Created in COMSOL Multiphysics 6.0

Fundamental Eigenfrequency of a Rotating Blade

Introduction

High rotational speed in, for example, gas turbine machinery can result in centrifugal forces of considerable magnitude. This example studies the effects of such forces on the natural frequencies of the structure.

The fictitious forces induced by the rotation give rise to two counteracting effects: *stress-stiffening* and *spin-softening* (or *centrifugal softening*). The former is caused by the stationary stress field created by the centrifugal force and acts to increase the stiffness of the body, and so increase its resonance frequencies. At the same time, any radial displacement away from the axis of rotation increases the centrifugal force, while motion toward the axis decreases it. This effect therefore tends to amplify any radial motion, which is the opposite of stiffening — hence *spin-softening*. Which mechanism is dominating depends on the shape of the particular mode.

In some rotating systems, *Coriolis forces* can also play an important role. These apparent forces split some natural modes of vibration into one co-rotating and one counterrotating *precessing* mode. In particular, this happens for natural modes that include bending of the axis of rotation. For most modes of vibration, however, the Coriolis frequency shifts are small.

Model Definition

In the first part of the modeling, you conduct a modal analysis of a blade mounted on a rigid rotating cylinder. The rotational speed is 3000 rpm (100π rad/s) about the global y-axis. The goal is to compare the fundamental frequency in three cases:

- A basic modal analysis at 0 rpm, thus excluding any fictitious force effects.
- A modal analysis including stress-stiffening and spin-softening at 3000 rpm.
- A complete analysis also including Coriolis effects at 3000 rpm.

The result in the first two cases can be compared directly to an analytical reference solution. Once you are satisfied that the solutions are accurate in all cases, the second step proceeds to compute the first few natural frequencies for a large range of rotational speeds.

FICTITIOUS FORCES IN ROTATING COORDINATE SYSTEMS

The spin-softening and Coriolis effects both ultimately arise because the standard form of the laws governing the deformation of solid objects only hold in *inertial*, that is, nonaccelerating, coordinate systems. When you build and simulate a model in a rotating coordinate system, you must extend the basic laws of motion to account for the acceleration of the system itself.

In COMSOL Multiphysics, you can model the frame acceleration effects by using the Rotating Frame domain feature.

In this particular case, the axis of rotation is the y-axis, see Figure 1. The only explicit boundary condition in the model is on the blade's base, which is fixed to the axis.

STRESS-STIFFENING AND NONLINEAR EFFECTS

To include the stress-stiffening effects in the model, you activate the large deformation option. This redefines the strain measure to include second-order terms, which makes the strain-displacement relation nonlinear. The resulting strain measure is called Green strain.

For the stress stiffening to have any effect on the natural frequencies, you must first solve for the stationary stresses from the centrifugal loading in a stationary analysis. Then you perform the modal analysis using the static solution as a linearization point.

Note that the spin-softening affects not only the modal solution but also adds a positive feedback to the stationary nonlinear solution. If the structure has a natural mode for which spin-softening dominates over the stress-stiffening effect, the natural frequency of this mode becomes zero for some rotational frequency. This means that the structure looses all stiffness, and that no stable solution exists for higher angular velocities. So if the nonlinear solver does not converge for a very fast rotation, it should come as no surprise.

MATERIAL AND GEOMETRICAL PROPERTIES

The blade's material and geometrical properties are given in the table below (see also Figure 1):

MATERIAL PROPERTIES			**GEOMETRIC PROPERTIES**		
E	$221 \cdot 10^9$ Pa	Young's modulus	r	10 in	Cylinder radius
ν	0	Poisson's ratio	L	5 in	Blade length
ρ	7850 kg/m^3	Density	d	0.0625 in	Blade thickness
			b	1 in	Blade width

The approximate equation for the natural frequency in Ref. 1 does not include the Poisson's ratio. To facilitate easy comparison, the Poisson's ratio is set to zero in the modeling.

