Created in COMSOL Multiphysics 5.6

Rotors Connected Through Helical Gears

Introduction

This example illustrates the modeling of multiple rotors connected through helical gears. The presence of gears in the system induces lateral as well as torsional vibration in the rotors. An eigenfrequency analysis is performed for different speeds of the driver shaft in order to compute the critical speeds. The frequency response of the system is also computed for a range of frequencies, assuming the speed of the driver shaft to be the same as the excitation frequency. A transient analysis is performed for the given speed and load to compute the orbit and the dynamic transmission error of the gears as well as the forces acting on the bearings.

Model Definition

The model consists of three shafts connected through two pairs of helical gears. The helical gear on the first (driver) shaft transfers rotation to the larger helical gear of the second (intermediate) shaft. The smaller helical gear of the second (intermediate) shaft transfers rotation to the helical gear of the third (driven) shaft. All three shafts are simply supported at both the ends using journal bearings.

The geared rotor geometry is shown in Figure 1 below.

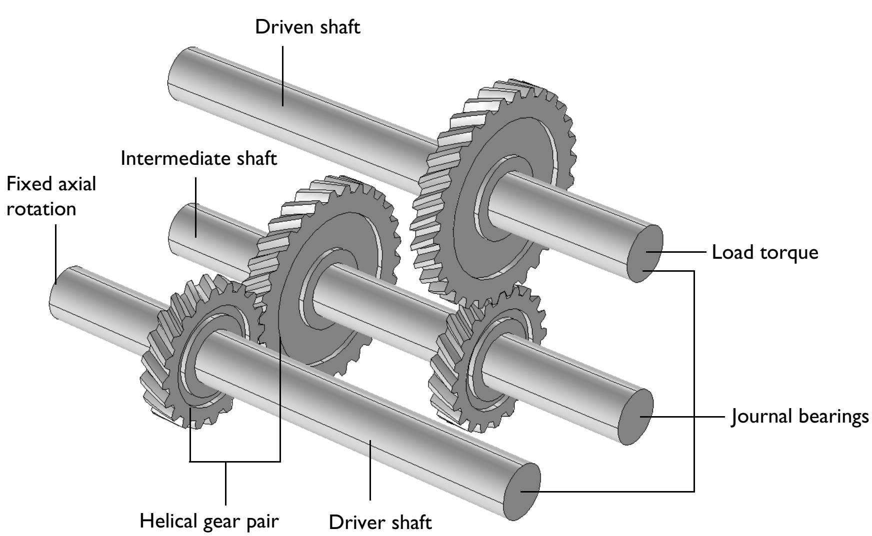

Figure 1: Geared rotor geometry.

SHAFTS

All the shafts are of the same dimensions and made of structural steel. The shafts' diameter and length are 40 mm and 600 mm, respectively.

In the time-dependent analysis, the angular speed of the driver shaft is assumed to be 1500 rpm. In the eigenfrequency, it is varied from 0 to 9000 rpm whereas in the frequency domain analysis, it is assumed to be the same as the excitation frequency. The angular speed of the other two shafts is computed using the gear ratio.

BEARINGS

Each shaft is supported by journal bearings at both the ends. All the bearings are assumed to have constant isotropic stiffness with a value of 10^7 N/m.

GEARS

The properties of the helical gears are given in the Table 1.

TABLE 1: GEAR PROPERTIES.

PROPERTY	VALUE
Number of teeth (Gear-1)	20
Pitch diameter (Gear-1)	100 mm
Number of teeth (Gear-2)	30
Pitch diameter (Gear-2)	150 mm
Pressure angle	25°
Helix angle	30°
Gear ratio	1.5
Gear mesh stiffness	$2 \cdot 10^6$ N/m

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

CONSTRAINTS AND LOADS

- The axial rotation is constrained at one end of the driver shaft.
- In the time dependent analysis, a resisting load torque of 10^2 Nm is applied at the opposite end of the driven shaft.

CRITICAL SPEEDS OF A ROTOR

The angular speeds of a rotor that match its natural frequencies are known as the critical speeds. Approximate values of the critical speeds of a rotor can be obtained as the rotor's eigenfrequencies computed while neglecting rotating frame forces. However, to predict the eigenfrequencies accurately, the rotating frame forces for a range of angular speeds must be included in the computations. The Campbell diagram, showing the variation of eigenfrequencies with angular speed, can be drawn and accurate critical speeds can be computed. A Campbell plot using the Solid Rotor interface plots the frequencies in the rotating frame. In such a plot critical speeds are obtained by the intersection of the backward whirl frequency curve with $\omega = 2\Omega$ curve and intersection of the forward whirl curve with the x-axis. In the rotating frame, generally the backward whirl frequencies increase with the rotor speed and forward whirl frequencies decrease.

