Model created in COMSOL Multiphysics 6.4

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.

Read more about the Composite Materials Module and the optimization of composite wheel rim designs in the following COMSOL blogs:

- Introduction to the Composite Materials Module.
- Optimizing Composite Wheel Rim Designs with COMSOL Multiphysics®.

Model Definition

The wheel rim geometry is shown in Figure 1.

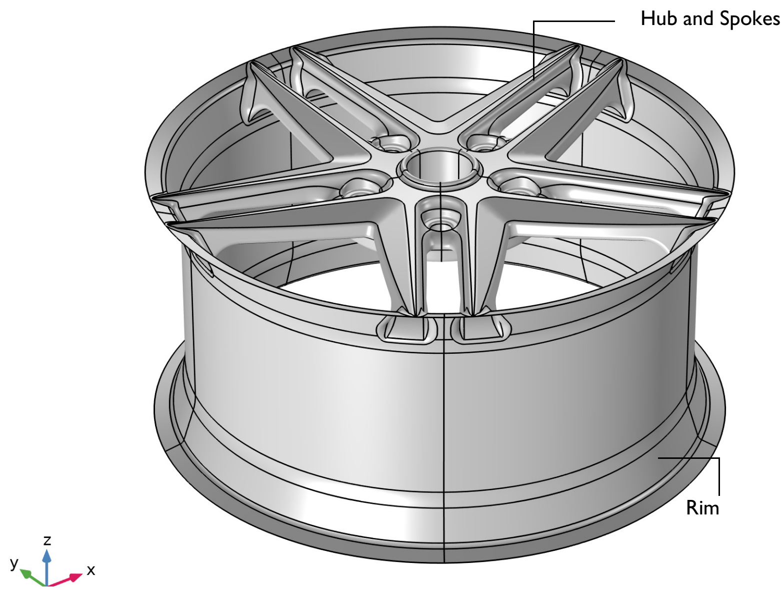

Figure 1: Model geometry of a composite wheel rim.

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)

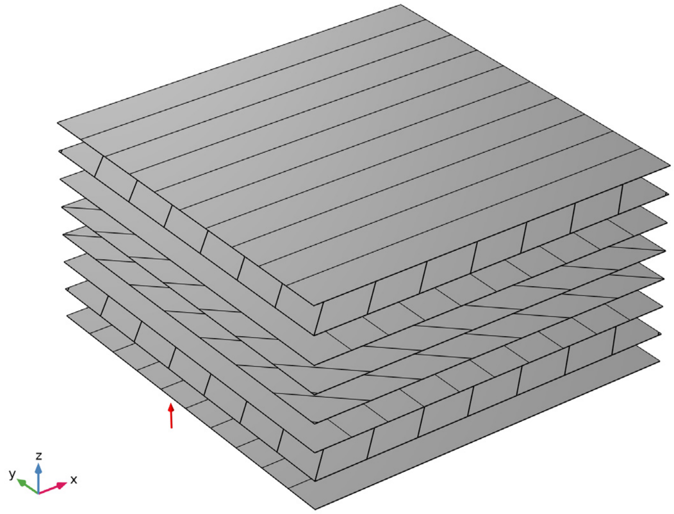

Figure 2: Stacking sequence [0/45/90/-45]$_s$ *in hub-spoke region.*

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.