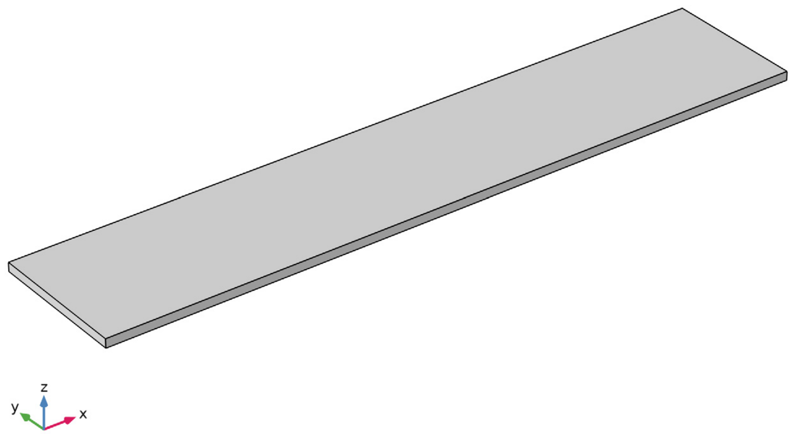


Figure 1: Blade geometry.

ANALYTICAL SOLUTIONS

You can compare COMSOL Multiphysics' results to an accurate analytical approximation. Using Euler beam theory, which is sufficiently accurate for the present geometry, the fundamental angular frequency for the nonrotating case is given by

$$w_0 = 1.875^2\sqrt{\frac{EI}{mL^4}}$$

where E is the Young's modulus, I is the area moment of inertia, m is the mass per unit length, and L is the blade length.

For a rotating blade attached to a rigid axis with radius r and rotating with angular frequency Ω, the fundamental angular frequency according to Ref. 1 is

$$w_\Omega = w_0\sqrt{1+\frac{(m\Omega^2r^4)}{EI}\cdot\left[\frac{1}{8}\left(\frac{L}{r}\right)^3+\frac{1}{10.6}\left(\frac{L}{r}\right)^4-\frac{(\cos\phi)^2}{12.45}\left(\frac{L}{r}\right)^4\right]}$$

This formula assumes a blade of uniform cross section slanted at the so-called stagger angle ϕ. In the present case, $\phi = 0$. The author states without proof that the Coriolis effect is negligible. You can check the validity of this statement by comparing the finite element solutions with and without the Coriolis term included.

Results and Discussion

The following table shows a comparison between the analytical results and the results computed in COMSOL Multiphysics:

TABLE 1: INFLUENCE OF CORIOLIS EFFECT.

CASE	ANALYTICAL RESULT	COMSOL RESULTS
0 rpm	84.4 Hz	84.4 Hz
3000 rpm, no Coriolis effect	123.6 Hz	124 Hz
3000 rpm, with Coriolis effect		124 Hz

Apparently, the Coriolis effect is negligible as stated in Ref. 1. Because including the Coriolis terms makes the system of equations nonsymmetric — and therefore more expensive to solve — there is, in general, no need to include the Coriolis effect when modeling systems with a rigid axis.

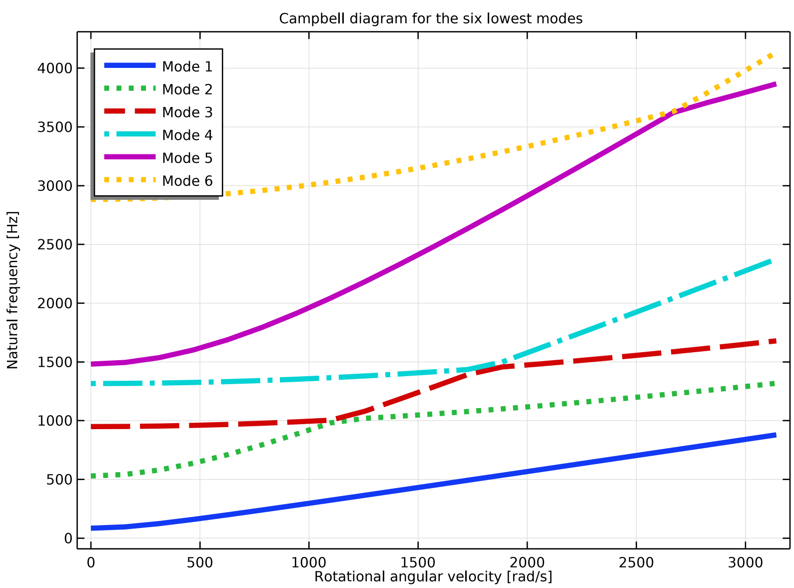


Figure 2: Campbell diagram created from the parametric sweep output file. Notice the crossing modes, which are misinterpreted because of the sorting of the natural frequencies.

