



# Stress and Modal Analysis of a Composite Wheel Rim

## Introduction

---

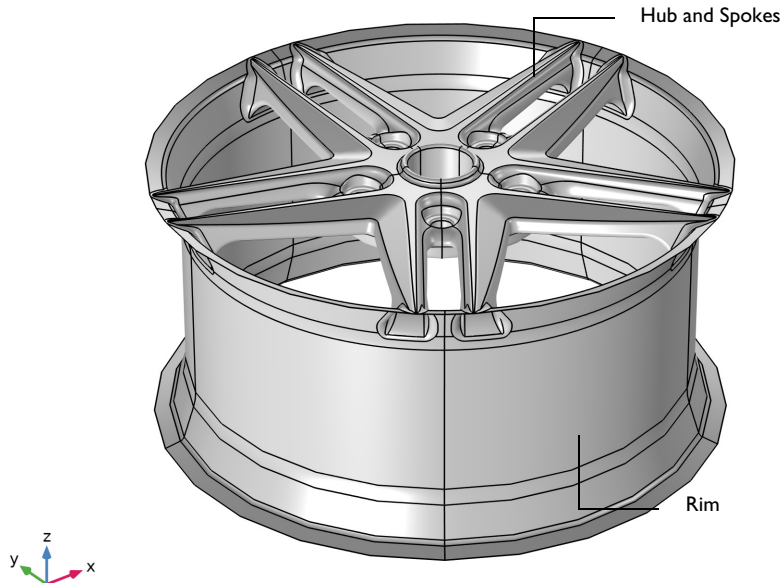
Nowadays, there is a trend to manufacture wheel rims using composite material instead of aluminum. The primary reason is to reduce the unsprung mass which leads to a faster response time and thus better acceleration, braking and cornering performances. Typically, a carbon fiber composite material is 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:  $[[0/45/90/-45]_s]_2$  (Figure 3)

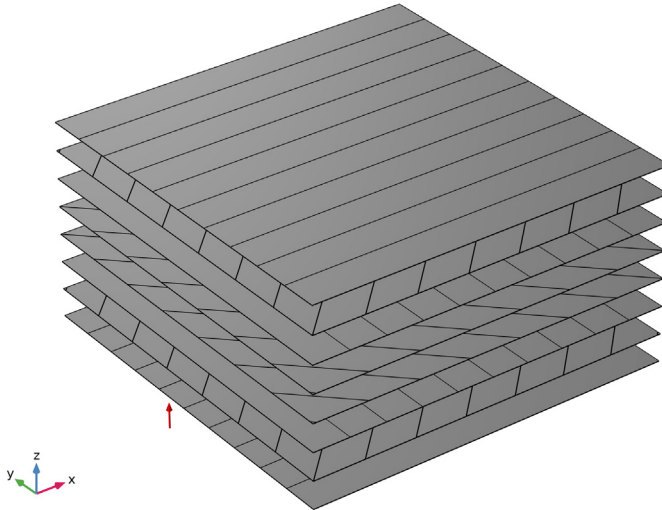


Figure 2: Stacking sequence  $[0/45/90/-45]_s$  in hub-spoke region.

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.

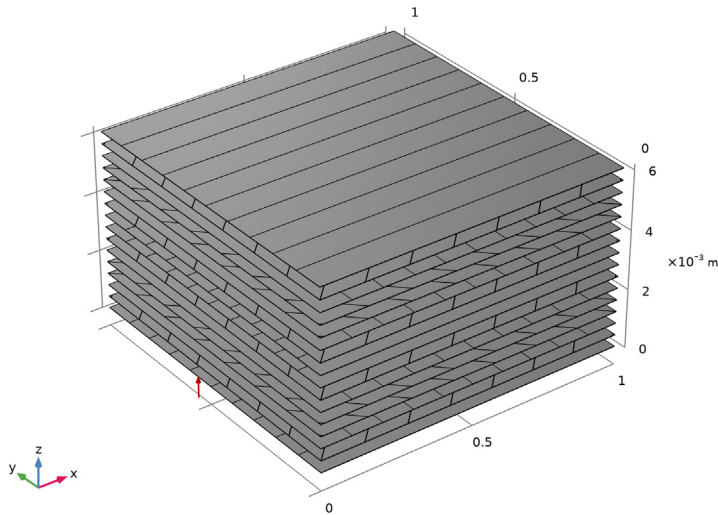


Figure 3: Stacking sequence  $[[0/45/90/-45]_s]_2$  in rim region.