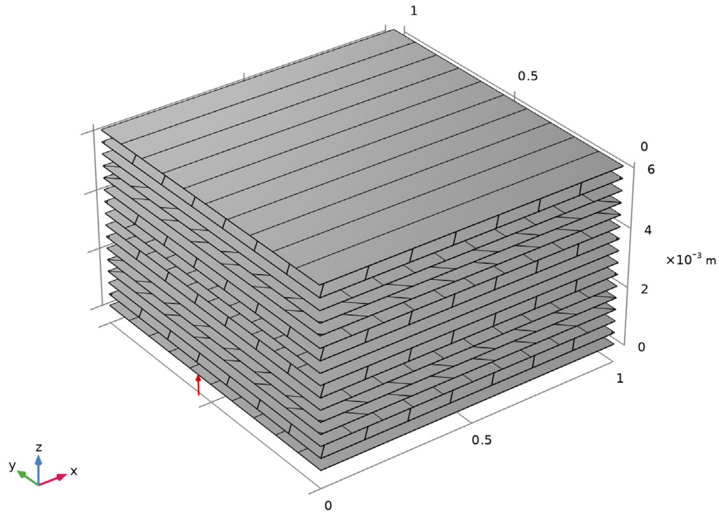

Figure 3: Stacking sequence $[[0/45/90/-45]_s]_2$ *in rim region.*

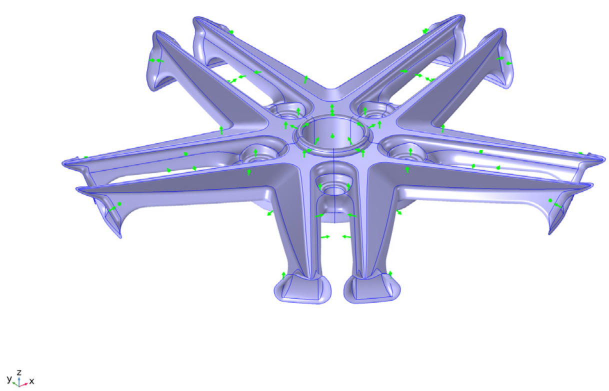

Figure 4: The normal vector on each hub and spokes boundary. This is the direction in which the layer stacking is interpreted from bottom to top.

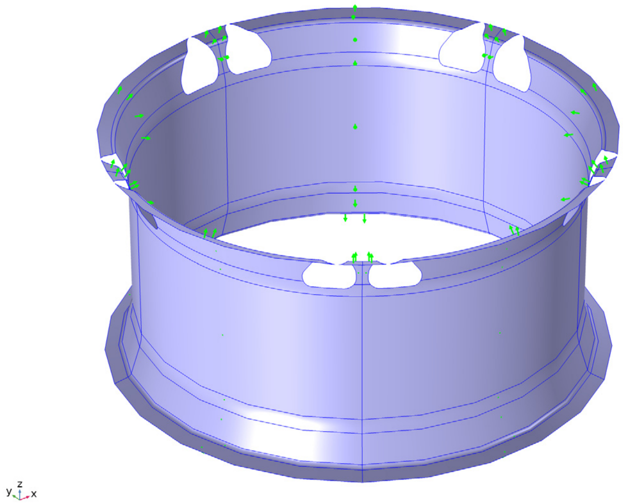

Figure 5: The normal vector on each rim boundary. This is the direction in which the layer stacking is interpreted from bottom to top.

MATERIAL PROPERTIES

Each ply is made up of AS4/APC carbon–thermoplastic composite material. The AS4/APC carbon–thermoplastic is a built-in material in the **Composites** material library. The transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Material property	Value
ρ	1570 kg/m^3
$\{E_{11}, E_{22}\}$	{138, 8.7} GPa
G_{12}	5 GPa
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.28, 0.45}

FINITE ELEMENT MESH

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

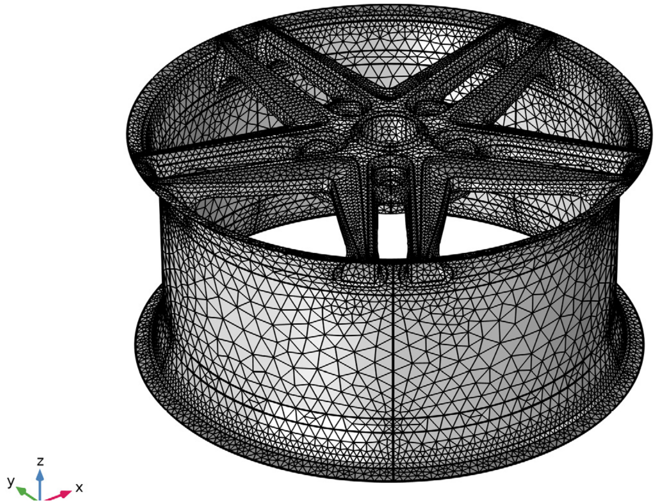

Figure 6: The finite element mesh for the wheel rim.

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 $p = p_0 \cos(3\vartheta)$, 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.

Figure 7 shows the stress in the fiber direction in each layer of 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 stress in the fiber direction 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.