Results and Discussion

There are five points where the $\omega = 2\Omega$ curve intersects with the eigenfrequency curve. These are the critical speeds of the rotor system below 9000 rpm and they are listed in the Table 2.

TABLE 2: CRITICAL SPEEDS OF THE ROTOR.

Mode	Critical speeds (rad/s)
First	281.76
Second	473.68
Third	548.91
Fourth	595.05
Fifth	668.68

.

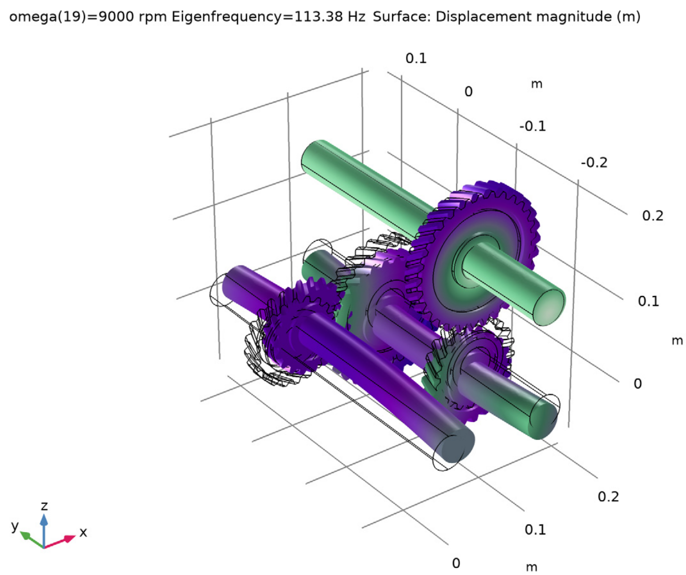

Figure 2: The second mode of the geared rotors at 9000 rpm.

Figure 2 shows the second mode (f = 113.4 Hz) of the geared rotors at 9000 rpm. The Campbell Plot, showing the variation of eigenfrequencies with the rotational speed, is shown in Figure 3. The three critical speeds of the rotor, where the straight line ($\omega = 2\Omega$) intersects the eigenfrequency curves, can be seen.

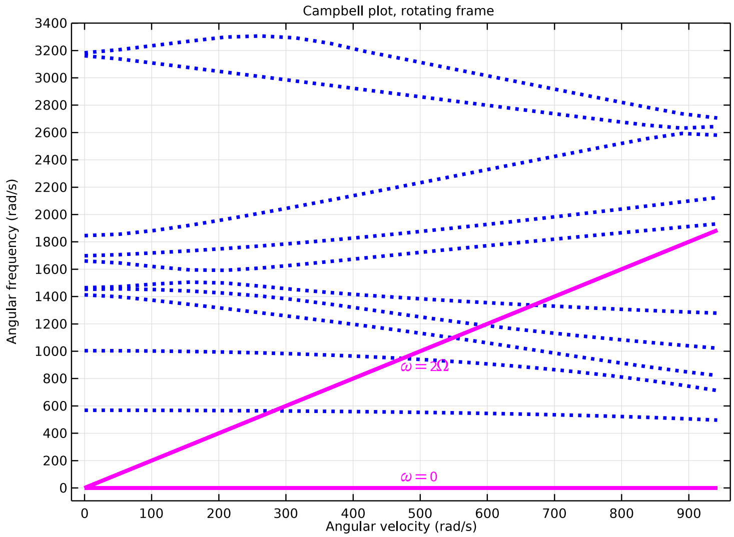

Figure 3: Campbell diagram.

The frequency response curves for the gear displacement and rotation are shown in Figure 4 and Figure 5. It can be seen that both the displacement and rotation of all gears have a peak around the critical speeds computed from the Campbell plot.

