Created in COMSOL Multiphysics 6.0

Comparison of Campbell Plots Using Different Rotor Interfaces

Introduction

When modeling rotor systems, different physics interfaces can be used depending on the required level of detail and the type of system. The modeling steps as well as the result interpretations may differ for the different interfaces. In this example, an eigenfrequency analysis of a rotor system is performed using different physics interfaces available for rotor modeling in COMSOL Multiphysics; namely the *Solid Rotor* interface, the *Solid Rotor, Fixed Frame* interface, and the *Beam Rotor* interface. The analyzed rotor consists of a stepped shaft supported by bearings at three different locations. The Campbell diagrams as computed with the different interfaces are compared to each other. The model also helps in understanding the modeling steps involved in each interface.

Model Definition

The model consists of a stepped rotor supported on three bearings. The overall rotor configuration is shown in Figure 1.

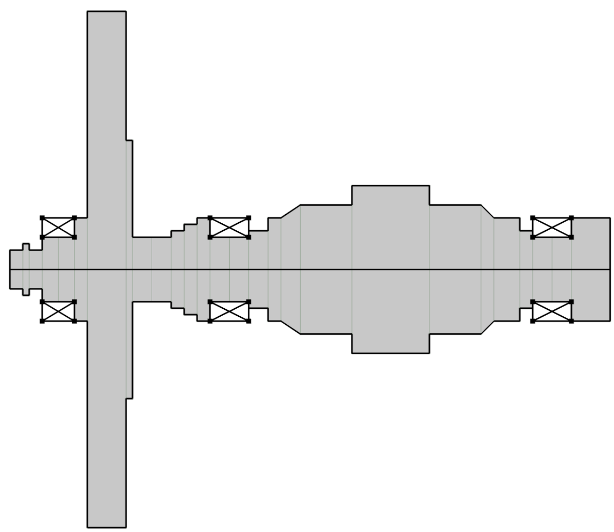

Figure 1: Rotor geometry.

The locations of different rotor stations along the axial direction are given in Table 1.

TABLE 1: LOCATIONS OF STATIONS ON THE ROTOR

STATION NUMBER	AXIAL COORDINATE
1	0 m
2	0.01 m
3	0.015 m
4	0.025 m
5	0.0375 m
6	0.05 m
7	0.06 m
8	0.09 m
9	0.095 m
10	0.11 m
11	0.125 m
12	0.135 m
13	0.145 m
14	0.155 m
15	0.17 m
16	0.185 m
17	0.2 m
18	0.21 m
19	0.225 m
20	0.265 m
21	0.325 m
22	0.365 m
23	0.375 m
24	0.395 m
25	0.405 m
26	0.42 m
27	0.435 m
28	0.465 m

Diameters of the rotor between different stations are given in Table 2.

TABLE 2: ROTOR DIAMETERS

ROTOR SEGMENT	DIAMETER
Station 1 and 2	30 mm
Station 2 and 3	40 mm
Station 3 and 4	30 mm
Station 4 and 5	50 mm
Station 5 and 6	50 mm
Station 6 and 7	80 mm
Station 7 and 8	400 mm
Station 8 and 9	200 mm
Station 9 and 10	50 mm
Station 10 and 11	50 mm
Station 11 and 12	60 mm
Station 12 and 13	70 mm
Station 13 and 14	80 mm
Station 14 and 15	50 mm
Station 15 and 16	50 mm
Station 16 and 17	60 mm
Station 17 and 18	80 mm
Station 19 and 20	100 mm
Station 20 and 21	130 mm
Station 21 and 22	100 mm
Station 23 and 24	80 mm
Station 24 and 25	60 mm
Station 25 and 26	50 mm
Station 26 and 27	50 mm
Station 27 and 28	80 mm

The rotor is made of structural steel. It is tapered between stations 18 and 19 as well as between stations 22 and 23. Three identical bearings hold the rotor in place. The first bearing spans from stations 4 to 6, the second bearing spans from stations 14 to 16 and the third bearing spans from stations 25 to 27. All bearings have an isotropic stiffness of 1e8 N/m.

