

1	x
0	y
0	z

7 Locate the **Geometric Properties** section. In the d_b text field, type `db`.

8 In the d_p text field, type `dp`.

9 In the r_{in} text field, type `rin`.

10 In the r_{out} text field, type `rout`.

Radial Roller Bearing 2

1 Right-click **Radial Roller Bearing 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 97, 98, 134, and 135 only.

Radial Roller Bearing 3

1 Right-click **Radial Roller Bearing 2** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 343, 344, 378, and 385 only.

Radial Roller Bearing 4

1 Right-click **Radial Roller Bearing 3** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 353, 354, 382, and 389 only.

Add angular misalignment in the bearing located opposite to the prescribed end of the driver shaft. Use a parameter `isMisaligned` to enable/disable the misalignement in the bearing.

Radial Roller Bearing 2

In the **Model Builder** window, click **Radial Roller Bearing 2**.

Misalignment 1

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

2 In the **Settings** window for **Misalignment**, locate the **Angular Misalignment** section.

3 In the θ_{0y} text field, type `1[deg]*isMisaligned`.

Spur Gear 1

1 In the **Physics** toolbar, click **Domains** and choose **Spur Gear**.

2 Select Domain 1 only.

3 In the **Settings** window for **Spur Gear**, locate the **Gear Properties** section.

4 In the n text field, type `N1`.

5 In the d_p text field, type `dp1`.

6 In the α text field, type `25[deg]`.

7 Locate the **Gear Axis** section. Specify the $\mathbf{e}_g$ vector as

0	x
1	y
0	z

8 Locate the **Center of Rotation** section. From the list, choose **User defined**.

Spur Gear 2

1 Right-click **Spur Gear 1** and choose **Duplicate**.

2 In the **Settings** window for **Spur Gear**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 7 only.

5 Locate the **Gear Properties** section. In the n text field, type `N2`.

6 In the d_p text field, type `dp2`.

7 Locate the **Center of Rotation** section. Specify the $\mathbf{X}_c$ vector as

rc	x
0	y
0	z

Gear Pair 1

1 In the **Physics** toolbar, click **Global** and choose **Gear Pair**.

2 In the **Settings** window for **Gear Pair**, locate the **Gear Selection** section.

3 From the **Wheel** list, choose **Spur Gear 1**.

4 From the **Pinion** list, choose **Spur Gear 2**.

5 Locate the **Gear Pair Properties** section. Select the **Include backlash** check box.

6 Locate the **Contact Force Computation** section. From the list, choose **Computed using penalty method**.

7 In the p_c text field, type `1e8`.

Backlash 1

1 In the **Model Builder** window, click **Backlash 1**.

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

3 In the b_l text field, type `bl`.

Increase the penalty factor 100 times to reduce the error in the backlash.

4 In the p_b text field, type `((1[1/ms])^2)*mbd.grp1.Ie*100`.

Rigid Connector 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundary 359 only.

Add a loading torque on the driven shaft. Activate the torque only after the driver shaft has completed a 45 degree rotation.

Applied Moment 1

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

2 In the **Settings** window for **Applied Moment**, locate the **Applied Moment** section.

3 Specify the **M** vector as

`0`	x
`T0*(t>T/8)`	y
`0`	z

Use a swept mesh for the shafts.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 86 and 342 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

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

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–6 and 8–12 only.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

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,T/5000,T)`.

Add an auxiliary sweep to solve for two cases: first, all bearings aligned, and second, one of the bearings misaligned.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
isMisaligned (Is misaligned)	0 1	

Change the settings for the time stepping and maximum number of iterations in the solver to reduce the computation time.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Intermediate**.

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

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

7 From the **Jacobian update** list, choose **Once per time step**.

8 In the **Maximum number of iterations** text field, type `10`.

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

RESULTS

Displacement (mbd)

Displacement is the default plot. Duplicate it and follow the instructions below to create the stress plot as shown in Figure 2.

Stress (mbd)

1 Right-click **Displacement (mbd)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Stress (mbd)` in the **Label** text field.