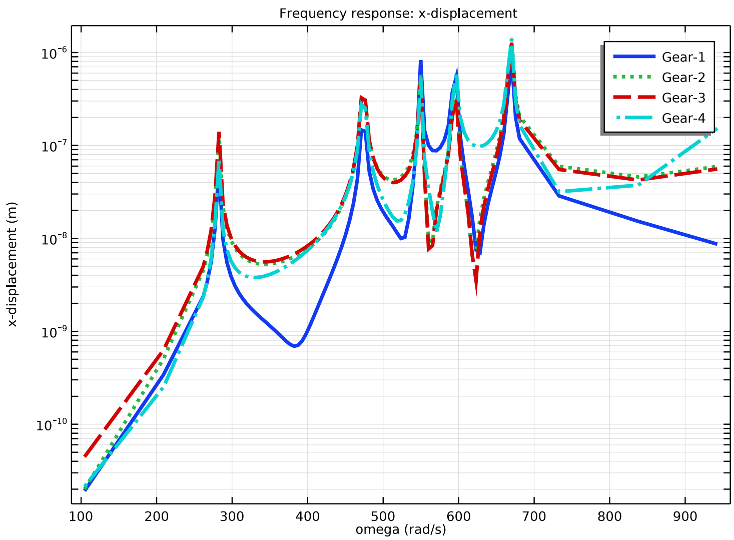

Figure 4: Frequency response of the x-direction displacement of all the gears.

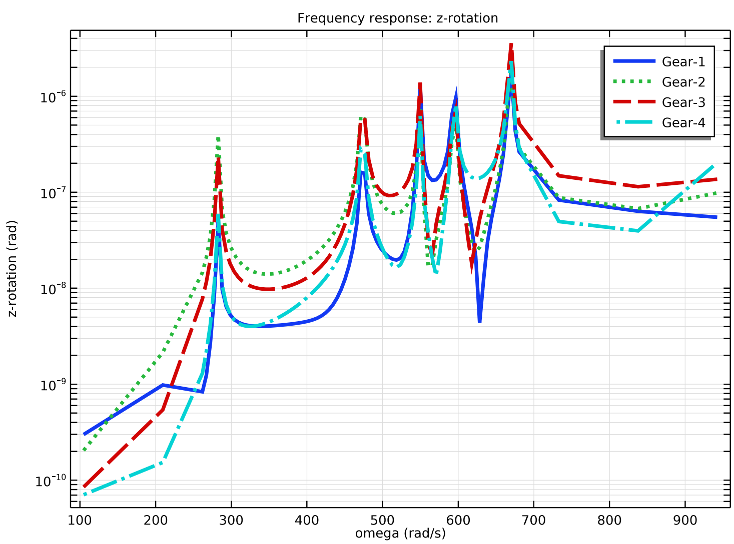

Figure 5: Frequency response of the z-direction rotation of all the gears.

Figure 6 shows the von Mises stress distribution in the shafts. It can be observed that the twisting of different shafts start from the end of the driven shaft, where the load is applied, to the opposite end of the driver shaft, where the axial rotation is constrained. The remaining parts of the shafts experience mostly the bending load generated due to gearing action.

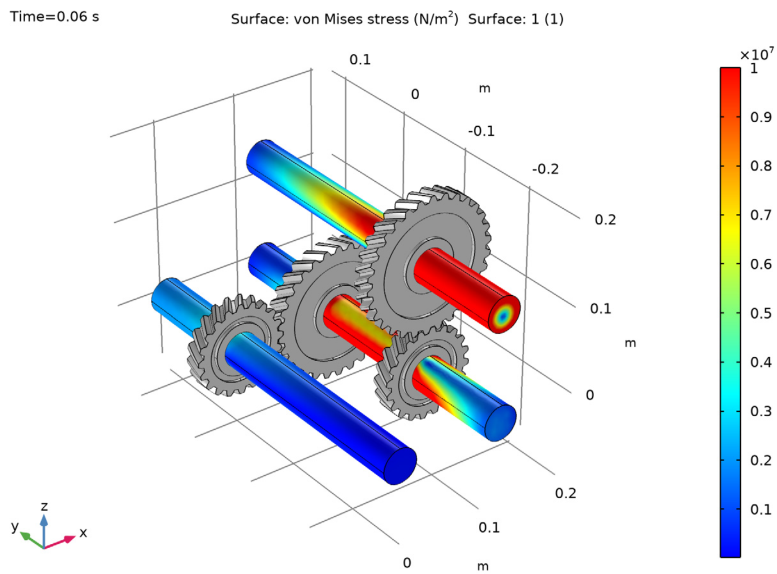

Figure 6: von Mises stress distribution in the shafts at a particular instance.

Figure 7 and Figure 8 show the orbits of the gear centers in the rotating and the fixed frame of reference, respectively. It can be observed that in the rotating frame of reference, all the gears are settled in different directions except gear-2 and gear-3, as they are both mounted on the intermediate shaft.

The gear mesh is assumed to be elastic, which causes an error while transferring rotation from one shaft to another. This error, known as the dynamic transmission error (DTE), is shown in Figure 9.