Stress Stiffening and Gyroscopic Moment

The rotor spin produces stresses even in the absence of external loads. Therefore, the effective stiffness of a stationary rotor differs from a spinning rotor due to the so-called stress stiffening effect. This difference in the stiffness is also referred to as geometric stiffness. When performing an eigenfrequency analysis by itself using the Solid Rotor interface, the geometric stiffness effect will be absent. To capture the geometric stiffness, a prestressed eigenfrequency analysis must be performed. This analysis consists of a study sequence including a stationary and an eigenfrequency analysis. The stationary study captures the stress state in the rotor due to it spinning. The stationary solution is then used as a linearization point in the eigenfrequency study to account for the stress stiffening effect. Note that the eigenfrequency study should use a geometrically nonlinear formulation, as otherwise, it would be equivalent to considering an unstressed state of the rotor as a linearization point and hence there is no stress stiffening effect. If you add the special study type **Eigenfrequency, Prestressed**, these settings are automatically taken care of. For a manually added study steps you need to change the settings manually.

In a Beam Rotor interface, only the rotor axis is explicitly considered as part of the rotor geometry. Therefore, the stress state due to rotor spin cannot be obtained in a beam rotor model. However, the gyroscopic moment in the beam rotor allows for an equivalent consideration of the geometric stiffness. Since gyroscopic moments are always present in the beam rotor interface, a prestressed analysis is not required.

Coordinate Frames and Result Interpretations

The Solid Rotor interface formulates the problem in a co-rotating frame. Thus, all results, including eigenfrequencies, should be interpreted with respected to an observer sitting in the co-rotating frame. As a result, the eigenfrequencies as observed from a space-fixed frame require some adjustment with respect to the corresponding eigenfrequencies in the co-rotating frame. The whirling mode frequencies in co-rotating frame should be shifted by the angular speed of the rotor in either direction depending on the relative direction of the whirl with respect to the spin direction of the rotor. If the whirl and spin directions are equal, then the rotor speed is added in the co-rotating frame frequency, otherwise the rotor speed is subtracted to get the effective frequency in a space-fixed frame. Axial and torsional vibration frequencies usually remain same in both co-rotating and space fixed frames, thus, do not require any adjustment. This transformation is done internally in the Solid Rotor interface, and corresponding variables in a space-fixed frame can be conveniently evaluated in post-processing. By default, the Solid Rotor interface generates two Campbell diagrams, one in a co-rotating frame and a second in a fixed frame. Critical speeds of the rotor can be obtained using both Campbell diagrams. However, different procedures are required. You can follow the standard procedure of looking at the intersection points of the ω=Ω curve with the eigenfrequency curves to determine the critical speed from a Campbell diagram in a fixed frame. For a Campbell diagram in a co-rotating frame, forward whirl critical speeds are the intersection of eigenfrequency curves with the x-axis (ω=0) and the backward whirl critical speeds are the intersection of eigenfrequency curves with the ω=2Ω curve.

The Beam Rotor and Solid Rotor, Fixed Frame interfaces formulate the problem in a space-fixed frame. The results from these interfaces must therefore be interpreted as observed from a space-fixed frame. Eigenfrequencies do not require any transformation in these interfaces. The default Campbell diagram from these interfaces should be compared with the space-fixed Campbell diagram in the Solid Rotor interface.

Results and Discussion

The sixth eigenmode of the rotor is shown in Figure 2 and a corresponding whirl plot is shown in Figure 3.

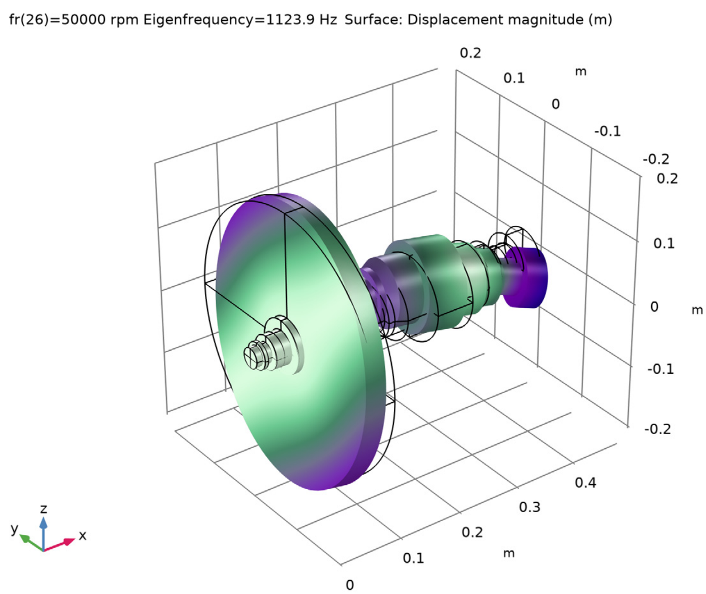

Figure 2: Sixth eigenmode computed using the Solid Rotor interface.