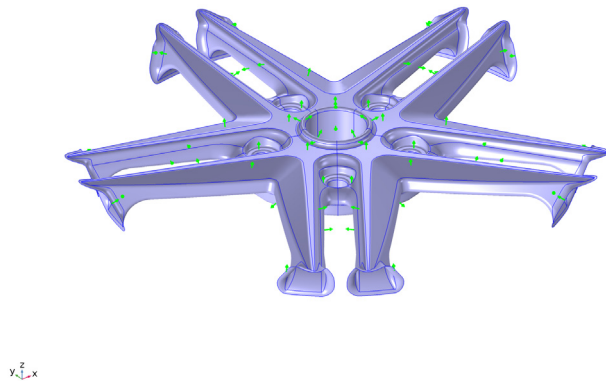


Figure 4: The normal vector on each hub and spokes boundary. This is the direction in which the layer stacking is interpreted from bottom to top.

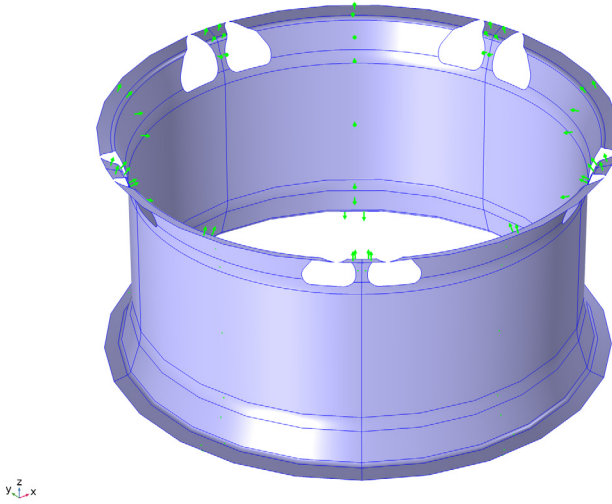


Figure 5: The normal vector on each rim boundary. This is the direction in which the layer stacking is interpreted from bottom to top.

## MATERIAL PROPERTIES

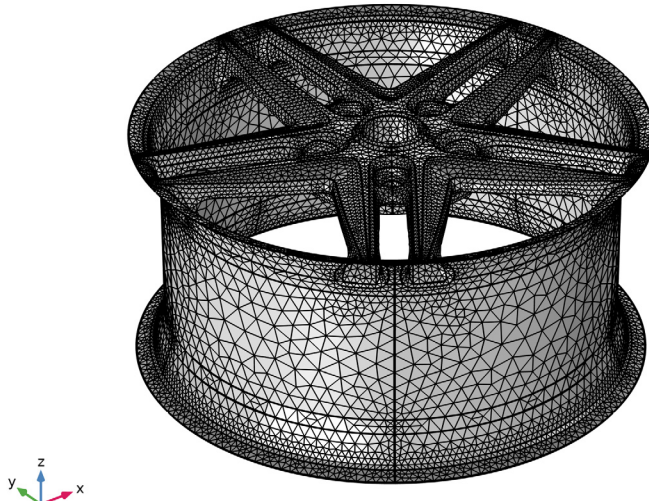
Each ply is made up of carbon–epoxy composite material. The homogenized transversely isotropic material properties (Young’s modulus, shear modulus, and Poisson’s ratio) are given in [Table 1](#) and the averaged density of the lamina is  $1700 \text{ kg/m}^3$ .

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Material property	Value
$\{E_{11}, E_{22}\}$	$\{134, 9.2\} \text{ GPa}$
$G_{12}$	$4.8 \text{ GPa}$
$\{\nu_{12}, \nu_{23}\}$	$\{0.28, 0.28\}$

## FINITE ELEMENT MESH

The structure is discretized using a free triangular mesh, as shown in [Figure 6](#).



*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  $p = p_0 \cos(\vartheta)$ , where  $\vartheta$  is the angle from the point of contact between the road and the tire. The loaded area thus extends  $30^\circ$  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.