Finally, Figure 10 shows the time variation of the bearing loads on one of the journal bearings.

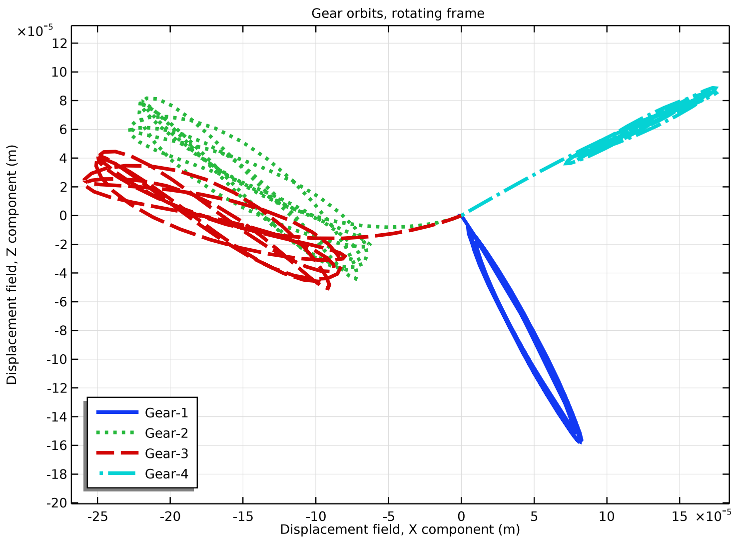

Figure 7: Orbits of gear center in the rotating frame of reference.

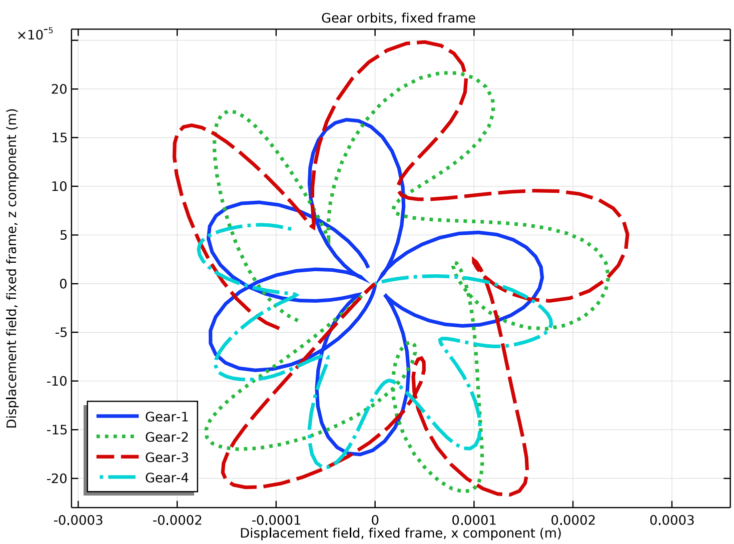

Figure 8: Orbits of gear center in the fixed frame of reference.

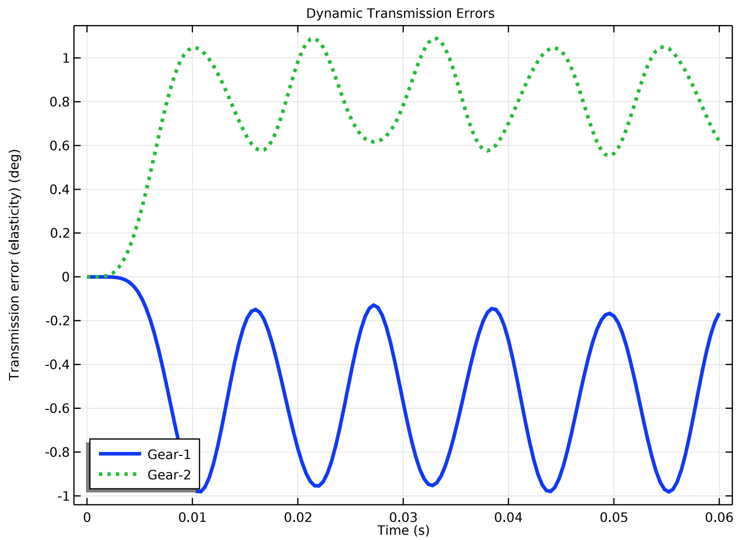

Figure 9: Time variation of the dynamic transmission error (DTE) of both gear pairs.