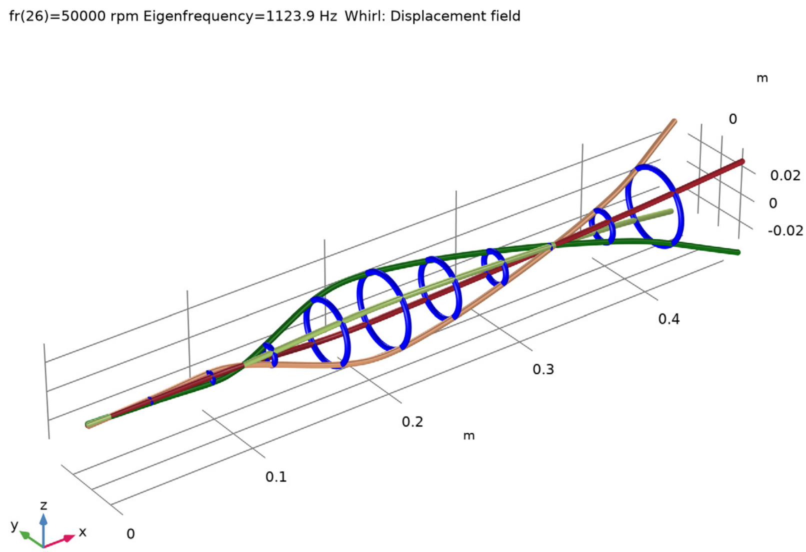

Figure 3: Whirl plot for the sixth mode computed using the Solid Rotor interface.

Figure 4 below shows the Campbell diagram obtained from the Solid Rotor interface in a space-fixed frame.

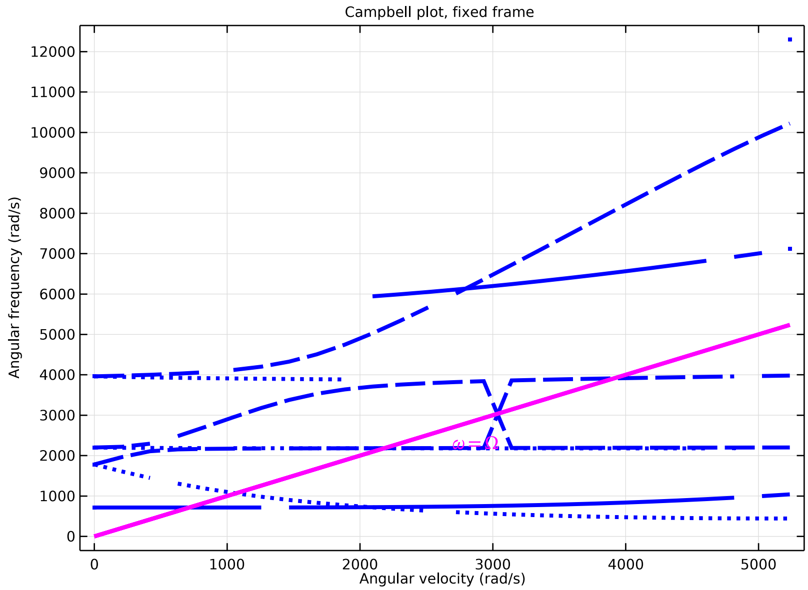

Figure 4: Campbell diagram computed using the Solid Rotor interface.

The eighth eigenmode in the Solid Rotor, Fixed Frame interface is shown in Figure 5.

