

Stress and Modal Analysis of a Composite Wheel Rim

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Nowadays, there is a trend to manufacture wheel rims using composite material instead of aluminum. The primary reason is to reduce the unsprung mass which leads to a faster response time and thus better acceleration, braking and cornering performances. Typically, a carbon fiber composite material is used to manufacture a composite wheel rim.

To understand and improve the design of a composite wheel rim, an example model is built in COMSOL. The example demonstrates the modeling of a wheel rim made up of a carbon-epoxy laminate. The composite laminate has a different number of plies in different regions of the wheel rim. First, a stress analysis of the wheel rim is performed in which the rim is subjected to the inflation pressure and the tire load. In order to compute the modal response of the wheel rim, a prestressed eigenfrequency analysis is performed in which the rim is subjected to the rotating frame forces.

Model Definition

The wheel rim geometry is shown in Figure 1.



Figure 1: Model geometry of a composite wheel rim.

The geometry consists of two main regions which have different laminate stacking sequences:

- Rim region with a 16 layer laminate
- Hub-spoke region with a 8 layer laminate

STACKING SEQUENCE

A symmetric ply layup is considered in all the regions. The stacking sequence for the two regions is as follows:

- Hub and spokes: [0/45/90/-45]_s (Figure 2)
- Rim: [[0/45/90/-45]₈]₂ (Figure 3)



Figure 2: Stacking sequence [0/45/90/-45]_s in hub-spoke region.

Each ply is made up of carbon-epoxy material and has a thickness of 0.4 mm.

The normal vector orientations for the hub-spoke and rim regions are shown in Figure 4 and Figure 5, respectively. This is the direction in which layer stacking is interpreted from bottom to top. Note that the geometric surface (that is, the meshed boundary) represents

the laminate's top surface. This is defined using the corresponding setting in the Layered Material Link and Layered Material Stack nodes.



Figure 3: Stacking sequence $[[0/45/90/-45]_s]_2$ in rim region.



y, Z x

Figure 4: The normal vector on each hub and spokes boundary. This is the direction in which the layer stacking is interpreted from bottom to top.



Figure 5: The normal vector on each rim boundary. This is the direction in which the layer stacking is interpreted from bottom to top.

MATERIAL PROPERTIES

Each ply is made up of carbon-epoxy composite material. The homogenized orthotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1 and the averaged density of the lamina is 1700 kg/m^3 .

Material property	Value
$\{E_1, E_2, E_3\}$	{134, 9.2, 9.2} GPa
{G ₁₂ , G ₂₃ , G ₁₃ }	{4.8, 4.8, 4.8} GPa
$\{v_{12}, v_{23}, v_{13}\}$	{0.28, 0.28, 0.28}

TABLE I: MATERIAL PROPERTIES OF A LAMINA.

FINITE ELEMENT MESH

The structure is discretized using a free triangular mesh, as shown in Figure 6.



y, z x

Figure 6: The finite element mesh for the wheel rim.

CONSTRAINTS

• Each bolt hole where the wheel rim is attached to the wheel hub is fixed.

LOADS

Two types of analyses are performed:

- Stationary analysis: This analysis is performed for the tire pressure and total load on the wheels. The overpressure is 2 bar = 200 kPa. The total load carried by the wheel corresponds to a weight of 1120 kg. It is applied as a pressure on the rim surfaces where the tire is in contact. Assume that the load distribution in the circumferential direction can be approximated as $p = p_0 \cos(3\vartheta)$, where ϑ is the angle from the point of contact between the road and the tire. The loaded area thus extends 30° in each direction from the load peak.
- Prestressed Eigenfrequency analysis: This analysis is performed with rotating frame forces when the wheel rim rotates with 3000 rpm.

Results and Discussion

Figure 7 shows the von Mises stress distribution in the rim and hub-spokes regions under inflation pressure and the tire load. High stresses are present in the spoke where the tire load is acting.

