Model created in COMSOL Multiphysics 6.4

Comparison of Campbell Diagrams Using Different Rotor Interfaces

Introduction

When modeling rotor systems in COMSOL Multiphysics, 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 analysis is performed over a range of rotational speeds, and the resulting Campbell diagrams obtained with the different physics interfaces are compared.

Model Definition

In order to compare the results obtained with the three rotor interfaces, the geometry must fulfill any underlying assumptions of all the interfaces. An axisymmetric rotor with isotropic bearings fulfills these requirements.

A cross-section of the axisymmetric rotor is shown in Figure 1. The rotor has a varying diameter along the axial direction, and is supported by bearings at three distinct locations.

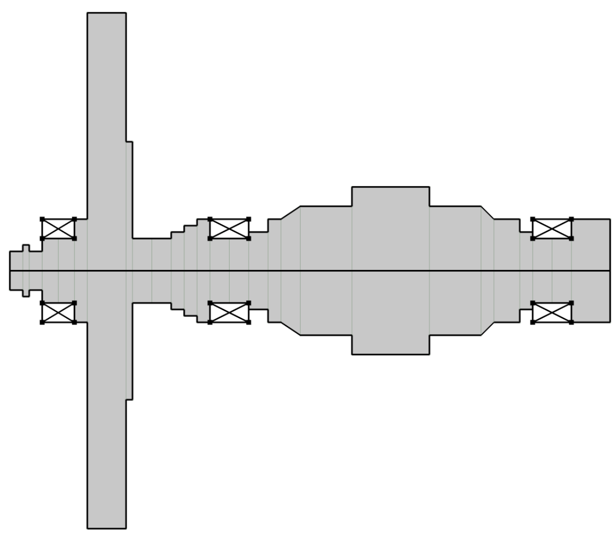

Figure 1: A cross-section of the rotor geometry.

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

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

TABLE 2: ROTOR DIAMETERS.

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

The rotor is made of structural steel. It has two tapered segments, the first between station 18 and 19, and the second between station 22 and 23.

Three bearings support the rotor. These are all isotropic with a direct stiffness of 20 MN/m. The first bearing is located between station 4 to 6, the second bearing between station 14 to 16, and the third bearing spans from stations 25 to 27.

Stress Stiffening and Gyroscopic Moment

A spinning rotor produces stresses even in the absence of external loads. Therefore, the effective stiffness of a stationary and a spinning rotor differs. This is due to the so-called *stress stiffening* effect. This difference in the stiffness is sometimes also referred to as *geometric stiffness*.

When performing an eigenfrequency analysis by itself using the Solid Rotor interface, the stress stiffening effect will be absent. To capture this effect, 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 the 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 results processing. The Solid Rotor interface generates the Campbell diagram in the co-rotating frame. For a Campbell diagram in a co-rotating frame, forward whirl critical speeds are the intersection of eigenfrequency curves with the x-axis ($0\times\Omega$) and the backward whirl critical speeds are the intersection of eigenfrequency curves with the $2\times\Omega$ 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.

Results and Discussion

Figure 2 shows the resulting Campbell diagram obtained with the Solid Rotor interface. As previously mentioned, the eigenfrequencies are obtained in a co-rotating frame of reference. The straight black line indicates the $2\times\Omega$ curve.

Intersections between this line, or the horizontal axis, and the eigenfrequencies are considered as *potential* critical speeds.

The color of the lines indicates another important property of the mode, namely the *directivity* of the mode. The red, green and purple color indicate whether the corresponding mode is a *forward*, *straight-line*, or *backward* mode relative to the co-rotating frame, respectively.