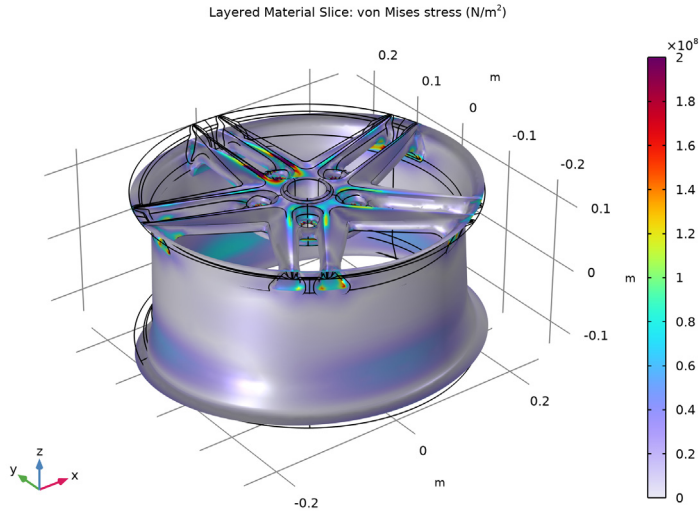


Figure 7: The von Mises stress distribution in a composite wheel rim.

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.

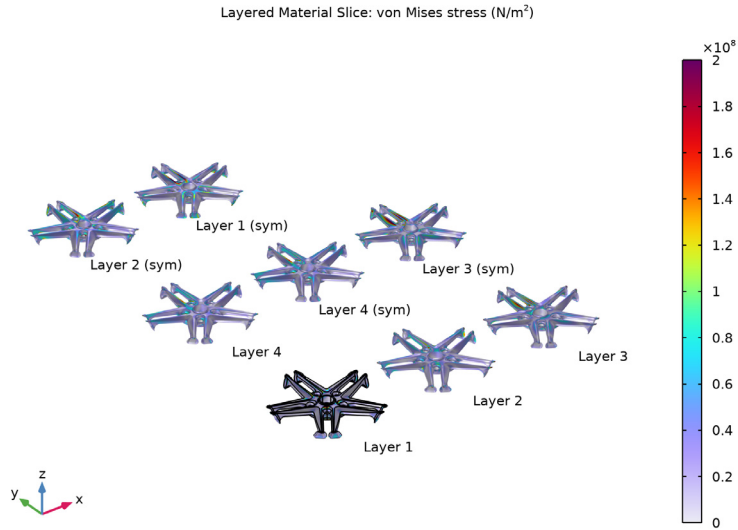


Figure 8: The von Mises stress distribution in each layer of hub and spokes region.

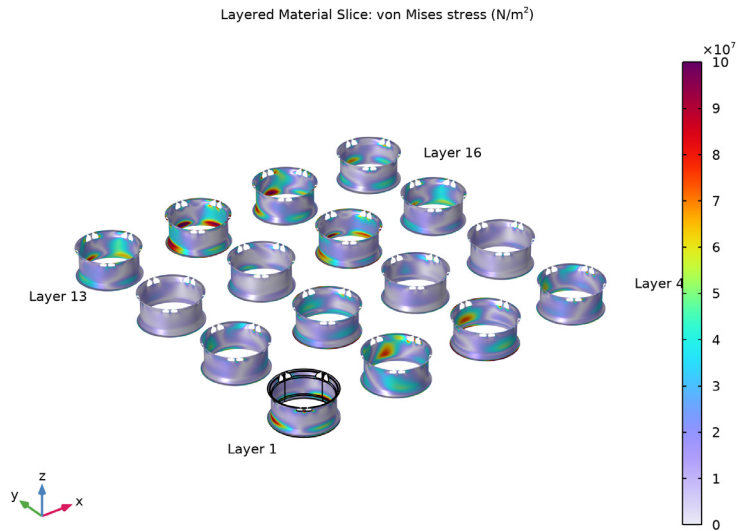


Figure 9: The von Mises stress distribution in each layer of rim region.



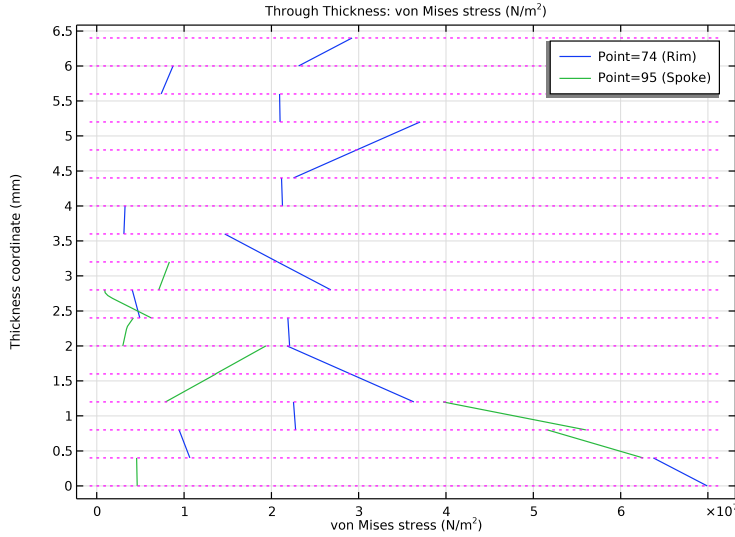


Figure 10: Through-thickness variation of von Mises stress for a point on rim and spoke.