The von Mises stress distribution for each hub-spoke and rim layer is shown in Figure 8 and Figure 9, respectively. High stresses occur in layer 3 (symmetric) of the hub-spoke region and layer 14, 15 of the rim region.

Layered Material Slice: von Mises Stress, Gauss Point Evaluation (N/m²)



Figure 7: The von Mises stress distribution in a composite wheel rim.

The through-thickness variations of the von Mises stress at two points on the rim and spoke region are shown in Figure 10. Each ply exhibits different stress levels depending on the fiber orientation. The maximum stresses, at the particular geometric locations, are found in the layer 1 of rim region and layer 2 of spoke region.

Figure 11 shows the first four eigenmodes of composite wheel rim when the rim is rotating at 3000 rpm. The first natural frequency found for the composite wheel rim corresponds to approximately 8000 rpm. This is much higher compared to the operating range of the wheel and hence the structure is in a safe zone and does not have a critical speed. A rather high natural frequency is obtained because of the high strength-to-weight ratio of the carbon-epoxy composite material.



Figure 8: The von Mises stress distribution in each layer of hub and spokes region.



Layered Material Slice: von Mises Stress, Gauss Point Evaluation (N/m²)

Figure 9: The von Mises stress distribution in each layer of rim region.



Figure 10: Through-thickness variation of von Mises stress for a point on rim and spoke.



Figure 11: First four Eigenfrequency and mode shapes of the composite wheel rim.

Notes About the COMSOL Implementation

- Modeling a composite laminate as a layered shell requires a surface geometry, in general referred to as a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers, and control the number of through-thickness mesh elements for each layer.
- From a constitutive model's point of view, the *equivalent single layer (ESL)* theory is used in the **Layered Linear Elastic** material model of the Shell interface.
- By default, loads are assumed to be applied on the shell's midsurface. In reality, however, loads may often act on the top or bottom surface such as the tire pressure and weight in this case. If forces are applied on a different surface than the midsurface, the actual area of applied load may also differ, which is generally true for curved surfaces. The forces applied away from the midsurface also gives rise to moments which are significant for thick shells. To take this effect into account loads must be scaled properly to achieve a correct load contribution. In particular for curved and/or tick shells care must be taken with respect to the load's location. The load location is be specified using the **Through-Thickness Location** section in the **Face Load** feature.
- In this example, Coriolis force giving rise to gyroscopic damping effect is neglected. The gyroscopic damping can change the eigenmodes and natural frequencies especially at high angular velocities.

Application Library path: Composite_Materials_Module/
Dynamics_and_Vibration/composite_wheel_rim

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🔗 Model Wizard.

MODEL WIZARD

I In the Model Wizard window, click 间 3D.

- 2 In the Select Physics tree, select Structural Mechanics>Shell (shell).
- 3 Click Add.
- 4 Click 🔿 Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click 🗹 Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- 3 Click **b** Load from File.
- 4 Browse to the model's Application Libraries folder and double-click the file composite_wheel_rim_parameters.txt.

DEFINITIONS

Analytic I (an I)

- I In the Model Builder window, expand the Component I (compl)>Definitions node.
- 2 Right-click **Definitions** and choose **Functions>Analytic**.
- 3 In the Settings window for Analytic, type loadDistr in the Function name text field.
- 4 Locate the Definition section. In the Expression text field, type (abs(atan2(x,y)-z*pi/180)<pi/6)*cos(3*(atan2(x,y)-z*pi/180)).</p>
- 5 In the Arguments text field, type x, y, z.
- 6 Locate the Units section. In the table, enter the following settings:

Argument	Unit
x	m
у	m
z	1

7 In the Function text field, type Pa.

Cylindrical System 2 (sys2) In the **Definitions** toolbar, click $\begin{bmatrix} z & y \\ z & z \end{bmatrix}$ **Coordinate Systems** and choose **Cylindrical System**.