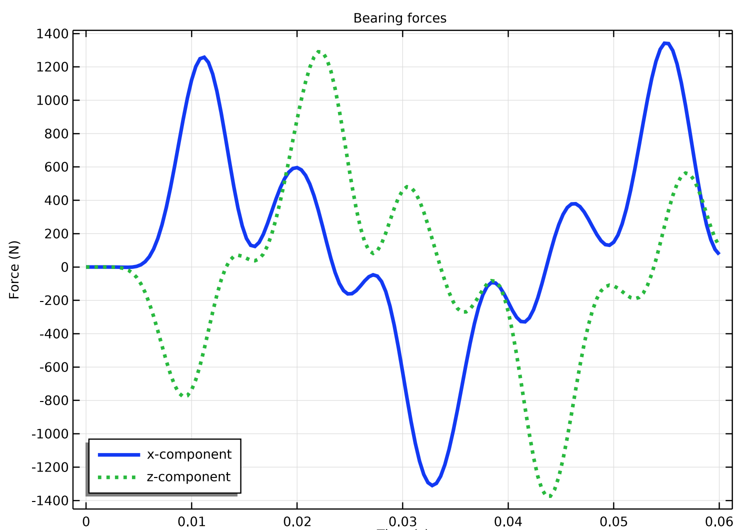

Figure 10: Time variation of the bearing forces.

Notes About the COMSOL Implementation

The **Solid Rotor** physics interface is used to model multiple rotors connected through gears.

To model multiple rotors, use **Rotor Axis** and **Change Rotor Speed** features to define the axis of rotation and speed of each rotor. In cases where two shafts are connected through gears, the shaft speeds are related through the gear ratio.

The gear mesh stiffness is assumed constant. However, in general it is a function of gear rotation. The varying gear mesh stiffness causes additional sustained vibration in the system.

It is possible to visualize results in both the rotating as well as the fixed frame of reference. In this model, the fixed frame of reference is used when animating the results.

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

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 (rotsld)**.

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 Click **Load from File**.

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

DEFINITIONS

Step 1 (step1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Functions>Step**.

3 In the **Settings** window for **Step**, locate the **Parameters** section.

4 In the **Location** text field, type `t_end/12`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `t_end/6`.

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 `geared_rotors.mphbin`.

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 In the **Home** toolbar, click **Build All**.

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 ROTOR (ROTSLD)

This model has multiple rotors having different axis of rotation as well as different speeds. So first set the axis of rotation and speed for all three rotors.

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 `omega`.

First Support 1

1 In the **Model Builder** window, expand the **Rotor Axis 1** node, then click **First Support 1**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `97-98` in the **Selection** text field.

5 Click **OK**.

Second Support 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `119-120` in the **Selection** text field.

5 Click **OK**.

Rotor Axis 2

1 In the **Physics** toolbar, click **Domains** and choose **Rotor Axis**.

2 Select Domains 5–7 only.

First Support 1

1 In the **Model Builder** window, expand the **Rotor Axis 2** node, then click **First Support 1**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `744, 756` in the **Selection** text field.

5 Click **OK**.

Second Support 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `749, 761` in the **Selection** text field.

5 Click **OK**.

Rotor Axis 3

1 In the **Physics** toolbar, click **Domains** and choose **Rotor Axis**.

2 Select Domains 3 and 4 only.

First Support 1

1 In the **Model Builder** window, expand the **Rotor Axis 3** node, then click **First Support 1**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `355, 363` in the **Selection** text field.

5 Click **OK**.

Second Support 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `358, 366` in the **Selection** text field.

5 Click **OK**.

Change Rotor Speed 1

1 In the **Physics** toolbar, click **Domains** and choose **Change Rotor Speed**.

2 Select Domains 5–7 only.

3 In the **Settings** window for **Change Rotor Speed**, locate the **Rotor Speed** section.

4 In the text field, type `-omega/gr`.

Change Rotor Speed 2

1 In the **Physics** toolbar, click **Domains** and choose **Change Rotor Speed**.

2 Select Domains 3 and 4 only.

3 In the **Settings** window for **Change Rotor Speed**, locate the **Rotor Speed** section.

4 In the text field, type `omega/gr^2`.

Fixed Axial Rotation 1

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

2 Select Boundary 57 only.

Now define helical gears to connect the three rotors.