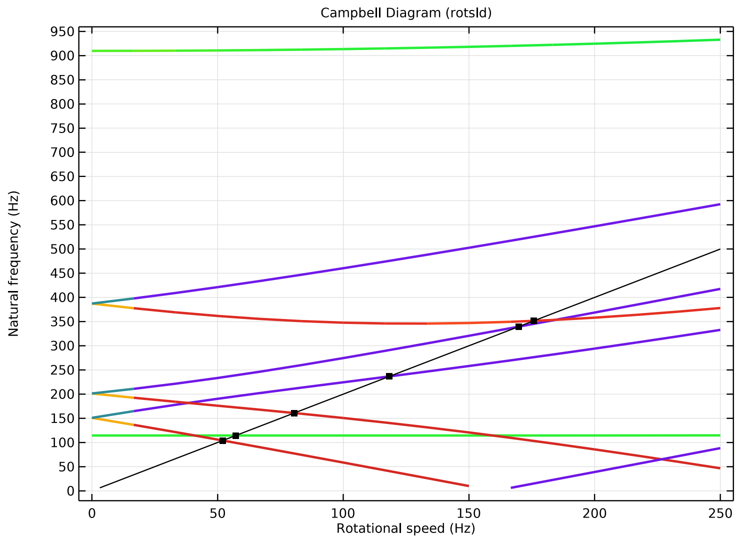

Figure 2: Campbell diagram computed using the Solid Rotor interface.

The fifth mode shape and the associated whirl are shown in Figure 3 and Figure 4, respectively.

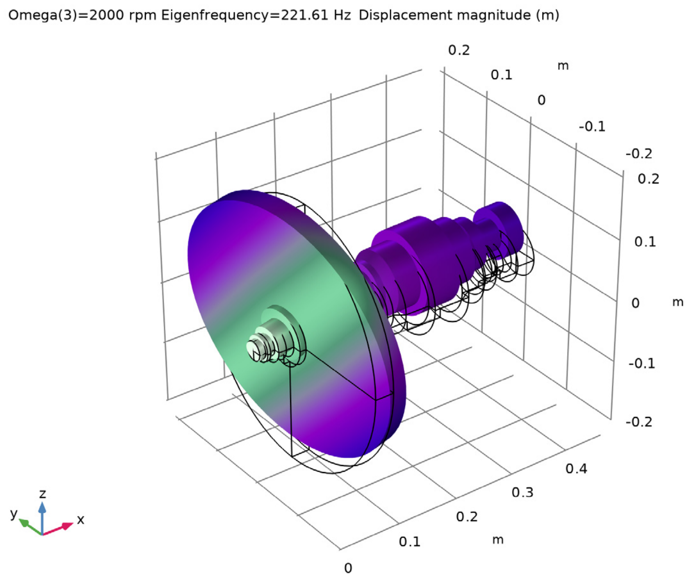

Figure 3: Fifth eigenmode computed using the Solid Rotor interface.

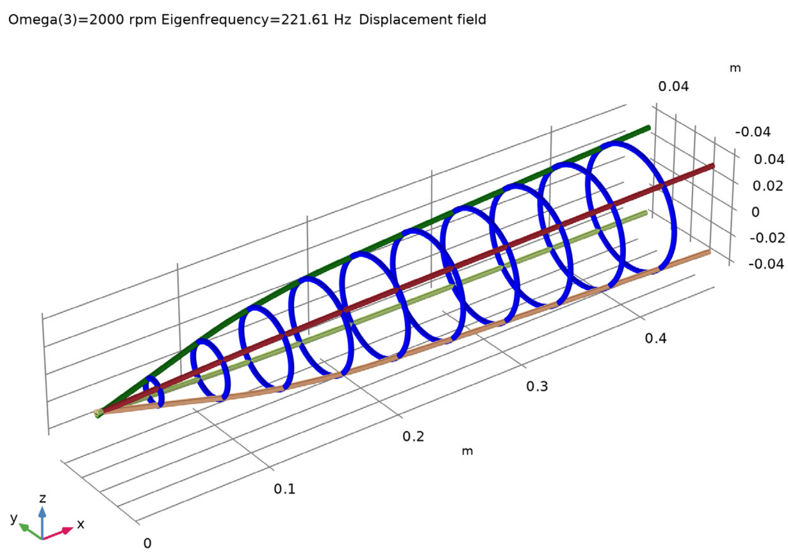

Figure 4: Whirl plot for the fifth eigenmode computed using the Solid Rotor interface.

The corresponding Campbell diagram obtained using the Solid Rotor, Fixed Frame interface is shown in Figure 5. The eigenfrequencies are here obtained relative to the space-fixed frame of reference.

The straight line indicates the rotational speed of the rotor (1×Ω). Intersections between this line and the eigenfrequencies are considered potential critical speeds.

The line color indicates again the directivity of the mode, but here relative to the fixed frame of reference.

The transformed eigenfrequencies obtained with the Solid Rotor interface are included as markers. A very good agreement is found between the eigenfrequencies obtained with the two interfaces.