GEOMETRY I

Import I (impl)

- I In the Model Builder window, expand the Component I (compl)>Geometry I node.
- 2 Right-click Geometry I and choose Import.
- 3 In the Settings window for Import, locate the Import section.
- 4 From the Source list, choose COMSOL Multiphysics file.
- 5 Click 📂 Browse.
- 6 Browse to the model's Application Libraries folder and double-click the file composite_wheel_rim.mphbin.
- 7 Click ा Import.
- 8 Click 틤 Build Selected.

Rim

- I In the Geometry toolbar, click 🝖 Selections and choose Explicit Selection.
- 2 In the Settings window for Explicit Selection, type Rim in the Label text field.
- **3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
- **4** On the object **imp1**, select Boundaries 11–19, 24, 26, 28, and 34–36 only.

HubAndSpokes

- I In the Geometry toolbar, click 💁 Selections and choose Complement Selection.
- **2** In the **Settings** window for **Complement Selection**, type HubAndSpokes in the **Label** text field.
- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Input Entities section. Click + Add.
- 5 In the Add dialog box, select Rim in the Selections to invert list.
- 6 Click OK.

TireAttachment

- I In the Geometry toolbar, click 🔓 Selections and choose Explicit Selection.
- 2 In the Settings window for Explicit Selection, type TireAttachment in the Label text field.
- **3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 On the object impl, select Boundaries 16–19, 24, 26, 35, and 36 only.

FixedToHub

- I In the Geometry toolbar, click 🛯 Selections and choose Explicit Selection.
- 2 In the Settings window for Explicit Selection, locate the Entities to Select section.
- **3** From the **Geometric entity level** list, choose **Boundary**.
- 4 In the Label text field, type FixedToHub.
- 5 On the object impl, select Boundaries 8 and 30 only.

Rotate | (rot])

- I In the Geometry toolbar, click 💭 Transforms and choose Rotate.
- 2 Select the object impl only.
- 3 In the Settings window for Rotate, locate the Rotation section.
- 4 In the Angle text field, type range(0,72,288).
- 5 Click 🟢 Build All Objects.

SprokeRimUnit

- I In the Geometry toolbar, click 😼 Selections and choose Cylinder Selection.
- 2 In the Settings window for Cylinder Selection, type SprokeRimUnit in the Label text field.
- 3 Locate the Geometric Entity Level section. From the Level list, choose Boundary.
- 4 Locate the Size and Shape section. In the Outer radius text field, type inf.
- 5 In the Start angle text field, type 18.
- 6 In the End angle text field, type 90.
- 7 Locate the Output Entities section. From the Include entity if list, choose Entity inside cylinder.
- 8 Click 틤 Build Selected.

DEFINITIONS

Integration 1 (intop1)

- I In the Definitions toolbar, click 🖉 Nonlocal Couplings and choose Integration.
- 2 In the Settings window for Integration, locate the Source Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 From the Selection list, choose TireAttachment.
- 5 Locate the Advanced section. From the Frame list, choose Material (X, Y, Z).

GLOBAL DEFINITIONS

Material: Carbon-Epoxy

The laminate is symmetric. Therefore, it is sufficient to define only a part of it in the **Layered Material** node; the transformation into the full laminate is performed through the layered material settings in the **Layered Material Link** and the **Layered Material Stack** nodes.

- I In the Model Builder window, under Global Definitions right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Material: Carbon-Epoxy in the Label text field.

Layered Material I (Imat I)

- I Right-click Materials and choose Layered Material.
- 2 In the Settings window for Layered Material, locate the Layer Definition section.
- **3** In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Material: Carbon- Epoxy (mat1)	0	th	1

4 Click **Add** three times.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 2	Material: Carbon- Epoxy (mat1)	45	th	1
Layer 3	Material: Carbon- Epoxy (matl)	90	th	1
Layer 4	Material: Carbon- Epoxy (matl)	- 45	th	1

6 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type 0.6.

The laminate part defined in the **Layered Material** node can be transformed into a full symmetric laminate using a transform option in the layered material settings.