Helical Gear 1

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

2 Select Domain 1 only.

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

4 In the n text field, type `n1`.

5 In the d_p text field, type `d1`.

6 In the α text field, type `alpha`.

7 In the β text field, type `beta`.

8 Locate the **Center of Rotation** section. From the list, choose **Centroid of selected entities**.

9 From the **Entity level** list, choose **Point**.

Center of Rotation: Point 1

1 In the **Model Builder** window, expand the **Helical Gear 1** node, then click **Center of Rotation: Point 1**.

2 In the **Settings** window for **Center of Rotation: Point**, locate the **Point Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `101, 115` in the **Selection** text field.

5 Click **OK**.

Helical Gear 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor (rotsld)** right-click **Helical Gear 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Domain 5 only.

5 Locate the **Gear Properties** section. In the n text field, type `n2`.

6 In the d_p text field, type `d2`.

7 In the β text field, type `-beta`.

Center of Rotation: Point 1

1 In the **Model Builder** window, expand the **Helical Gear 2** node, then click **Center of Rotation: Point 1**.

2 In the **Settings** window for **Center of Rotation: Point**, locate the **Point Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `747, 760` in the **Selection** text field.

6 Click **OK**.

Helical Gear 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor (rotsld)** right-click **Helical Gear 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Domain 6 only.

Center of Rotation: Point 1

1 In the **Model Builder** window, expand the **Helical Gear 3** node, then click **Center of Rotation: Point 1**.

2 In the **Settings** window for **Center of Rotation: Point**, locate the **Point Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `745, 758` in the **Selection** text field.

6 Click **OK**.

Helical Gear 4

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor (rotsld)** right-click **Helical Gear 2** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Domain 3 only.

Center of Rotation: Point 1

1 In the **Model Builder** window, expand the **Helical Gear 4** node, then click **Center of Rotation: Point 1**.

2 In the **Settings** window for **Center of Rotation: Point**, locate the **Point Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `357, 364` in the **Selection** text field.

6 Click **OK**.

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 **Helical Gear 1**.

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

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

Gear Elasticity 1

1 In the **Model Builder** window, expand the **Gear Pair 1** node, then click **Gear Elasticity 1**.

2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.

3 From the **Specify** list, choose **Total stiffness of gear pair**.

4 In the k_g text field, type `kg`.

Gear Pair 2

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 **Helical Gear 3**.

4 From the **Pinion** list, choose **Helical Gear 4**.

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

Gear Elasticity 1

1 In the **Model Builder** window, expand the **Gear Pair 2** node, then click **Gear Elasticity 1**.

2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.

3 From the **Specify** list, choose **Total stiffness of gear pair**.

4 In the k_g text field, type `kg`.

Now define journal bearings to provide the support for the rotors on both ends.

Journal Bearing 1

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

2 Select Boundary 52 only.

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

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

1	x
0	y
0	z

5 Locate the **Bearing Properties** section. From the **Bearing model** list, choose **Total spring and damping constant**.

6 In the $\mathbf{k}_u$ table, enter the following settings:

kb	0
0	kb

Journal Bearing 2

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

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

3 Click **Clear Selection**.

4 Select Boundary 57 only.

5 Locate the **Bearing Properties** section. In the $\mathbf{k}_u$ table, enter the following settings:

kb	0
0	kb

6 Clear the **Constrain axial motion** check box.

Journal Bearing 3

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

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

3 Click **Clear Selection**.

4 Select Boundary 397 only.

5 Locate the **Bearing Properties** section. In the $\mathbf{k}_u$ table, enter the following settings:

kb	0
0	kb

6 Select the **Constrain axial motion** check box.

Journal Bearing 4

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

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

3 Click **Clear Selection**.

4 Select Boundary 406 only.

5 Locate the **Bearing Properties** section. In the $\mathbf{k}_u$ table, enter the following settings:

kb	0
0	kb

6 Clear the **Constrain axial motion** check box.

Journal Bearing 5

1 Right-click **Journal Bearing 4** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 191 only.

5 Locate the **Bearing Properties** section. In the $\mathbf{k}_u$ table, enter the following settings:

kb	0
0	kb

Journal Bearing 6

1 Right-click **Journal Bearing 5** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 196 only.

5 Locate the **Bearing Properties** section. In the $\mathbf{k}_u$ table, enter the following settings:

kb	0
0	kb

6 Select the **Constrain axial motion** check box.

Applied Torque 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Applied Torque**.

2 Select Boundary 191 only.

3 In the **Settings** window for **Applied Torque**, locate the **Torque** section.

4 In the T_{ax} text field, type `Text*step1(t[1/s])`.

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 **Coarse**.