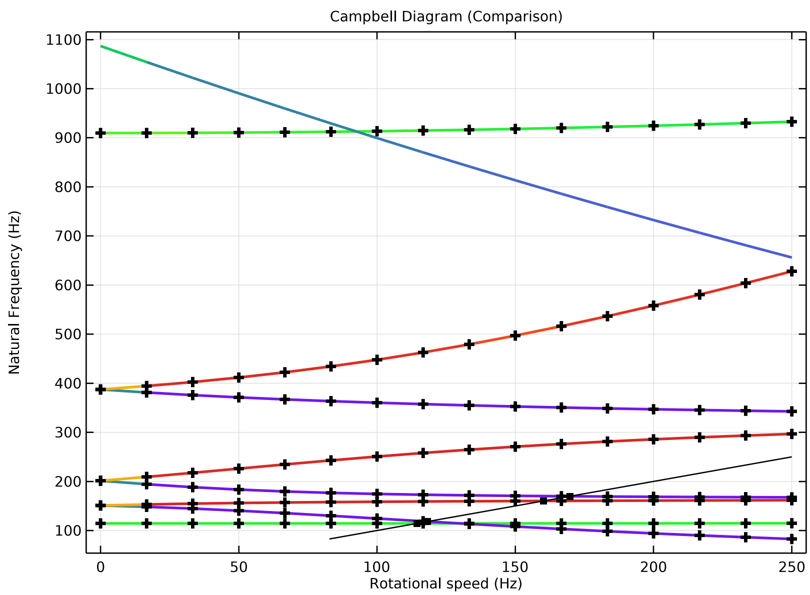

Figure 5: Campbell diagram obtained with the Solid Rotor, Fixed Frame interface compared with the transformed eigenfrequencies obtained with the Solid Rotor interface.

The sixth mode shape and the associated whirl are shown in Figure 6 and Figure 7.

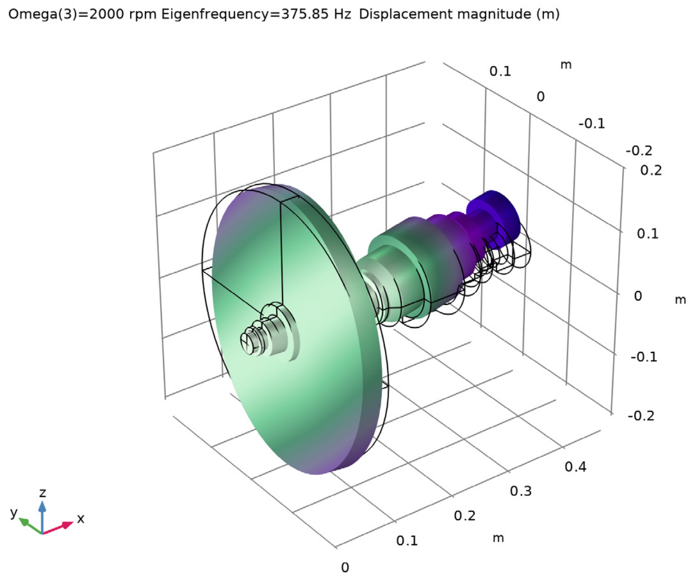

Figure 6: The sixth eigenmode computed with the Solid Rotor, Fixed Frame interface.

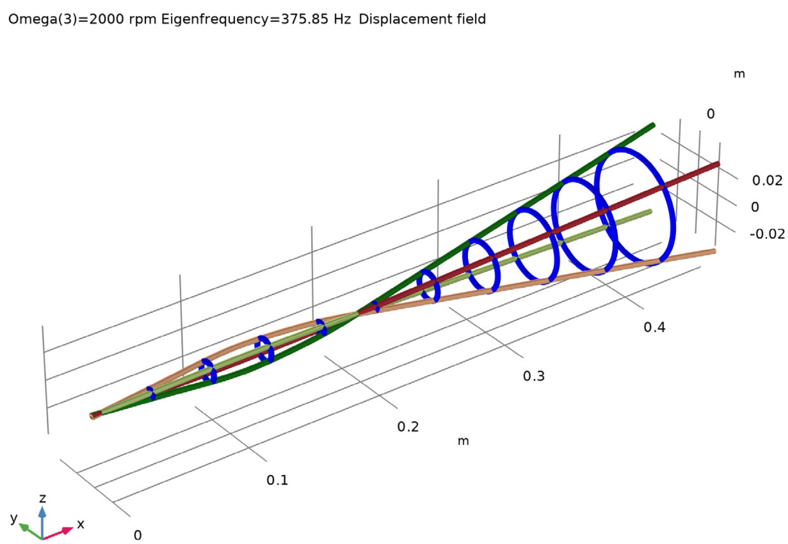

Figure 7: Whirl plot of the sixth eigenmode computed with the Solid Rotor, Fixed Frame interface.