MATERIALS

Layered Material Link 1 (Ilmat1)

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Layers>Layered Material Link.
- 2 In the Settings window for Layered Material Link, locate the Boundary Selection section.
- 3 From the Selection list, choose HubAndSpokes.
- 4 Locate the Layered Material Settings section. From the Transform list, choose Symmetric.
- 5 Locate the Orientation and Position section. From the Position list, choose Top side on boundary.
- **6** Locate the **Layered Material Settings** section. Click **Layer Stack Preview** in the upper-right corner of the section.
- 7 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type 0.6.

Layered Material Stack 1 (stlmat1)

- I Right-click Materials and choose Layers>Layered Material Stack.
- 2 In the Settings window for Layered Material Stack, locate the Boundary Selection section.
- 3 From the Selection list, choose Rim.
- 4 Locate the Layered Material Settings section. From the Transform list, choose Repeated.
- 5 In the Number of repeats text field, type 2.
- 6 Locate the Orientation and Position section. From the Position list, choose Top side on boundary.

Layered Material Link 1 (stlmat1.stllmat1)

- I In the Model Builder window, click Layered Material Link I (stlmat1.stllmat1).
- 2 In the Settings window for Layered Material Link, locate the Link Settings section.
- 3 From the Transform list, choose Symmetric.

Layered Material Stack 1 (stlmat1)

- I In the Model Builder window, click Layered Material Stack I (stlmat1).
- 2 In the Settings window for Layered Material Stack, click to expand the Preview Plot Settings section.
- 3 In the Thickness-to-width ratio text field, type 0.6.
- **4** Locate the **Layered Material Settings** section. Click **Layer Stack Preview** in the upper-right corner of the section.

SHELL (SHELL)

Layered Linear Elastic Material I

- I In the Model Builder window, under Component I (compl) right-click Shell (shell) and choose Material Models>Layered Linear Elastic Material.
- **2** In the Settings window for Layered Linear Elastic Material, locate the Boundary Selection section.
- 3 From the Selection list, choose All boundaries.
- 4 Locate the Linear Elastic Material section. From the Material symmetry list, choose Orthotropic.

GLOBAL DEFINITIONS

Material: Carbon-Epoxy (mat1)

- I In the Model Builder window, under Global Definitions>Materials click Material: Carbon-Epoxy (matl).
- 2 In the Settings window for Material, locate the Material Contents section.

Property	Variable	Value	Unit	Property group	
Young's modulus	{Evector1, Evector2, Evector3}	{134e9, 9.2e9, 9.2e9}	Pa	Orthotropic	
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	{0.28, 0.28, 0.28}	1	Orthotropic	
Shear modulus	{Gvector I, Gvector2, Gvector3}	{4.8e9, 4.8e9, 4.8e9}	N/m²	Orthotropic	
Density	rho	1700	kg/m³	Basic	

3 In the table, enter the following settings:

SHELL (SHELL)

Fixed Constraint I

- I In the Physics toolbar, click 🔚 Boundaries and choose Fixed Constraint.
- 2 In the Settings window for Fixed Constraint, locate the Boundary Selection section.
- 3 From the Selection list, choose FixedToHub.

Face Load I

- I In the Physics toolbar, click 🔚 Boundaries and choose Face Load.
- 2 In the Settings window for Face Load, locate the Boundary Selection section.
- 3 From the Selection list, choose Rim.
- 4 Locate the Through-Thickness Location section. From the list, choose Bottom surface.
- 5 Locate the Force section. From the Load type list, choose Pressure.
- **6** In the *p* text field, type -pInflation.

Face Load 2

- I Right-click Face Load I and choose Duplicate.
- 2 In the Settings window for Face Load, locate the Boundary Selection section.
- 3 From the Selection list, choose TireAttachment.
- 4 Locate the Coordinate System Selection section. From the Coordinate system list, choose Cylindrical System 2 (sys2).
- 5 Locate the Force section. From the Load type list, choose Force per unit area.
- **6** Specify the \mathbf{F}_{A} vector as

<pre>-loadAmpl*loadDistr(X,Y,phiLoad)</pre>	
0	phi
<pre>0.2*loadAmpl*loadDistr(X,Y,phiLoad)*(2*(Z>0)-1)</pre>	a

- 7 Click the 🐱 Show More Options button in the Model Builder toolbar.
- 8 In the Show More Options dialog box, select Physics>Equation-Based Contributions in the tree.