Surface

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

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `mbd.SYY`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `-8000000`.

6 In the **Maximum** text field, type `8000000`.

Gears use the **Rigid Domain** material model. Therefore, it is not possible to plot the stress in the gears. Duplicate the existing dataset and restrict the new dataset selection to gears only. You will use this dataset to display the gears in a stress plot.

Study 1/Solution 1 (2) (sol1)

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 7 only.

Surface 2

1 In the **Model Builder** window, right-click **Surface** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

7 From the **Solution parameters** list, choose **From parent**.

Add arrow plots to plot the force distribution in the roller bearings.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Stress (mbd)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Radial Roller Bearing 1>mbd.rrb1.fbx,...,mbd.rrb1.fbz - Bearing force distribution (spatial frame)**.

3 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

4 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

5 In the associated text field, type `6E-4`.

Deformation 1

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

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

3 In the **X component** text field, type `mbd.rrb1.u_cage`.

4 In the **Y component** text field, type `mbd.rrb1.v_cage`.

5 In the **Z component** text field, type `mbd.rrb1.w_cage`.

6 Locate the **Scale** section. Select the **Scale factor** check box.

7 In the associated text field, type `1`.

Arrow Surface 2

1 In the **Model Builder** window, under **Results>Stress (mbd)** right-click **Arrow Surface 1** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Arrow Surface 1**.

Due to misalignment, the reaction forces are significantly larger in this bearing. Do not inherit the arrow scale for this bearing.

4 Clear the **Arrow scale factor** check box.

5 Locate the **Coloring and Style** section. In the **Scale factor** text field, type `2E-5`.

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

7 Locate the **Expression** section. In the **X component** text field, type `mbd.rrb2.fbx`.

8 In the **Y component** text field, type `mbd.rrb2.fby`.

9 In the **Z component** text field, type `mbd.rrb2.fbz`.

Deformation 1

1 In the **Model Builder** window, expand the **Arrow Surface 2** node, then click **Deformation 1**.

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

3 In the **X component** text field, type `mbd.rrb2.u_cage`.

4 In the **Y component** text field, type `mbd.rrb2.v_cage`.

5 In the **Z component** text field, type `mbd.rrb2.w_cage`.

Arrow Surface 3

1 In the **Model Builder** window, under **Results>Stress (mbd)** right-click **Arrow Surface 1** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **X component** text field, type mbd.rrb3.fbx.

4 In the **Y component** text field, type mbd.rrb3.fby.

5 In the **Z component** text field, type mbd.rrb3.fbz.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Arrow Surface 1**.

Deformation 1

1 In the **Model Builder** window, expand the **Arrow Surface 3** node, then click **Deformation 1**.

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

3 In the **X component** text field, type mbd.rrb3.u_cage.

4 In the **Y component** text field, type mbd.rrb3.v_cage.

5 In the **Z component** text field, type mbd.rrb3.w_cage.

Arrow Surface 4

1 In the **Model Builder** window, under **Results>Stress (mbd)** right-click **Arrow Surface 3** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **X component** text field, type mbd.rrb4.fbx.

4 In the **Y component** text field, type mbd.rrb4.fby.

5 In the **Z component** text field, type mbd.rrb4.fbz.

Deformation 1

1 In the **Model Builder** window, expand the **Arrow Surface 4** node, then click **Deformation 1**.

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

3 In the **X component** text field, type mbd.rrb4.u_cage.

4 In the **Y component** text field, type mbd.rrb4.v_cage.

5 In the **Z component** text field, type mbd.rrb4.w_cage.

Stress (mbd)

1 In the **Model Builder** window, click **Stress (mbd)**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type Surface: Second Piola-Kirchhoff stress, YY component (N/m²) .

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

6 In the **Stress (mbd)** toolbar, click **Plot**.

Follow the instructions below to plot the angular speed of the driving shaft, wheel and pinion, as shown in Figure 3.

Angular Speed

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

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

3 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **First**.