The Campbell diagram obtained using the Beam Rotor interface is visualized in Figure 8. The disk bending modes cannot be resolved with a beam rotor due to the kinematic assumptions in the beam formulation.

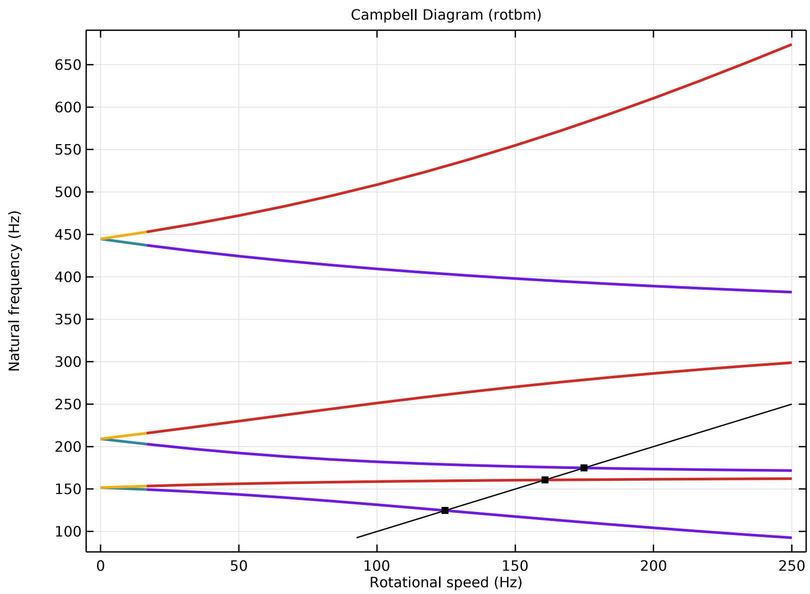

Figure 8: Campbell diagram computed with the Beam Rotor interface.

The whirl plot for the first mode computed with the Beam Rotor interface is shown in Figure 9.

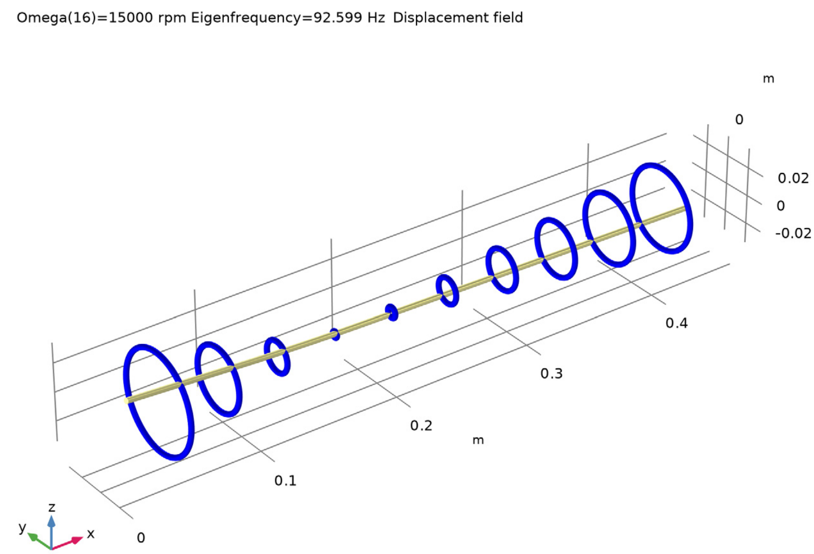

Figure 9: Whirl plot for the first mode computed with the Beam Rotor interface.

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** checkbox.

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** checkbox.

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 numerous definitions of that node, an interpolation function is used to describe the variations in diameter along the rotor axis. 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**.