9 In the tree, select the check box for the node Physics>Equation-Based Contributions.

IO Click OK.

Global Equations 1

- I In the Physics toolbar, click 🖄 Global and choose Global Equations.
- 2 In the Settings window for Global Equations, locate the Global Equations section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (l)	Initial value (u_0) (I)	Initial value (u_t0) (1/s)	Description
loadAmpl	<pre>loadAmpl* intop1(loadDistr(X,Y,0)* cos(atan2(X,Y)))- tireLoad</pre>	0	0	

4 Locate the Units section. Click **Select Source Term Quantity**.

- 5 In the Physical Quantity dialog box, type force in the text field.
- 6 Click 🔫 Filter.
- 7 In the tree, select General>Force (N).
- 8 Click OK.

Rotating Frame 1

- I In the Physics toolbar, click 🕞 Boundaries and choose Rotating Frame.
- 2 In the Settings window for Rotating Frame, locate the Rotating Frame section.
- **3** In the Ω text field, type omega.
- 4 From the Rotational direction list, choose Clockwise.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Sequence Type section.
- 3 From the list, choose User-controlled mesh.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.07.
- 5 In the Minimum element size text field, type 0.006.
- 6 In the Maximum element growth rate text field, type 1.2.
- 7 Click 📗 Build All.

STUDY: STATIC

- I In the Model Builder window, click Study I.
- 2 In the Settings window for Study, type Study: Static in the Label text field.

Step 1: Stationary

- I In the Model Builder window, under Study: Static click Step I: 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 I (compl)>Shell (shell)>Rotating Frame I.
- 5 Click 🕢 Disable.
- 6 In the Home toolbar, click **=** Compute.

RESULTS

Stress (shell)

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

Surface 1

- I In the Model Builder window, expand the Stress (shell) node, then click Surface I.
- 2 In the Settings window for Surface, click to expand the Range section.
- 3 Select the Manual color range check box.
- **4** In the **Minimum** text field, type **0**.
- 5 In the Maximum text field, type 2e8.
- 6 In the Stress (shell) toolbar, click 💽 Plot.
- 7 In the Home toolbar, click 📕 Add Predefined Plot.

ADD PREDEFINED PLOT

- I Go to the Add Predefined Plot window.
- 2 In the tree, select Study: Static/Solution I (soll)>Shell>Stress, Slice (shell).
- 3 Click Add Plot in the window toolbar.
- 4 In the tree, select Study: Static/Solution 1 (soll)>Shell>Stress, Through Thickness (shell).
- 5 Click Add Plot in the window toolbar.
- 6 In the Home toolbar, click **and Add Predefined Plot**.

RESULTS

Layered Material Slice I

- I In the Model Builder window, expand the Results>Stress, Slice (shell) node, then click Layered Material Slice I.
- 2 In the Settings window for Layered Material Slice, click to expand the Range section.
- **3** Select the Manual color range check box.
- **4** In the **Minimum** text field, type **0**.
- 5 In the Maximum text field, type 2e8.
- 6 In the Stress, Slice (shell) toolbar, click 💿 Plot.

Through Thickness I

- I In the Model Builder window, expand the Results>Stress, Through Thickness (shell) node, then click Through Thickness I.
- 2 In the Settings window for Through Thickness, locate the Selection section.
- **3** Click to select the **EXACTIVATE Selection** toggle button.
- 4 Click Clear Selection.
- 5 Select Points 74 and 95 only.
- 6 Locate the y-Axis Data section. From the Unit list, choose mm.
- 7 Find the Interface positions subsection. From the Show interface positions list, choose All interfaces.
- 8 Click to expand the Legends section. From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

Legends

Point=74 (Rim)

Point=95 (Spoke)

IO In the Stress, Through Thickness (shell) toolbar, click 💿 Plot.