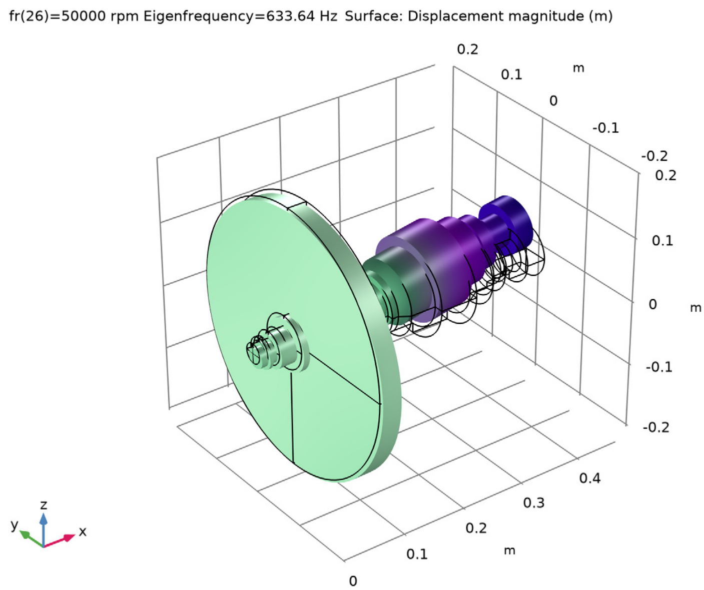

Figure 5: The eighth mode computed with the Solid Rotor, Fixed Frame interface.

The corresponding whirl plot is shown in Figure 6.

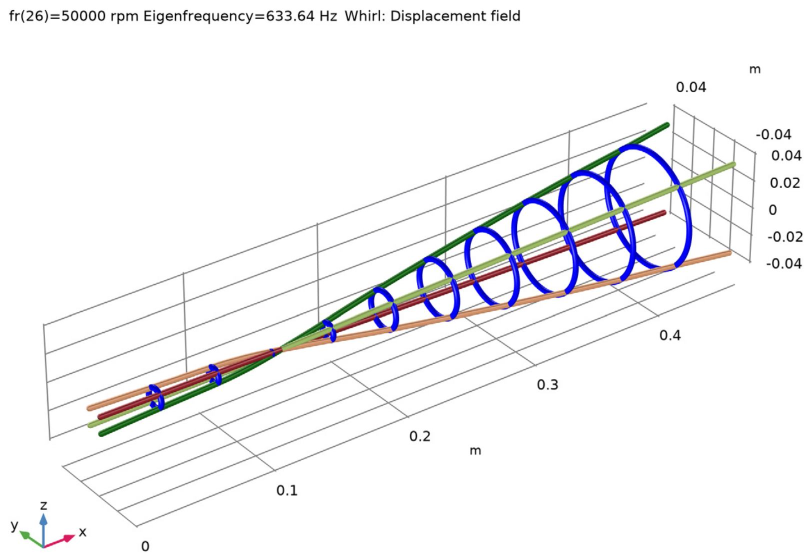

Figure 6: Whirl plot of the eighth mode computed with the Solid Rotor, Fixed Frame interface.

The Campbell plot for the same problem modeled using the Solid Rotor, Fixed Frame interface is shown in Figure 7. In this case, the low frequency curves correspond to the disk bending modes.

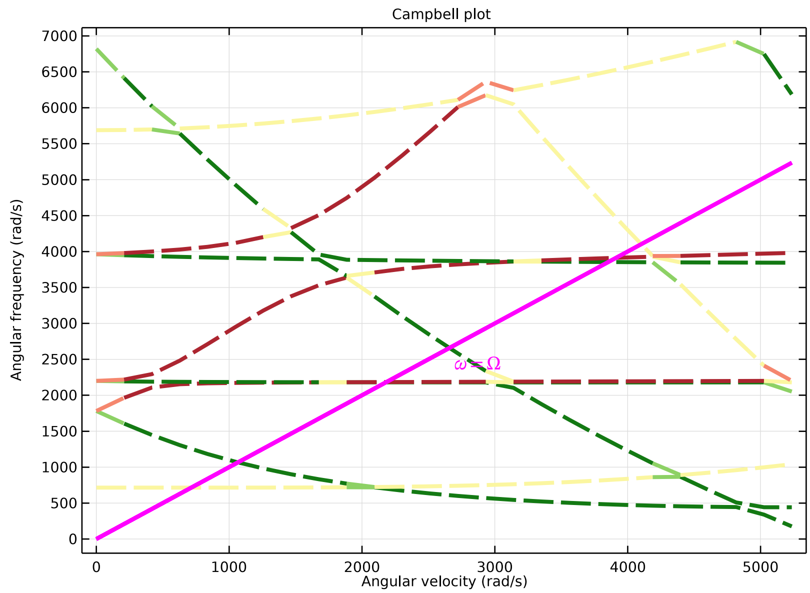

Figure 7: Campbell diagram computed with the Solid Rotor, Fixed Frame interface.