ADD MATERIAL

1 In the **Materials** 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 the **Add to Global Materials** button in the window toolbar.

5 In the **Materials** 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)

Rotating Frame 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Rotor (rotsld)** click **Rotating Frame 1**.
2. In the **Settings** window for **Rotating Frame**, locate the **Axis of Rotation** section.
3. From the $\mathbf{e}_3$ list, choose **x-axis**.
4. Locate the **Rotational Velocity** section. In the Ω_r text field, type `Omega`.

Fixed Axial Rotation 1

1. In the **Model Builder** window, click **Fixed Axial Rotation 1**.
2. 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. Specify the $\mathbf{k}_u$ matrix as

kb	0
0	kb

6. Specify the $\mathbf{k}_\theta$ matrix as

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. Specify the $\mathbf{k}_\theta$ matrix as

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. Specify the $\mathbf{k}_\theta$ matrix as

0	0
0	0

SOLID ROTOR, FIXED FRAME (SROTF)

Rotating Frame 1

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

2 In the **Settings** window for **Rotating Frame**, locate the **Axis of Rotation** section.

3 From the $\mathbf{e}_3$ list, choose **x-axis**.

4 Locate the **Rotational Velocity** section. In the Ω_r text field, type `Omega`.

Fixed Axial Rotation 1

1 In the **Model Builder** window, 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)

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

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

2 Specify the $\mathbf{k}_\theta$ matrix as

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 Specify the $\mathbf{k}_\theta$ matrix as

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 Specify the $\mathbf{k}_\theta$ matrix as

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 `Omega`.

5 Click to expand the **Result Settings** section. Find the **General settings** subsection. Clear the **Include undeformed geometry in stress/whirl plot** checkbox.

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 35 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 Specify the $\mathbf{k}_u$ matrix as

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 `24`.

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

Edge 1

1 In the **Mesh** toolbar, click **More Generators** 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
Omega (Angular speed of shaft)	range(0,1000,15000)	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** checkbox.

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**.

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** checkbox. In the associated text field, type 8.

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

5 In the tree, select **Component 1 (comp1)** > **Solid Rotor, Fixed Frame (srotf), Controls spatial frame** and **Component 1 (comp1)** > **Beam Rotor (rotbm)**.

6 Click **Disable in Model**.

Activate mode following for continuous tracking of the eigenfrequencies over the swept interval.

7 Click to expand the **Filtering and Sorting** section. Find the **Sorting** subsection. Select the **Mode following** checkbox.

Next, increase the storage of extra eigenmodes.

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **Study: Solid Rotor** > **Solver Configurations** > **Solution 1 (sol1)** > **Eigenvalue Solver 1** node, then click **Mode Following 1**.

4 In the **Settings** window for **Mode Following**, locate the **Mode Following** section.

5 Select the **Follow extra modes** checkbox.

6 From the **Selection** list, choose **Number of expected extra modes**.

7 In the **Number of expected extra modes** text field, type 1.

8 In the **Threshold for detection of new modes** text field, type 1e-5.

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 3.

1. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2. From the **Parameter value (Omega (rpm))** list, choose **2000**.
3. From the **Eigenfrequency (Hz)** list, choose **221.61**.
4. In the **Mode Shape (rotsld)** toolbar, click **Plot**.
5. Click the **Go to Default View** button in the **Graphics** toolbar.

Now, add a Whirl plot to visualize the associated whirl shape.

RESULT TEMPLATES

1. In the **Home** toolbar, click **Windows** and choose **Result Templates**.
2. Go to the **Result Templates** window.
3. In the tree, select **Study: Solid Rotor/Parametric Solutions 1 (sol3)** > **Solid Rotor** > **Whirl (rotf1)**.
4. Click the **Add Result Template** button in the window toolbar.

RESULTS

Whirl (rotf1)

1. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2. From the **Parameter value (Omega (rpm))** list, choose **2000**.
3. From the **Eigenfrequency (Hz)** list, choose **221.61**.
4. In the **Whirl (rotf1)** toolbar, click **Plot**.

Now, add the predefined Campbell diagram for the **Solid Rotor** interface.