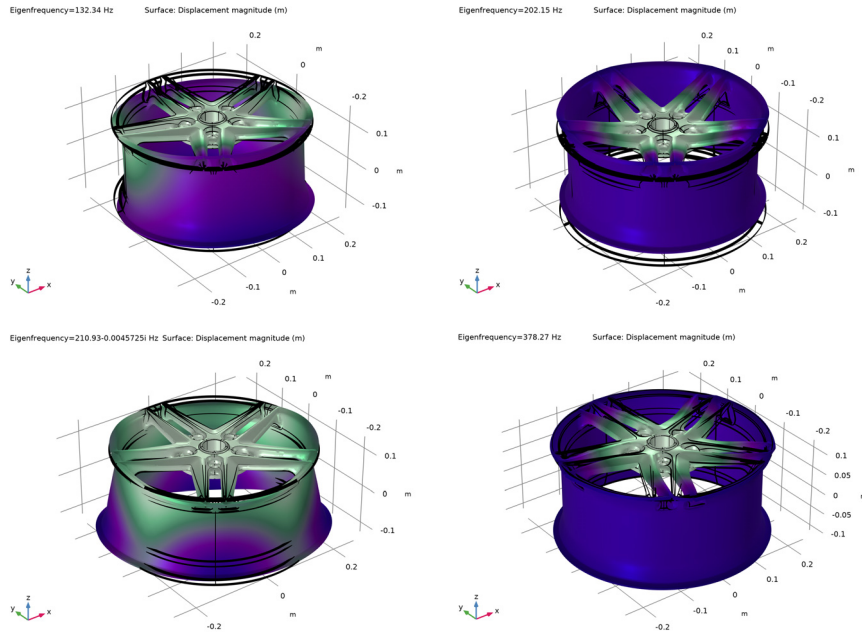


Figure 11: First four Eigenfrequency and mode shapes of the composite wheel rim.

## 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 **Linear Elastic Material, Layered** 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 thick shells care must be taken with respect to the load's location. The load location is 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

1 In the **Model Wizard** window, click  **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*

- 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 `composite_wheel_rim_parameters.txt`.

## DEFINITIONS

### *Analytic 1 (an1)*

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>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  $(\text{abs}(\text{atan2}(x,y) - z * \pi / 180) < \pi / 6) * \cos(3 * (\text{atan2}(x,y) - z * \pi / 180))$ .
- 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
y	m
z	1




- 7 In the **Function** text field, type `Pa`.

### *Cylindrical System 2 (sys2)*


In the **Definitions** toolbar, click  **Coordinate Systems** and choose **Cylindrical System**.

## GEOMETRY I



### *Import I (impl)*

- 1 In the **Model Builder** window, expand the **Component I (comp1)>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  **Build Selected**.


### *Rim*

- 1 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 **impl**, select Boundaries 11–19, 24, 26, 28, and 34–36 only.


### *HubAndSpokes*

- 1 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*

- 1 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*

- 1 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 **impI**, select Boundaries 8 and 30 only.

#### *Rotate I (rotI)*


- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Rotate**.
- 2 Select the object **impI** 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*

- 1 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 In the **Geometry** toolbar, click  **Build All**.

### **DEFINITIONS**

#### *Integration I (intopI)*

- 1 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.

- 1 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 1 (lmat1)*

- 1 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 (mat1)	90	th	1
Layer 4	Material: Carbon-Epoxy (mat1)	-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 I (llmatI)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** 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 I (stlmatI)*

- 1 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 I (stlmatI.stllmatI)*

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

### *Layered Material Stack I (stlmatI)*

- 1 In the **Model Builder** window, click **Layered Material Stack I (stlmatI)**.
- 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)

### *Linear Elastic Material, Layered 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Shell (shell)** and choose **Material Models>Linear Elastic Material, Layered**.
- 2 In the **Settings** window for **Linear Elastic Material, Layered**, 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**.
- 5 Select the **Transversely isotropic** check box.