Global 1

1 Right-click **Angular Speed** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Variables>Omega - Angular speed - rad/s**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`Omega`	`rad/s`	`Driving Shaft`

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Gear pairs>Gear Pair 1> Wheel>mbd.grp1.tht_wh - Wheel angular velocity - rad/s**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`mbd.grp1.tht_wh`	`rad/s`	`Wheel`

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Gear pairs>Gear Pair 1> Pinion>mbd.grp1.tht_pn - Pinion angular velocity - rad/s**.

7 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`-mbd.grp1.tht_pn`	`rad/s`	`Pinion (-ve)`

8 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

9 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Driving Shaft`
`Wheel`
`Pinion (-ve)`

Angular Speed

1 In the **Model Builder** window, click **Angular Speed**.

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 `Angular speed (rad/s)`.

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

6 Select the **Allow evaluation of expressions** check box.

7 In the **Title** text area, type `Misaligned=eval(isMisaligned)`.

8 In the **Angular Speed** toolbar, click **Plot**.

Change the `isMisaligned` parameter to plot the angular velocity for the misaligned case.

9 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **Last**.

10 In the **Angular Speed** toolbar, click **Plot**.

You can compare the reaction forces and moments of the bearing for aligned and misaligned cases. These plots are shown in Figure 6 and Figure 7 and can be reproduced using the instructions below.

Bearing Force

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

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

3 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **First**.

Global 1

1 Right-click **Bearing Force** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Radial Roller Bearing 2>Force on shaft (rrb2) (spatial frame) - N>All expressions in this group**.

3 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

4 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Force on shaft (rrb2), x component`
`Force on shaft (rrb2), y component`
`Force on shaft (rrb2), z component`

6 In the **Bearing Force** toolbar, click **Plot**.

Bearing Force

1 In the **Model Builder** window, click **Bearing Force**.

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 `Bearing Force on Shaft (N)`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 Select the **Allow evaluation of expressions** check box.

7 In the **Title** text area, type `Misaligned=eval(isMisaligned)`.

8 In the **Bearing Force** toolbar, click **Plot**.

9 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **Last**.

10 In the **Bearing Force** toolbar, click **Plot**.

Bearing Moment

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

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

3 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **First**.

Global 1

1 Right-click **Bearing Moment** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Radial Roller Bearing 2>Moment on shaft (rrb2) (spatial frame) - N·m>All expressions in this group**.

3 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

4 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Moment on shaft (rrb2), x component`
`Moment on shaft (rrb2), y component`
`Moment on shaft (rrb2), z component`

Bearing Moment

1 In the **Model Builder** window, click **Bearing Moment**.

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 `Bearing Moment on Shaft (N*m)`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 Select the **Allow evaluation of expressions** check box.

7 In the **Title** text area, type `Misaligned=eval(isMisaligned)`.

8 In the **Bearing Moment** toolbar, click **Plot**.

9 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **Last**.

10 In the **Bearing Moment** toolbar, click **Plot**.

Angular misalignment in the bearings causes large axial vibrations. Start by creating a **Cut Point** at the center of the wheel to plot the axial vibration at this location.

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`.

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

5 In the **Z** text field, type `0`.

6 Click **Plot**.

Follow the instructions below to plot the axial vibration at the center of the wheel for aligned and misaligned cases. This plot is shown in Figure 4.

Y Acceleration of Wheel

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

2 In the **Settings** window for **1D Plot Group**, type `Y Acceleration of Wheel` in the **Label** text field.

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

4 From the **Time selection** list, choose **Manual**.

5 In the **Parameter indices (1-5001)** text field, type `range(1250,1,5001)`.

6 From the **Parameter selection (isMisaligned)** list, choose **First**.

Point Graph 1

1 Right-click **Y Acceleration of Wheel** 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 `vtt`.

Y Acceleration of Wheel

1 In the **Model Builder** window, click **Y Acceleration of Wheel**.

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

3 From the **Title type** list, choose **Manual**.

4 Select the **Allow evaluation of expressions** check box.

5 In the **Title** text area, type `Misaligned=eval(isMisaligned)`.

6 In the **Y Acceleration of Wheel** toolbar, click **Plot**.

7 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **Last**.