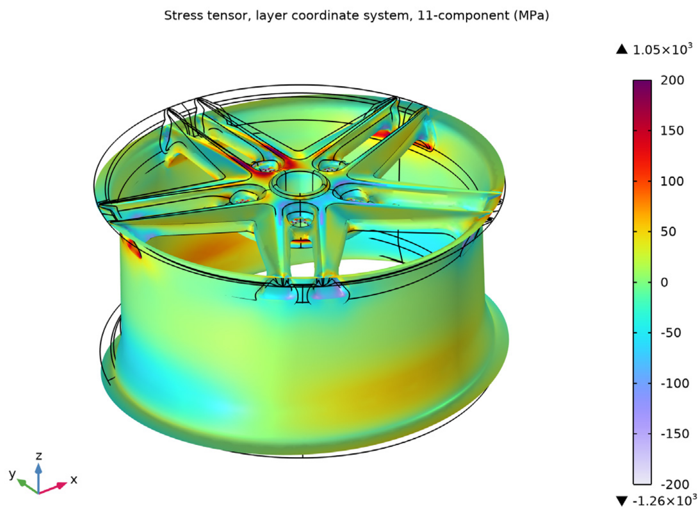

Figure 7: The stress distribution in a composite wheel rim.

The through-thickness variations of the transverse shear stress in first tangent direction of laminate at two points on the rim and spoke region are shown in Figure 10 and Figure 11, respectively. Each ply exhibits different stress levels depending on the fiber orientation.

Figure 12 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.

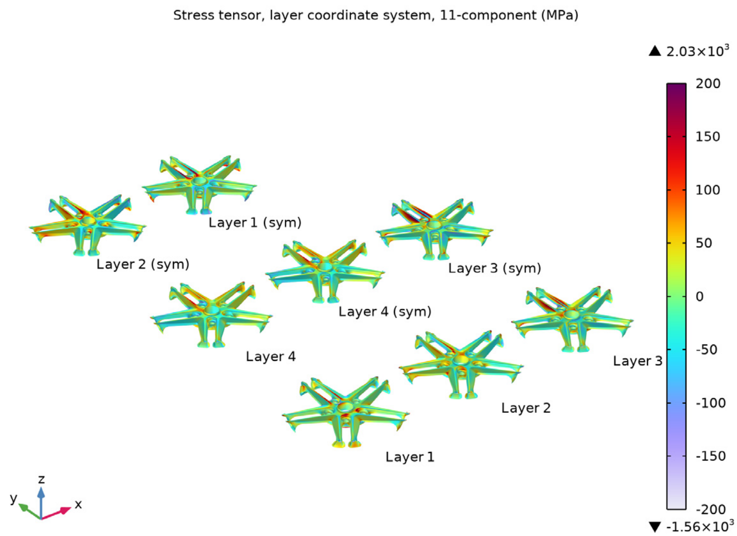

Figure 8: The stress distribution in each layer of hub and spokes region.

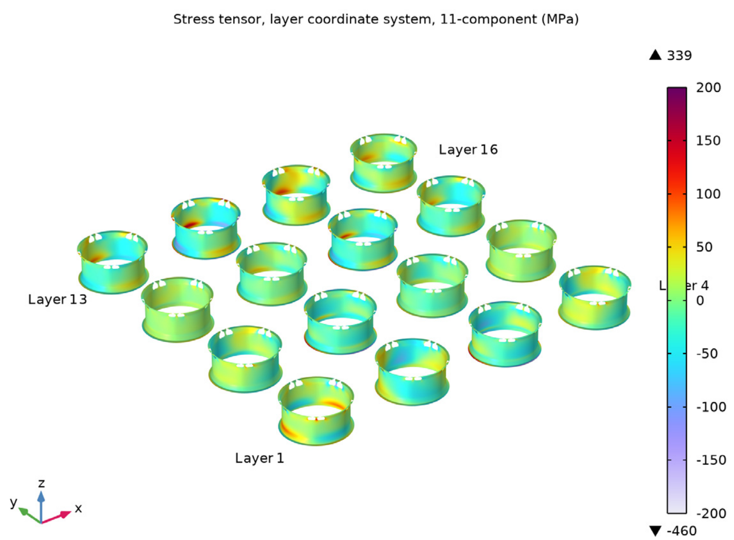

Figure 9: The stress distribution in each layer of rim region.