The Campbell diagram, see Figure 2, shows that the first 6 natural frequencies all increase with increasing rotational speed. However, the balance between stress stiffening and spin softening is different for the different modes. Stress stiffening is more pronounced for modes 2 and 5 (referring to the mode order at zero rotational frequency) compared to the others. Therefore, the order of the modes in the result file — which is sorted on mode frequency — changes with increasing rotational velocity.

Notes About the COMSOL Implementation

Note that the eigenvalue solver always finds the requested number of modes but not necessarily the ones with lowest eigenfrequencies. Sometimes it can converge on a higher-order mode instead. A jagged appearance of the highest frequency mode in the Campbell diagram is a result of the solver's picking a higher mode instead of requested highest mode for some rotational frequencies. In order to avoid this behavior it is recommended to solve for one additional eigenfrequency in the study and display results for all eigenfrequencies except the one of the highest mode.

When eigenfrequencies are calculated, numerical noise can occasionally be seen via the imaginary part of the eigenfrequency. If the imaginary part is several orders of magnitude smaller than the real part then use the real part as the eigenfrequency and disregard the imaginary part.

Reference

1. W. Carnegie, "Vibrations of Rotating Cantilever Blading", *J. Mech. Engrg Sci.*, vol. 1, no. 3, London, 1959.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/rotating_blade`

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>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

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 In the table, enter the following settings:

Name	Expression	Value	Description
Omega	0*pi[rad/s]	0 rad/s	Angular velocity
L	5[in]	0.127 m	Blade length
b	1[in]	0.0254 m	Blade width
d	0.0625[in]	0.0015875 m	Blade thickness
E0	221[GPa]	2.21E11 Pa	Young's modulus
rho0	7850[kg/m^3]	7850 kg/m³	Density
r	10[in]	0.254 m	Inner radius
mpl	rho0*b*d	0.31653 kg/m	Mass per unit length
I	b*d^3/12	8.4683E-12 m^4	Area moment of inertia
f_0	1.875^2*sqrt(E0*I/(mpl*L^4))/(2*pi)	84.353 1/s	Euler beam resonance frequency
f_ref	f_0*sqrt(1+mpl*Omega^2*r^4/(E0*I)*((L/r)^3/8+(L/r)^4*(1/10.6-1/12.45)))	84.353 rad/s	Reference resonance frequency

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

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

4 In the **Depth** text field, type `b`.

5 In the **Height** text field, type `d`.

6 Locate the **Position** section. In the **x** text field, type `r`.

7 In the **z** text field, type `-d/2`.

8 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

MATERIALS

Material 1 (mat1)

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

2 Select Domain 1 only.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E0	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0	1	Young's modulus and Poisson's ratio
Density	rho	rho0	kg/m³	Basic

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Volume Forces>Rotating Frame**.

2 Select Domain 1 only.

3 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

4 From the **Axis of rotation** list, choose **y-axis**.

5 In the Ω text field, type `Omega`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 1 only.

MESH 1

Mapped 1

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

2 Select Boundary 4 only.

Distribution 1

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

2 Select Edge 4 only.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edge 5 only.

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

4 In the **Number of elements** text field, type 25.

Mapped 1

Right-click **Mapped 1** and choose **Build Selected**.

Swept 1

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

Distribution 1

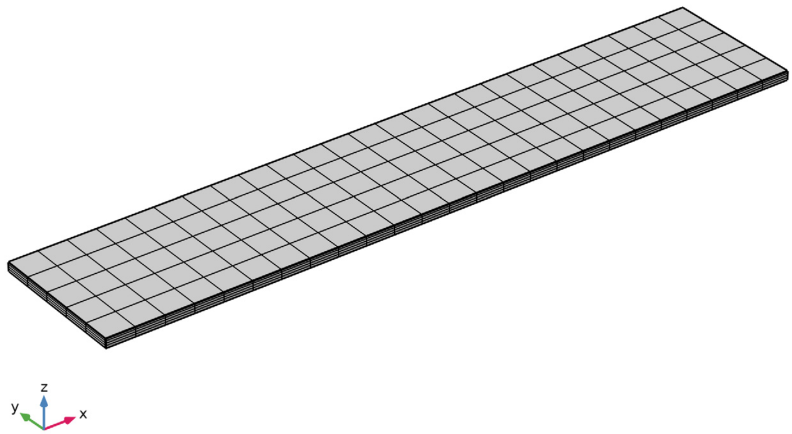
1 Right-click **Swept 1** and choose **Distribution**.