Stress, Slice (Hub and Spokes)

- I In the Model Builder window, right-click Stress, Slice (shell) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Stress, Slice (Hub and Spokes) in the Label text field.
- **3** Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 From the Selection list, choose HubAndSpokes.

- 5 Select the Apply to dataset edges check box.
- 6 Locate the Plot Settings section. From the View list, choose New view.

Layered Material Slice 1

- I In the Model Builder window, expand the Stress, Slice (Hub and Spokes) node, then click Layered Material Slice I.
- **2** In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.
- **3** From the Location definition list, choose Layer midplanes.
- 4 Locate the Layout section. From the Displacement list, choose Rectangular.
- 5 In the **Relative y-separation** text field, type 0.15*6.
- 6 Select the Show descriptions check box.
- 7 In the **Relative separation** text field, type 0.2*2.

Deformation

- I In the Model Builder window, expand the Layered Material Slice I node, then click Deformation.
- 2 In the Settings window for Deformation, locate the Scale section.
- 3 Select the Scale factor check box. In the associated text field, type 1.
- **4** In the Stress, Slice (Hub and Spokes) toolbar, click **O** Plot.

Stress, Slice (Rim)

- I In the Model Builder window, right-click Stress, Slice (Hub and Spokes) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Stress, Slice (Rim) in the Label text field.
- 3 Locate the Selection section. From the Selection list, choose Rim.
- 4 Locate the Plot Settings section. From the View list, choose New view.
- **5** Click the **Com Extents** button in the **Graphics** toolbar.

Layered Material Slice 1

- I In the Model Builder window, expand the Stress, Slice (Rim) node, then click Layered Material Slice I.
- 2 In the Settings window for Layered Material Slice, locate the Range section.
- 3 In the Maximum text field, type 1e8.
- 4 Locate the Layout section. In the Relative x-separation text field, type 0.15*4.

- 5 In the Relative y-separation text field, type 0.15*4.
- 6 Clear the Show descriptions check box.

Table Annotation 1

- I In the Model Builder window, right-click Stress, Slice (Rim) and choose More Plots> Table Annotation.
- 2 In the Settings window for Table Annotation, locate the Data section.
- 3 From the Source list, choose Local table.
- **4** In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
-0.7	0	0	Layer 1
2.8	0	0	Layer 4
-0.7	2.4	0	Layer 13
2.8	2.4	0	Layer 16

5 Locate the Coloring and Style section. Clear the Show point check box.

6 In the Stress, Slice (Rim) toolbar, click 💿 Plot.

ADD STUDY

- I In the Home toolbar, click \sim°_{1} Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select General Studies>Stationary.
- 4 Click Add Study in the window toolbar.
- 5 In the Home toolbar, click ~ 2 Add Study to close the Add Study window.

STUDY: EIGENFREQUENCY

- I In the Model Builder window, click Study 2.
- 2 In the Settings window for Study, type Study: Eigenfrequency in the Label text field.

Step 1: Stationary

- I In the Model Builder window, under Study: Eigenfrequency 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.
- In the tree, select Component I (comp1)>Shell (shell)>Face Load I,
 Component I (comp1)>Shell (shell)>Face Load 2, and Component I (comp1)>Shell (shell)>
 Global Equations 1.

5 Click 🖉 Disable.

Eigenfrequency

- I In the Study toolbar, click 🔀 Study Steps and choose Eigenfrequency>Eigenfrequency.
- 2 In the Settings window for Eigenfrequency, locate the Study Settings section.
- **3** Find the **Values of linearization point** subsection. Select the **Include geometric nonlinearity** check box.
- **4** In the **Study** toolbar, click **= Compute**.

RESULTS

Mode Shape (shell) In the Mode Shape (shell) toolbar, click 🗿 Plot.

24 | STRESS AND MODAL ANALYSIS OF A COMPOSITE WHEEL RIM