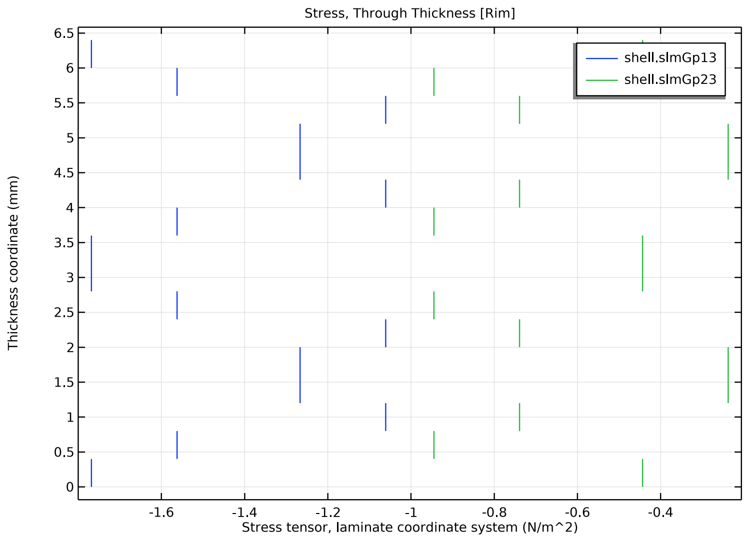

Figure 10: Through-thickness variation of laminate transverse shear stresses for a point on rim.

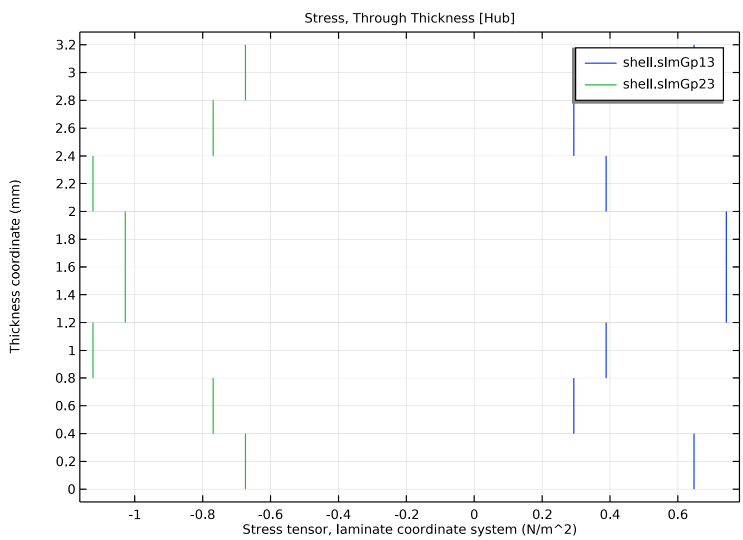

Figure 11: Through-thickness variation of laminate transverse shear stresses for a point on hub.

.

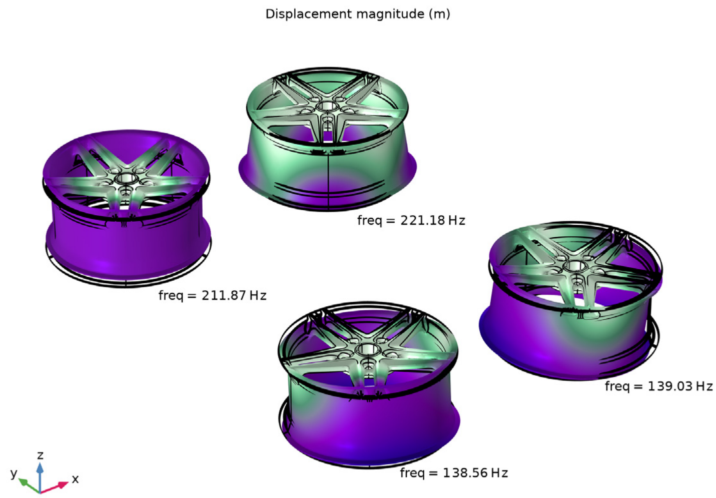

Figure 12: First four Eigenfrequency and mode shapes of the composite wheel rim.

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.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction of the **Layered Shell** or **Shell** physics. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to know it during modeling. There are two ways to achieve this:
  - Using physics symbols: Go to the physics settings, find the **Physics Symbols** section, and select the **Enable physics symbols** checkbox. Then go to the material feature, for

instance, **Linear Elastic Material**, to see the normal direction represented by green arrows in the geometry.

- Using result templates: When a solution dataset is available, use the result template **Thickness and Orientation** to plot the normal direction.