2 Select Domain 1 only.

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

4 In the **Number of elements** text field, type 4.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



STUDY 1

The default solver settings consist of a sequence of a stationary study step followed by an eigenfrequency analysis. COMSOL Multiphysics stores the solution from the stationary solver and uses it as the linearization point for the eigenfrequency analysis. Together with the fact that a geometrically nonlinear analysis is preselected, this ensures that effects of stress-stiffening and spin-softening are included in the calculated eigenfrequencies.

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

RESULTS

Surface 1

The default plot shows the scaled deformation for the first mode of vibration for the nonrotating blade.

The first mode occurs at 84.4 Hz. This agrees well with the analytical solution, which you can compute as described below.

Global Evaluation 1

1 In the **Model Builder** window, expand the **Results>Mode Shape (solid)** node.

2 Right-click **Results>Derived Values** and choose **Global Evaluation**.

3 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

4 From the **Eigenfrequency selection** list, choose **First**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Global definitions>Parameters>f_ref - Reference resonance frequency - rad/s**.

6 Click **Evaluate**.

The table in the **Table** window shows that the analytical value of the first mode is also 84.4 Hz.

Next, solve for the eigenfrequencies of the blade rotating at 3000 RPM.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Omega	100*pi[rad/s]	314.16 rad/s	Angular velocity

STUDY 1

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

RESULTS

Mode Shape (solid)

The default plot now shows the scaled deformation for the first mode of vibration when the blade is rotating at 3000 RPM. In this case the first eigenmode occurs at 124 Hz. This result clearly shows the effect of stress-stiffening.

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, click **Evaluate**.

The table in the Table window shows that the analytical value of the first mode is 123.6 Hz.

In the next section, you will also include the Coriolis effect and solve for the eigenfrequencies of the blade rotating at 3000 RPM.

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Rotating Frame 1**.

2 In the **Settings** window for **Rotating Frame**, locate the **Frame Acceleration Effect** section.

3 Select the **Coriolis force** check box.

STUDY 1

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

RESULTS

Mode Shape (solid)

The default plot now shows the scaled deformation for the first mode of vibration when the blade is rotating at 3000 RPM. In this case the first eigenmode also occurs at 124 Hz. This result shows that enabling the Coriolis effect in the equations does not change the first natural frequency appreciably compared to the previous case.

In the next section, you perform a parametric study of the effect of rotational velocity and use the results to generate a Campbell diagram.

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Rotating Frame 1**.

2 In the **Settings** window for **Rotating Frame**, locate the **Frame Acceleration Effect** section.

3 Clear the **Coriolis force** check box.

STUDY 1

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 velocity)	`pi*range(0,50,1000)`	`rad/s`

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 In the **Search for eigenfrequencies around** text field, type `1000`.

Compute one additional eigenfrequency to avoid the possibility that the solver converges on a higher-order mode.

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

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

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

RESULTS

Mode Shape (solid) 1

The default plot shows the scaled displacement magnitude for the first mode for a rotational velocity of 30,000 RPM as a surface plot in Mode Shape (solid) 1. This mode occurs at a frequency of 723 Hz.

Use the **Parameter value (Omega)** and **Eigenfrequency** lists to find out the six lowest eigenfrequencies for each rotational velocity. A more elegant way to visualize this result is by creating a Campbell diagram.

1D Plot Group 3

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 1/Parametric Solutions 1 (sol3)**.

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

Display the 6 lowest eigenfrequencies.

5 In the **Eigenfrequency indices (1-7)** text field, type `range(1,1,6)`.

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

7 In the **Title** text area, type `Campbell diagram for the six lowest modes`.

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

9 In the associated text field, type `Rotational angular velocity [rad/s]`.

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

11 In the associated text field, type `Natural frequency [Hz]`.

12 Locate the **Axis** section. Select the **Manual axis limits** check box.

13 In the **x minimum** text field, type `-100`.

14 In the **x maximum** text field, type `4000`.

15 In the **y minimum** text field, type `0`.

16 In the **y maximum** text field, type `4500`.

Global 1

1 Right-click **1D Plot Group 3** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>solid.freq - Frequency - Hz**.

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

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

5 In the **Width** text field, type `4`.

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

7 In the table, enter the following settings:

Legends
Mode 1
Mode 2
Mode 3
Mode 4
Mode 5
Mode 6

Campbell diagram

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

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

3 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

4 In the **Campbell diagram** toolbar, click **Plot**.