The whirl plot for the first mode computed with the Beam Rotor interface is shown in Figure 8.

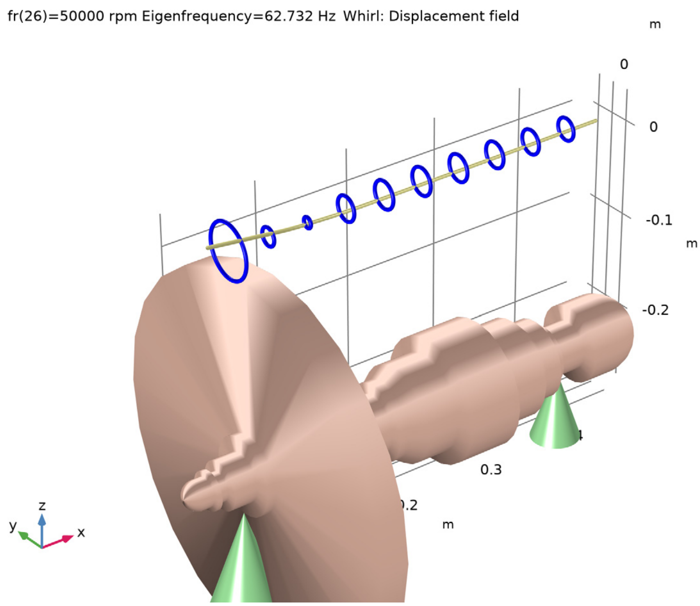

Figure 8: Whirl plot for the first mode computed with the Beam Rotor interface.

The Campbell diagram of the rotor using the Beam Rotor model is shown in Figure 9. The disk bending modes cannot be resolved with a beam rotor due to the kinematic assumptions in the beam formulation.

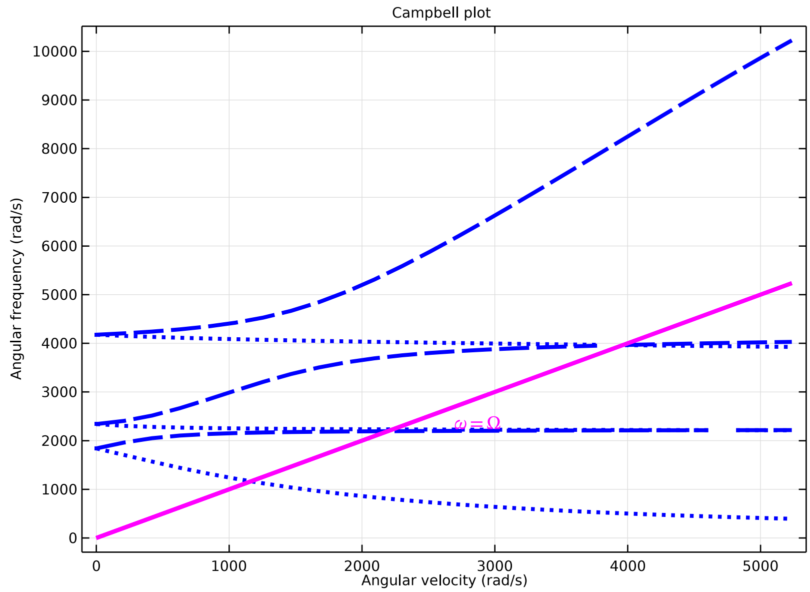

Figure 9: Campbell diagram computed with the Beam Rotor interface.

The Campbell diagrams computed with the different interfaces look similar. The eigenfrequencies from the beam rotor model may be slightly higher side due to the lower flexibility.

Application Library path: `Rotordynamics_Module/Tutorials/campbell_plot_comparison`

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**.

   You will compare the modeling of rotor using different interfaces. Add all the rotor interfaces in the physics.

2 In the **Select Physics** tree, select **Structural Mechanics>Rotordynamics>Solid Rotor (rotsld)**, **Structural Mechanics>Rotordynamics>Solid Rotor, Fixed Frame (srotf)**, and **Structural Mechanics>Rotordynamics>Beam Rotor (rotbm)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Eigenfrequency, Prestressed**.
6 Click **Done**.