- From a constitutive model point of view, you can either use the *Layerwise (LW)* theory based **Layered Shell** interface, or the *Equivalent Single Layer (ESL)* theory based **Linear Elastic Material, Layered** node in the **Shell** interface. The laminated composite shell presented in the current model is modeled using a **Linear Elastic Material, Layered** node in the **Shell** interface.
- The built-in **Composites** material library contains data for fiber and matrix constituents as well as for unidirectional and bidirectional laminae.
- 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, the 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 `(abs(atan2(x,y)-z*pi/180)<pi/6)*cos(3*(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 1

Import 1 (imp1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** node.

2 Right-click **Geometry 1** and choose **Import**.

3 In the **Settings** window for **Import**, locate the **Source** 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 **imp1**, 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, 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 **imp1**, 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 **imp1**, select Boundaries 8 and 30 only.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
2 Select the object **imp1** 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**.

   Disable the analysis of the geometry as the remaining small geometric details can be kept.

8 In the **Model Builder** window, click **Geometry 1**.
9 In the **Settings** window for **Geometry**, locate the **Cleanup** section.
10 Clear the **Automatic detection of small details** checkbox.
11 In the **Geometry** toolbar, click **Build All**.

DEFINITIONS

Integration 1 (intop1)

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

COMSOL Multiphysics is equipped with built-in material properties for a number of lamina materials. Select the materials needed from the **Composites** material folder in the built-in material library.

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 **Composites** > **Laminae** > **Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%]**.

4 Right-click and choose **Add to Global Materials**.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Layered Material 1 (lmat1)

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 **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	Value	Thickness	Mesh elements
`Layer 1`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	`0`	0 rad	`th`	`1`

4 Click **Add** three times.

5 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 2`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	`45`	0.7854 rad	`th`	`1`
`Layer 3`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	`90`	1.5708 rad	`th`	`1`
`Layer 4`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	`-45`	-0.7854 rad	`th`	`1`

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 (llmat1)

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 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type `0.6`.
7 Locate the **Layered Material Settings** section. Click **Layer Stack Preview** in the upper-right corner of the section.

Layered Material Stack 1 (stlmat1)

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 1 (stlmat1.stllmat1)

1 In the **Model Builder** window, click **Layered Material Link 1 (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)

1 In the **Model Builder** window, click **Layered Material Stack 1 (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)

Linear Elastic Material, Layered 1

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

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 `-pInflation`.

Face Load 2

1 Right-click **Face Load 1** 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 reference area**.

6 Specify the $\mathbf{f}_A$ vector as

-loadAmpl*loadDistr(X,Y,phiLoad)	r

0	phi
0.2*loadAmpl*loadDistr(X,Y,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, select **Physics** > **Equation Contributions** in the tree.

9 In the tree, select the checkbox for the node **Physics** > **Equation 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) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
loadAmpl	loadAmpl* 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, type `force` in the text field.

6 In the tree, select **General** > **Force (N)**.

7 Click **OK**.

Rotating Frame 1

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 ω_r text field, type `omega`.

MESH 1

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 Parameters** section.

3 In the **Maximum element size** text field, type `0.07`.

4 In the **Minimum element size** text field, type `0.006`.

5 In the **Maximum element growth rate** text field, type `1.2`.

6 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** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Shell (shell)** > **Rotating Frame 1**.

5 Click **Disable**.

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Apply**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `shell.s11Gp11`.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Minimum** text field, type `-200`.

6 In the **Maximum** text field, type `200`.

Stress (shell)

1 In the **Model Builder** window, click **Stress (shell)**.

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

3 Select the **Show maximum and minimum values** checkbox.

4 In the **Stress (shell)** toolbar, click **Plot**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

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

3 In the tree, select **Study: Static/Solution 1 (sol1)** > **Shell** > **Stress, Slice (shell)**.

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

5 In the tree, select **Study: Static/Solution 1 (sol1)** > **Shell** > **Stress, Through Thickness (shell)**.

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

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

RESULTS

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Results** > **Stress, Slice (shell)** node, then click **Layered Material Slice 1**.

2 In the **Settings** window for **Layered Material Slice**, click to expand the **Range** section.

3 Select the **Manual color range** checkbox.

4 In the **Minimum** text field, type `-200`.

5 In the **Maximum** text field, type `200`.

Stress, Slice (shell)

1 In the **Model Builder** window, click **Stress, Slice (shell)**.

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

3 Select the **Show maximum and minimum values** checkbox.

4 In the **Stress, Slice (shell)** toolbar, click **Plot**.

Through Thickness 1

1 In the **Model Builder** window, expand the **Results > Stress, Through Thickness (shell)** node, then click **Through Thickness 1**.