8 In the **Y Acceleration of Wheel** toolbar, click **Plot**.

Duplicate the axial vibration plot and follow the instructions below to plot the frequency spectrum of the axial vibrations for aligned and misaligned cases, as shown in Figure 5.

Y Acceleration of Wheel (Frequency Spectrum)

1 Right-click **Y Acceleration of Wheel** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Y Acceleration of Wheel (Frequency Spectrum)` in the **Label** text field.

The gear rattle due to unloaded shaft causes initial transient vibration in the system. Remove this transient vibration data for the frequency spectrum plot.

3 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **First**.

Point Graph 1

1 In the **Model Builder** window, expand the **Y Acceleration of Wheel (Frequency Spectrum)** node, then click **Point Graph 1**.

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

3 From the **Parameter** list, choose **Frequency spectrum**.

4 Select the **Frequency range** check box.

5 In the **Maximum** text field, type 2000.

6 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

7 In the **Y Acceleration of Wheel (Frequency Spectrum)** toolbar, click **Plot**.

Y Acceleration of Wheel (Frequency Spectrum)

1 In the **Model Builder** window, click **Y Acceleration of Wheel (Frequency Spectrum)**.

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

3 From the **Parameter selection (isMisaligned)** list, choose **Last**.

4 In the **Y Acceleration of Wheel (Frequency Spectrum)** toolbar, click **Plot**.

The rotor tilting in the bearing is compared in Figure 8 for aligned and misaligned cases. Follow the instructions below to reproduce this plot.

Rotor tilt at Bearing 2

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

2 In the **Settings** window for **1D Plot Group**, type Rotor tilt at Bearing 2 in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **First**.

Global 1

1 Right-click **Rotor tilt at Bearing 2** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
mbd.rrb2.alphay	deg	Tilt of rotor about local y direction
mbd.rrb2.alphaz	deg	Tilt of rotor about local z direction

4 Locate the **Coloring and Style** section. In the **Width** text field, type 2.

5 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
Rotor tilt, local y direction
Rotor tilt, local z direction

Rotor tilt at Bearing 2

1 In the **Model Builder** window, click **Rotor tilt at Bearing 2**.

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 `Rotor tilt (deg)`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 Select the **Allow evaluation of expressions** check box.

7 In the **Title** text area, type `Misaligned=eval(isMisaligned)`.

8 In the **Rotor tilt at Bearing 2** toolbar, click **Plot**.

9 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **Last**.

10 In the **Rotor tilt at Bearing 2** toolbar, click **Plot**.

Gear mesh contact force is an indicator of the rattling vibrations in shafts. This plot is shown in Figure 9, and it can be reproduced by using the instructions below.

Gear Mesh Contact Force

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

2 In the **Settings** window for **1D Plot Group**, type `Gear Mesh Contact Force` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **First**.

Global 1

1 Right-click **Gear Mesh Contact Force** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Gear pairs>Gear Pair 1>mbd.grp1.Fc - Force at contact point - N**.

3 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

Gear Mesh Contact Force

1 In the **Model Builder** window, click **Gear Mesh Contact Force**.

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

3 From the **Title type** list, choose **Manual**.

4 Select the **Allow evaluation of expressions** check box.

5 In the **Title** text area, type `Misaligned=eval(isMisaligned)`.

6 Locate the **Legend** section. Clear the **Show legends** check box.

7 In the **Gear Mesh Contact Force** toolbar, click **Plot**.

8 Locate the **Data** section. From the **Parameter selection (isMisaligned)** list, choose **Last**.

9 In the **Gear Mesh Contact Force** toolbar, click **Plot**.

Finally, use the instructions below to generate the animation of the stress variation in the shafts.

Animation 1

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

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

3 From the **Subject** list, choose **Stress (mbd)**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `50`.

5 Locate the **Animation Editing** section. From the **Parameter value (isMisaligned)** list, choose **1**.

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