Start by importing the parameters for modeling the rotor.

GLOBAL DEFINITIONS

Parameters: General

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 `campbell_plot_comparison_general.txt`.
5 In the **Label** text field, type `Parameters: General`.

Parameters: Stations

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.
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 `campbell_plot_comparison_stations.txt`.
5 In the **Label** text field, type `Parameters: Stations`.

Parameters: Shaft diameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.
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 `campbell_plot_comparison_diameters.txt`.

5 In the **Label** text field, type `Parameters: Shaft diameters`.

Now you will create the rotor geometry based on the imported parameters. Start by creating the 2D axisymmetric geometry of the rotor on a work plane.

GEOMETRY 1

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Polygon 1 (pol1)

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

You can choose to specify the coordinates for the polygons to create the 2D axisymmetric geometry yourself. To quickly create the geometry you can import these coordinates from a file.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 Click **Load from File**.

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

Work Plane 1 (wp1)

Revolve the 2D axisymmetric geometry to get the full rotor geometry.

Revolve 1 (rev1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Axis** section.

3 Find the **Direction of revolution axis** subsection. In the **xw** text field, type `1`.

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

5 Click **Build All Objects**.

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

Now create some selections of the rotor and bearings for later use.

DEFINITIONS

Beam Rotor

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

5 Select Edge 10 only.

6 Select the **Group by continuous tangent** check box.

7 In the **Label** text field, type `Beam Rotor`.

Journal Bearing 1

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 29 only.

5 Select the **Group by continuous tangent** check box.

6 In the **Label** text field, type `Journal Bearing 1`.

Journal Bearing 2

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 Click **Clear Selection**.

4 Select Boundaries 100, 101, 103, and 105–109 only.

5 In the **Label** text field, type `Journal Bearing 2`.

Journal Bearing 3

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 Click **Clear Selection**.

4 Select Boundaries 168, 169, 171, and 173–177 only.

5 In the **Label** text field, type `Journal Bearing 3`.

For a beam rotor, you can specify different diameters between different stations by using as many **Rotor Cross Section** nodes as rotor segments. To avoid multiple such nodes you can make use of interpolation function for the diameter as a function of the axial coordinate of the rotor. To create the steps in the rotor diameter, you can use a small tolerance near the stations.

Interpolation: rotor dia

1 In the **Definitions** toolbar, click **Interpolation**.
2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `campbell_plot_comparison_interpolation.txt`.
5 Click **Plot**.
6 In the **Label** text field, type `Interpolation: rotor dia`.
7 Locate the **Definition** section. In the **Function name** text field, type `dia`.
8 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
dia	m

9 In the **Argument** table, enter the following settings:

Argument	Unit
t	m

10 Click **Plot**.

GLOBAL DEFINITIONS

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in>Structural steel**.
3 Click **Add to Global Materials** in the window toolbar.
4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material Link: Solid

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, type `Material Link: Solid` in the **Label** text field.

Material Link: Beam

1 Right-click **Material Link: Solid** and choose **Duplicate**.

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

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

4 From the **Selection** list, choose **Beam Rotor**.

5 In the **Label** text field, type `Material Link: Beam`.

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 `fr`.

Rotor Axis 1

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

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

3 From the **Specified by** list, choose **Edge**.

Axis 1

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

2 In the **Settings** window for **Axis**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Beam Rotor**.

Fixed Axial Rotation 1

1 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

2 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor (rotsld)** click **Fixed Axial Rotation 1**.

3 Select Boundaries 186–189 only.

Journal Bearing 1

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

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

3 From the **Selection** list, choose **Journal Bearing 1**.

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

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

kb	0
0	kb

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

0	0
0	0

Now duplicate the current bearing node to create other bearings.

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 From the **Selection** list, choose **Journal Bearing 2**.

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

0	0
0	0

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 From the **Selection** list, choose **Journal Bearing 3**.

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

0	0
0	0

SOLID ROTOR, FIXED FRAME (SROTF)

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

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

3 In the text field, type `fr`.

Rotor Axis 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor, Fixed Frame (srotf)** click **Rotor Axis 1**.

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

3 From the **Specified by** list, choose **Edge**.

Axis 1

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

2 In the **Settings** window for **Axis**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Beam Rotor**.

Fixed Axial Rotation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor, Fixed Frame (srotf)** click **Fixed Axial Rotation 1**.