4 Click **Build All**.

STUDY 1: EIGENFREQUENCY

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

2 In the **Settings** window for **Study**, type `Study 1: Eigenfrequency` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega (Angular speed of driver shaft)	`range(0,500,9000)`	`rpm`

Step 1: Eigenfrequency

1 In the **Model Builder** window, click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `10`.

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

RESULTS

Mode Shape (rotsld)

Use the following instructions to plot the mode shape and Campbell diagram as shown in Figure 2 and Figure 3 respectively.

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

2 From the **Eigenfrequency (Hz)** list, choose **113.38**.

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

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

Backward Whirl Mode

Default **Campbell Plot, Rotating Frame** segregates the whirl modes, namely, forward and backward. Because of the multiple rotors rotating at different speeds whirl modes in all the rotors will not be identical. Therefore, automatic identification of the whirl mode is not correct in this case. Delete the **Backward Whirl Mode** node in this plot group.

1 In the **Model Builder** window, expand the **Campbell Plot, Rotating Frame (rotsld)** node.

2 Right-click **Backward Whirl Mode** and choose **Delete**.

Campbell Plot, Rotating Frame (rotsld)

1 In the **Model Builder** window, click **Campbell Plot, Rotating Frame (rotsld)**.

2 In the **Campbell Plot, Rotating Frame (rotsld)** toolbar, click **Plot**.

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

Campbell Plot, Fixed Frame (rotsld)

Delete the default **Campbell Plot, Fixed Frame** because the rotating frame to fixed frame transformation in the present case is not unique.

1 In the **Model Builder** window, right-click **Campbell Plot, Fixed Frame (rotsld)** and choose **Delete**.

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 **General Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

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

STUDY 2: FREQUENCY DOMAIN

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2: Frequency Domain` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega (Angular speed of driver shaft)	`range(1000,1000,2000) range(2500,50,6500) range(7000,1000,9000)`	`rpm`

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `2*omega`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Rotor (rotsld)>Applied Torque 1**.

6 Click **Disable**.

Set the **Null-space function** to **Orthonormal** for better constraint handling.

Solution 22 (sol22)

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

2 In the **Model Builder** window, expand the **Solution 22 (sol22)** node.

3 In the **Model Builder** window, expand the **Study 2: Frequency Domain>Solver Configurations>Solution 22 (sol22)>Stationary Solver 1** node, then click **Advanced**.

4 In the **Settings** window for **Advanced**, locate the **General** section.

5 From the **Null-space function** list, choose **Orthonormal**.

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

RESULTS

Use the following instructions to plot the frequency response curves as shown in Figure 4 and Figure 5 respectively.

1D Plot Group 4

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

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

3 From the **Dataset** list, choose **Study 2: Frequency Domain/Parametric Solutions 2 (sol23)**.

Global 1

1 Right-click **1D Plot Group 4** 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
abs(rotsld.hlg1.u)	m	
abs(rotsld.hlg2.u)	m	
abs(rotsld.hlg3.u)	m	
abs(rotsld.hlg4.u)	m	

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `2*pi[rad]*omega`.

7 Select the **Description** check box.

8 In the associated text field, type `omega`.

9 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

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

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

12 In the table, enter the following settings:

Legends
Gear-1
Gear-2
Gear-3
Gear-4

Frequency response: x-displacement

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Frequency response: x-displacement` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

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

5 In the associated text field, type `x-displacement (m)`.

6 Locate the **Axis** section. Select the **y-axis log scale** check box.

7 In the **Frequency response: x-displacement** toolbar, click **Plot**.

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

Frequency response: x-displacement 1

Right-click **Frequency response: x-displacement** and choose **Duplicate**.

Global 1

1 In the **Model Builder** window, expand the **Frequency response: x-displacement 1** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(rotsld.hlg1.thz)`	`rad`	
`abs(rotsld.hlg2.thz)`	`rad`	
`abs(rotsld.hlg3.thz)`	`rad`	
`abs(rotsld.hlg4.thz)`	`rad`	

Frequency response: z-rotation

1 In the **Model Builder** window, under **Results** click **Frequency response: x-displacement 1**.

2 In the **Settings** window for **1D Plot Group**, type `Frequency response: z-rotation` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `z-rotation (rad)`.

4 In the **Frequency response: z-rotation** toolbar, click **Plot**.

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

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 **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

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

STUDY 3: TIME DEPENDENT

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3: Time Dependent` in the **Label** text field.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3: Time Dependent** 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_end/150,t_end)`.

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

RESULTS

Study 3: Time Dependent/Solution 110 (sol110)

Use the following instructions to plot the von-Mises stress and gear orbits as shown in Figure 6, Figure 7, and Figure 8 respectively.