## GLOBAL DEFINITIONS


### *Material: Carbon-Epoxy (mat1)*

- 1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material: Carbon-Epoxy (mat1)**.
- 2 In the **Settings** window for **Material**, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evect1, Evect2}	{134e9, 9.2e9}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{0.28, 0.28}	1	Transversely isotropic
Shear modulus	Gvect1	4.8e9	N/m <sup>2</sup>	Transversely isotropic
Density	rho	1700	kg/m <sup>3</sup>	Basic

## SHELL (SHELL)

### *Fixed Constraint 1*

- 1 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 1*

- 1 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  $-p_{\text{Inflation}}$ .
- 7 Right-click **Face Load 1** and choose **Duplicate**.


*Face Load 2*

- 1 In the **Model Builder** window, click **Face Load 2**.
- 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

$-\text{loadAmpl1} * \text{loadDistr}(X, Y, \text{phiLoad})$	r
0	phi
$0.2 * \text{loadAmpl1} * \text{loadDistr}(X, Y, \text{phiLoad}) * (2 * (Z > 0) - 1)$	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**.
- 10 Click **OK**.

*Global Equations 1 (ODE1)*

- 1 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	$f(u, ut, utt, t)$ (I)	Initial value ( $u_0$ ) (I)	Initial value ( $u_{t0}$ ) (I/s)	Description
loadAmp1	loadAmp1* intop1(loadDistr( X,Y,0)* cos(atan2(X,Y)))- tireLoad	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 I*

1 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  $\Omega$  text field, type omega.

4 From the **Rotational direction** list, choose **Clockwise**.

#### **MESH I**

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

#### *Size*

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


- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, type **Study: Static** in the **Label** text field.

### Step 1: Stationary


- 1 In the **Model Builder** window, under **Study: Static** 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)>Shell (shell)>Rotating Frame 1**.
- 5 Click  **Disable**.
- 6 In the **Home** toolbar, click  **Compute**.

## RESULTS



### Stress (shell)

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

### Surface 1


- 1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click to expand the **Range** section.
- 3 Select the **Manual color range** check box.
- 4 In the **Maximum** text field, type  $2e8$ .
- 5 In the **Stress (shell)** toolbar, click  **Plot**.

## ADD PREDEFINED PLOT



- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study: Static/Solution 1 (sol1)>Shell>Stress, Slice (shell)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the tree, select **Study: Static/Solution 1 (sol1)>Shell>Stress, Through Thickness (shell)**.
- 6 Click **Add Plot** in the window toolbar.
- 7 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

**RESULTS**

*Layered Material Slice I*

- 1 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 **Maximum** text field, type 2e8.
- 5 In the **Stress, Slice (shell)** toolbar, click  **Plot**.

*Through Thickness I*

- 1 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  **Activate 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)

- 10 In the **Stress, Through Thickness (shell)** toolbar, click  **Plot**.

*Stress, Slice (shell)*

In the **Model Builder** window, under **Results** right-click **Stress, Slice (shell)** and choose **Duplicate**.

*Stress, Slice (Hub and Spokes)*


- 1 In the **Model Builder** window, under **Results** click **Stress, Slice (shell) I**.
- 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 I*

- 1 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*

- 1 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  **Plot**.

#### *Stress, Slice (Hub and Spokes)*

In the **Model Builder** window, under **Results** right-click **Stress, Slice (Hub and Spokes)** and choose **Duplicate**.

#### *Stress, Slice (Rim)*

- 1 In the **Model Builder** window, under **Results** click **Stress, Slice (Hub and Spokes) I**.
- 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  **Zoom Extents** button in the **Graphics** toolbar.


### Layered Material Slice I

- 1 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  $0.15 \times 4$ .
- 5 In the **Relative y-separation** text field, type  $0.15 \times 4$ .
- 6 Clear the **Show descriptions** check box.



### Table Annotation I

- 1 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

- 1 In the **Home** toolbar, click  **Add Study** to open the **Add Study** window.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.



### STUDY: EIGENFREQUENCY

- 1 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


- 1 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.
- 4 In the tree, select **Component 1 (comp1)>Shell (shell)>Face Load 1**, **Component 1 (comp1)>Shell (shell)>Face Load 2**, and **Component 1 (comp1)>Shell (shell)>Global Equations 1 (ODE1)**.
- 5 Click  **Disable**.

#### Step 2: Eigenfrequency

- 1 In the **Study** toolbar, click  **Study Steps** and choose **Eigenfrequency>Eigenfrequency**.
- 2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
- 3 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**.