RESULT TEMPLATES

1. Go to the **Result Templates** window.
2. In the tree, select **Study: Solid Rotor/Parametric Solutions 1 (sol3)** > **Solid Rotor** > **Campbell Diagram (rotsld)**.
3. Click the **Add Result Template** button in the window toolbar.

RESULTS

Natural Frequency

1 In the **Model Builder** window, expand the **Campbell Diagram (rotsld)** node, then click **Natural Frequency**.

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(freq)	Hz	Natural frequency

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

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

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 **Preset Studies for Selected Physics Interfaces** > **Eigenfrequency, Prestressed**.

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

5 Click the **Add Study** button in the window toolbar.

6 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.

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

8 Click the **Add Study** button in the window toolbar.

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

STUDY: SRFF

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
Omega (Angular speed of shaft)	`range(0,1000,15000)`	`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** checkbox.
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

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** checkbox. In the associated text field, type `9`.
4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.
5 In the tree, select **Component 1 (comp1)** > **Solid Rotor (rotsld), Controls spatial frame** and **Component 1 (comp1)** > **Beam Rotor (rotbm)**.
6 Click **Disable in Model**.
7 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh: Solid

8 Click to expand the **Filtering and Sorting** section. Find the **Sorting** subsection. Select the **Mode following** checkbox.

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 6.

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

2 From the **Parameter value (Omega (rpm))** list, choose **2000**.

3 From the **Eigenfrequency (Hz)** list, choose **375.85**.

4 In the **Mode Shape (srotf)** toolbar, click **Plot**.

Now, add a Whirl plot to visualize the associated whirl shape.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study: SRFF/Parametric Solutions 2 (sol22)** > **Solid Rotor, Fixed Frame** > **Whirl (rotf1)**.

3 Click the **Add Result Template** button in the window toolbar.

RESULTS

Whirl (rotf1) 1

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

2 From the **Parameter value (Omega (rpm))** list, choose **2000**.

3 From the **Eigenfrequency (Hz)** list, choose **375.85**.

4 In the **Whirl (rotf1) 1** toolbar, click **Plot**.

Now, create a Campbell diagram that compares the eigenfrequencies obtained with the Solid Rotor, and the Solid Rotor, Fixed Frame interfaces.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study: SRFF/Parametric Solutions 2 (sol22)** > **Solid Rotor, Fixed Frame** > **Campbell Diagram (srotf)**.

3 Click the **Add Result Template** button in the window toolbar.

RESULTS

Campbell Diagram (Comparison)

1 In the **Settings** window for **1D Plot Group**, type `Campbell Diagram (Comparison)` in the **Label** text field.

2 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.

3 Select the **y-axis label** checkbox. In the associated text field, type `Natural Frequency (Hz)`.

Natural Frequency (SRFF)

1 In the **Model Builder** window, expand the **Campbell Diagram (Comparison)** node, then click **Natural Frequency**.

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(freq)	Hz	Natural frequency

4 In the **Label** text field, type `Natural Frequency (SRFF)`.

Natural Frequency (Solid Rotor)

1 In the **Model Builder** window, right-click **Campbell Diagram (Comparison)** and choose **Global**.

2 In the **Settings** window for **Global**, type `Natural Frequency (Solid Rotor)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Solid Rotor/ Parametric Solutions 1 (sol3)**.

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

Expression	Unit	Description
abs(rotsld.omega_fix/2/pi)	rad/s	Natural Frequency

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

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

7 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Rotor** > **Acceleration and velocity** > **rotsld.rotf1.freqr - Revolutions per time - 1/s**.

8 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

10 From the **Color** list, choose **From theme**.

11 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

12 In the **Campbell Diagram (Comparison)** toolbar, click **Plot**.

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
Omega (Angular speed of shaft)	`range(0,1000,15000)`	`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** checkbox.

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

RESULTS

Whirl (rotbm)

The default whirl plot is shown in Figure 9.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study: Beam Rotor/Parametric Solutions 3 (sol40)** > **Beam Rotor** > **Campbell Diagram (rotbm)**.

3 Click the **Add Result Template** button in the window toolbar.

4 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

RESULTS

Campbell Diagram (rotbm)

The Campbell diagram from the **Beam Rotor** interface is shown in Figure 8.

Natural Frequency

1 In the **Model Builder** window, expand the **Campbell Diagram (rotbm)** node, then click **Natural Frequency**.

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(freq)`	`Hz`	`Natural frequency`