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 Point 74 only.

6 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.

Through Thickness 2

1 In the **Model Builder** window, click **Through Thickness 2**.

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 Point 74 only.

6 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.

Stress, Through Thickness [Rim]

1 In the **Model Builder** window, under **Results** click **Stress, Through Thickness (shell)**.

2 In the **Settings** window for **1D Plot Group**, type `Stress, Through Thickness [Rim]` in the **Label** text field.

3 In the **Stress, Through Thickness [Rim]** toolbar, click **Plot**.

Stress, Through Thickness [Rim] 1

Right-click **Stress, Through Thickness [Rim]** and choose **Duplicate**.

Through Thickness 1

1 In the **Model Builder** window, expand the **Stress, Through Thickness [Rim] 1** node, then click **Through Thickness 1**.

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 Point 95 only.

Through Thickness 2

1 In the **Model Builder** window, click **Through Thickness 2**.

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 Point 95 only.

Stress, Through Thickness [Hub]

1 In the **Model Builder** window, under **Results** click **Stress, Through Thickness [Rim] 1**.

2 In the **Settings** window for **1D Plot Group**, type `Stress, Through Thickness [Hub]` in the **Label** text field.

3 In the **Stress, Through Thickness [Hub]** toolbar, click **Plot**.

Stress, Slice (Hub and Spokes)

1 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** checkbox.

6 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Slice (Hub and Spokes)** node, then click **Layered Material Slice 1**.

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** checkbox.

7 In the **Relative separation** text field, type `0.2*2`.

Deformation

1 In the **Model Builder** window, expand the **Layered Material Slice 1** node, then click **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Stress, Slice (Hub and Spokes)

1 In the **Model Builder** window, under **Results** click **Stress, Slice (Hub and Spokes)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
3 Clear the **Plot dataset edges** checkbox.
4 In the **Stress, Slice (Hub and Spokes)** toolbar, click **Plot**.

Stress, Slice (Rim)

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

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Slice (Rim)** node, then click **Layered Material Slice 1**.
2 In the **Settings** window for **Layered Material Slice**, locate the **Layout** section.
3 In the **Relative x-separation** text field, type `0.15*4`.
4 In the **Relative y-separation** text field, type `0.15*4`.
5 Clear the **Show descriptions** checkbox.

Table Annotation 1

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** checkbox.

Stress, Slice (Rim)

1 In the **Model Builder** window, click **Stress, Slice (Rim)**.

2 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 the **Add Study** button in the window toolbar.

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

STUDY: EIGENFREQUENCY

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** checkbox.

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 **More Study Steps** and choose **Eigenfrequency** > **Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** checkbox.

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

RESULTS

Mode Shape (shell)

1 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 Clear the **Parameter indicator** text field.

4 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

5 In the **Relative row padding** text field, type `1.4`.

6 In the **Relative column padding** text field, type `0.5`.

Solution Array 1

1 In the **Model Builder** window, expand the **Mode Shape (shell)** node.

2 Right-click **Surface 1** and choose **Solution Array**.

3 In the **Settings** window for **Solution Array**, locate the **Data** section.

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

5 In the **Eigenfrequency indices (1-6)** text field, type `range(1,1,4)`.

6 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Annotation 1

1 In the **Model Builder** window, right-click **Mode Shape (shell)** and choose **Annotation**.

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

3 In the **Text** text field, type `freq = eval(real(freq),Hz,5) Hz`.

4 From the **Geometry level** list, choose **Global**.

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 Locate the **Position** section. In the **y** text field, type `-0.1`.

7 In the **z** text field, type `-0.2`.

8 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Solution Array 1

In the **Model Builder** window, under **Results** > **Mode Shape (shell)** > **Surface 1** right-click **Solution Array 1** and choose **Copy**.

Solution Array 1

In the **Model Builder** window, right-click **Annotation 1** and choose **Paste Solution Array**.

Mode Shape (shell)

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

2 From the **View** list, choose **New view**.

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

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

5 Click the **Show Grid** button in the **Graphics** toolbar.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

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

3 In the tree, select **Study: Eigenfrequency/Solution 2 (sol2)** > **Shell** > **Shell Geometry (shell)**.

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

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

RESULTS

Shell Geometry (shell)

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

2 From the **Color** list, choose **Black**.

3 In the **Shell Geometry (shell)** toolbar, click **Plot**.