2 Select Boundaries 186–189 only.

Copy the bearing nodes from the **Solid Rotor** to **Solid Rotor, Fixed Frame** interface to create similar bearing features.

SOLID ROTOR (ROTSLD)

Journal Bearing 1, Journal Bearing 2, Journal Bearing 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor (rotsld)**, Ctrl-click to select **Journal Bearing 1**, **Journal Bearing 2**, and **Journal Bearing 3**.

2 Right-click and choose **Copy**.

SOLID ROTOR, FIXED FRAME (SROTF)

Journal Bearing 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Rotor, Fixed Frame (srotf)** and choose **Paste Multiple Items**.

Journal Bearing 1, Journal Bearing 2, Journal Bearing 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Rotor, Fixed Frame (srotf)**, Ctrl-click to select **Journal Bearing 1**, **Journal Bearing 2**, and **Journal Bearing 3**.

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

3 In the $\mathbf{k}_\theta$ table, enter the following settings:

0	0
0	0

Journal Bearing 2

1. In the **Model Builder** window, click **Journal Bearing 2**.
2. In the **Settings** window for **Journal Bearing**, locate the **Bearing Properties** section.
3. In the $\mathbf{k}_\theta$ table, enter the following settings:

0	0
0	0

Journal Bearing 3

1. In the **Model Builder** window, click **Journal Bearing 3**.
2. In the **Settings** window for **Journal Bearing**, locate the **Bearing Properties** section.
3. In the $\mathbf{k}_\theta$ table, enter the following settings:

0	0
0	0

BEAM ROTOR (ROTBM)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Beam Rotor (rotbm)**.
2. In the **Settings** window for **Beam Rotor**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Beam Rotor**.
4. Locate the **Rotor Speed** section. In the text field, type `fr`.

Rotor Cross Section 1

Use the interpolation function with axial coordinate as an argument to specify the axially varying diameter of the rotor.

1. In the **Model Builder** window, under **Component 1 (comp1)>Beam Rotor (rotbm)** click **Rotor Cross Section 1**.
2. In the **Settings** window for **Rotor Cross Section**, locate the **Cross-Section Definition** section.
3. In the d_o text field, type `dia(x)`.

Journal Bearing 1

1. In the **Physics** toolbar, click **Points** and choose **Journal Bearing**.
2. Click the **Go to Default View** button in the **Graphics** toolbar.
3. Click the **Wireframe Rendering** button in the **Graphics** toolbar.
4. Select Point 42 only.
5. In the **Settings** window for **Journal Bearing**, locate the **Bearing Properties** section.

6 From the **Bearing model** list, choose **Total spring and damping constant**.

7 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 **Point Selection** section.

3 Click **Clear Selection**.

4 Select Point 117 only.

Journal Bearing 3

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

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

3 Click **Clear Selection**.

4 Select Point 196 only.

MESH: SOLID

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

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

Create a swept mesh for the solid geometry of the rotor. You can use this mesh for **Solid Rotor** and **Solid Rotor, Fixed Frame** interfaces.

Swept 1

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

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `30`.

4 Click **Build Selected**.

Now create another mesh along the axis of the rotor for the **Beam Rotor** interface.

MESH: BEAM

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** check boxes for **Solid Rotor (rotsld)** and **Solid Rotor, Fixed Frame (srotf)**.

4 In the **Label** text field, type `Mesh: Beam`.

Edge 1

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

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

3 From the **Selection** list, choose **Beam Rotor**.

4 Click **Build All**.

You will perform separate study for each interface to avoid cluster of modes from different physics into the same solution. Start with study for the **Solid Rotor** interface.

STUDY: SOLID ROTOR

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

2 In the **Settings** window for **Study**, type `Study: Solid Rotor` 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
fr (Angular speed of shaft)	range(0,2000,50000)	rpm

Disable all physics interfaces except **Solid Rotor** in the study to avoid the assembly of corresponding dofs from these interfaces.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Solid Rotor, Fixed Frame (Srotf)** and **Component 1 (Comp1)>Beam Rotor (Rotbm)**.

5 Click **Disable in Model**.

6 Clear the **Modify model configuration for study step** check box.

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: 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 7.

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

6 In the tree, select **Component 1 (Comp1)>Solid Rotor, Fixed Frame (Srotf)** and **Component 1 (Comp1)>Beam Rotor (Rotbm)**.