Study 3: Time Dependent/Solution 110 (6) (sol110)

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

2 Right-click **Results>Datasets>Study 3: Time Dependent/Solution 110 (sol110)** 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, 3, 5, and 6 only.

Surface

In the **Model Builder** window, expand the **Results>Stress (rotsld)** node.

Deformation

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

2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Rotor>Displacement>rotsld.us_fix,...,rotsld.ws_fix - Displacement field, fixed frame**.

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

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

Surface 2

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

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

3 From the **Dataset** list, choose **Study 3: Time Dependent/Solution 110 (6) (sol110)**.

4 Locate the **Expression** section. In the **Expression** text field, type 1.

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

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

Surface

1 In the **Model Builder** window, 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 **Maximum** text field, type 1e7.

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

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

Cut Point 3D 1

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

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

3 From the **Dataset** list, choose **Study 3: Time Dependent/Solution 110 (5) (sol110)**.

4 Locate the **Point Data** section. In the **X** text field, type rotsld.hlg1.xcx, rotsld.hlg2.xcx, rotsld.hlg3.xcx, rotsld.hlg4.xcx.

5 In the **Y** text field, type rotsld.hlg1.xcy, rotsld.hlg2.xcy, rotsld.hlg3.xcy, rotsld.hlg4.xcy.

6 In the **Z** text field, type rotsld.hlg1.xcz, rotsld.hlg2.xcz, rotsld.hlg3.xcz, rotsld.hlg4.xcz.

1D Plot Group 9

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

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

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

Point Graph 1

1 Right-click **1D Plot Group 9** 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 u.

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

7 In the **Width** text field, type 3.

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

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

10 In the table, enter the following settings:

Legends
Gear-1
Gear-2
Gear-3
Gear-4

Gear orbits, rotating frame

1 In the **Model Builder** window, under **Results** click **1D Plot Group 9**.

2 In the **Settings** window for **1D Plot Group**, type Gear orbits, rotating frame in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Axis** section. Select the **Preserve aspect ratio** check box.

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

6 In the **Gear orbits, rotating frame** toolbar, click **Plot**.

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

Gear orbits, rotating frame 1

Right-click **Gear orbits, rotating frame** and choose **Duplicate**.

Point Graph 1

1 In the **Model Builder** window, expand the **Gear orbits, rotating frame 1** node, then click **Point Graph 1**.

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

3 In the **Expression** text field, type rotsld.ws_fix.

4 Locate the **x-Axis Data** section. In the **Expression** text field, type rotsld.us_fix.

Gear orbits, fixed frame

1 In the **Model Builder** window, under **Results** click **Gear orbits, rotating frame 1**.

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

3 In the **Gear orbits, fixed frame** toolbar, click **Plot**.

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

1D Plot Group 11

Use the following instructions to plot the gear transmission errors and bearing forces as shown in Figure 9 and Figure 10 respectively.

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

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

3 From the **Dataset** list, choose **Study 3: Time Dependent/Solution 110 (5) (sol110)**.

Global 1

1 Right-click **1D Plot Group 11** 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
`rotsld.grp1.th_el`	`deg`	`Transmission error (elasticity)`
`rotsld.grp2.th_el`	`deg`	`Transmission error (elasticity)`

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

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

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

7 In the table, enter the following settings:

Legends
`Gear-1`
`Gear-2`

Gear DTEs

1 In the **Model Builder** window, under **Results** click **1D Plot Group 11**.

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

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

4 In the **Title** text area, type `Dynamic Transmission Errors`.

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

6 In the **Gear DTEs** toolbar, click **Plot**.

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

Cut Point 3D 2

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

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

3 From the **Dataset** list, choose **Study 3: Time Dependent/Solution 110 (5) (sol110)**.

4 Locate the **Point Data** section. In the **X** text field, type `0`.

5 In the **Y** text field, type `1.25*d1`.

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

7 Select the **Snap to closest boundary** check box.

1D Plot Group 12

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

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

3 From the **Dataset** list, choose **Cut Point 3D 2**.

Point Graph 1

1 Right-click **1D Plot Group 12** 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 `rotsld.jrb2.f2`.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

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

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

7 Select the **Show legends** check box.

8 In the table, enter the following settings:

Legends
x-component

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 `rotsld.jrb2.f3`.

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

Legends
z-component

Bearing forces

1 In the **Model Builder** window, under **Results** click **1D Plot Group 12**.

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

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

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

5 In the associated text field, type `Force (N)`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

7 In the **Bearing forces** toolbar, click **Plot**.

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

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 (rotsld)**.

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