7 Click **Disable in Model**.

8 Clear the **Modify model configuration for study step** check box.

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

RESULTS

Mode Shape (rotsld)

The mode shape is a default plot. You can change the eigenfrequency values to look at the different modes. The sixth mode is shown in Figure 2.

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

2 From the **Eigenfrequency (Hz)** list, choose **1123.9**.

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

4 Click the **Go to Default View** button in the **Graphics** toolbar.

Whirl (rotsld)

Set the same eigenfrequency value in the default whirl plot to analyze the corresponding whirl shape of the rotor. This plot is shown in Figure 3.

1 In the **Model Builder** window, click **Whirl (rotsld)**.

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

3 From the **Eigenfrequency (Hz)** list, choose **1123.9**.

4 In the **Whirl (rotsld)** toolbar, click **Plot**.

Campbell Plot, Fixed Frame (rotsld)

The **Solid Rotor** interface generates two Campbell plots by default, one in the corotating frame and other in the fixed frame. The fixed frame version, as shown in Figure 4, can be compared with the corresponding Campbell plots from other interfaces.

Study for the **Solid Rotor** is complete. Add a couple of new studies for the **Solid Rotor, Fixed Frame** and **Beam Rotor** interfaces.

ROOT

In the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency, Prestressed**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Solid Rotor (rotsld)** and **Beam Rotor (rotbm)**.

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

5 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Eigenfrequency**.

Note that you do not need **Eigenfrequency, Prestressed** study for the **Beam Rotor**. The spinning causes stress stiffening in the rotor. In the **Solid Rotor** and **Solid Roor, Fixed Frame** interfaces, geometric stiffness due to stress stiffening is captured using a prestressed study. In the **Beam Rotor**, the gyroscopic moment captures the same effect automatically.

6 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Solid Rotor (rotsld)** and **Solid Rotor, Fixed Frame (srotf)**.

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

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

STUDY: SRFF

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

2 In the **Settings** window for **Study**, type `Study: SRFF` 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
fr (Angular speed of shaft)	`range(0,2000,50000)`	`rpm`

Step 1: Stationary

In this study you will disable all other interfaces except **Solid Rotor, Fixed Frame**.

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Solid Rotor (Rotsld)** and **Component 1 (Comp1)> Beam Rotor (Rotbm)**.

5 Click **Disable in Model**.

6 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh: Solid

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

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Solid Rotor (Rotsld)** and **Component 1 (Comp1)> Beam Rotor (Rotbm)**.

5 Click **Disable in Model**.

6 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh: Solid

7 Locate the **Study Settings** section. Select the **Desired number of eigenfrequencies** check box.

8 In the associated text field, type 9.

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

RESULTS

Mode Shape (srotf)

The mode shape is a default plot. Change the eigenfrequency values to analyze different modes. The eighth mode is shown in Figure 5.

1. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2. From the **Eigenfrequency (Hz)** list, choose **633.64**.
3. In the **Mode Shape (srotf)** toolbar, click **Plot**.

Whirl (srotf)

Change the eigenfrequency value in the default whirl plot to analyze the corresponding whirl shape shown in Figure 6.

1. In the **Model Builder** window, click **Whirl (srotf)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Eigenfrequency (Hz)** list, choose **633.64**.
4. In the **Whirl (srotf)** toolbar, click **Plot**.

Campbell Plot (srotf)

The Campbell plot from the **Solid Rotor, Fixed Frame** interface is shown in Figure 7.

STUDY: BEAM ROTOR

1. In the **Model Builder** window, click **Study 3**.
2. In the **Settings** window for **Study**, type `Study: Beam Rotor` 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
fr (Angular speed of shaft)	`range(0,2000,50000)`	`rpm`

Step 1: Eigenfrequency

Disable other interfaces except **Beam Rotor** interface like previous studies.

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

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Solid Rotor (Rotsld)** and **Component 1 (Comp1)>Solid Rotor, Fixed Frame (Srotf)**.

5 Click **Disable in Model**.

6 Clear the **Modify model configuration for study step** check box.

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

RESULTS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
scale	0.3	0.3	

Whirl (rotbm)

The default whirl plot is shown in Figure 8.

Campbell Plot (rotbm)

The default Campbell plot from the **Beam Rotor** interface is shown in Figure 9.