Tire Inflation

Introduction

Tires play a fundamental structural role in vehicle dynamics by transmitting loads to the ground, thus making it possible for traction and brake to develop. At the same time, tires must contribute to limit the effect of road irregularity, concurring to the overall vibration isolation capability of the vehicle. These two functions require modern tires to have substantial structural loading capability, while at the same time being compliant enough to provide better comfort with respect to a solid wheel. A tire is thus a complicated composite made of various types of soft rubber locally reinforced by stiff cords usually made by steel and nylon.

Tire simulation requires the use of appropriate nonlinear anisotropic constitutive models to describe the rubber behavior with the inclusion of the effect of the cord reinforcements. This model showcases how to use fibers in thin layers to model thin anisotropic composites embedded in a solid without explicitly drawing either the layer of material or the reinforcing fibers. In particular, this is used to model steel cords in tire belts that are used to provide structural support to the tire below the treads. Moreover, a curvilinear coordinate system is used to define the anisotropic material properties of the carcass ply.

Model Definition

Both the geometry of the tire and the applied loads during inflation show axial symmetry. For this reason, you do not need to model the entire three dimensional tire, but a 2D axisymmetric model can be used to save computational time. Moreover, a plane of symmetry can be identified for the tire cross section. Figure 1 illustrates a simplified geometry of half of the cross section of a tire, along with the cross section of the rim in the locations where it enters in contact with the tire.

Note that when exploiting symmetries for reducing the computational cost in simulations care must be taken to check that the material properties show the same symmetry as exhibited by loads and geometry.

In the case of tires, the orientation of cords in the reinforcing plies usually induces displacements that have non zero components in the azimuthal direction, and that are not symmetric with respect to the plane of symmetry of the geometry. COMSOL Multiphysics provides a specific formulation for the 2D axisymmetric problem that allows to include the azimuthal “twist” without the need to switch to a full 3D model.

For what concerns the lack of symmetry with respect to the midplane of the cross section, it can be shown that by using an antisymmetry condition for the circumferential displacement on the symmetry plane of the geometry, the difference in the solution obtained with respect to the one computed with a full cross section is very small.

In Figure 1 different colors have been used to highlight different domains, based on the material properties used.

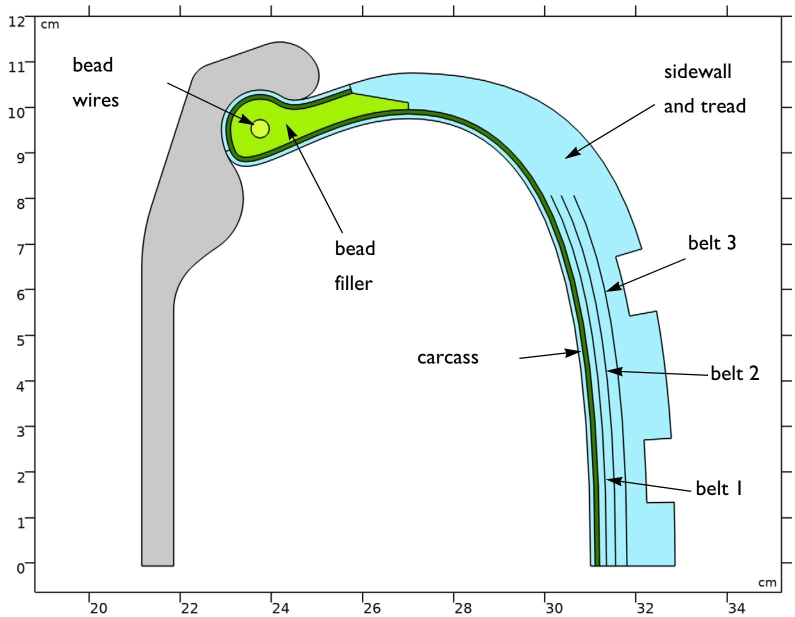

Figure 1: Geometry of the tire and rim cross sections.

The following modeling assumptions have been made:

- Sidewall and tread are assumed to be made by the same material, modeled with the Yeoh hyperelastic constitutive law
- Bead wires are modeled as a single uniform steel domain
- Bead filler can be also modeled with the Yeoh hyperelastic material modeled
- The carcass is modeled using a linear elastic transversely isotropic material model, that captures in a homogenized way the nylon and steel reinforcements
- The three belts are modeled as thin hyperelastic layers made by the same rubber as the sidewall and tread, with embedded steel cords
- The rim is considered to be rigid.

The material properties for each material domain are collected in Table 1 (Ref. 1 and Ref. 2).

TABLE 1: MATERIAL PROPERTIES.

Material properties	**Sidewall and tread**	**Bead**	**Carcass**
c_1	1 MPa	1.6 MPa	-
c_2	-0.3 MPa	-1.4 MPa	-
c_3	0.1 MPa	0.9 MPa	-
K	100 MPa	100 MPa	-
E_1	-	-	7 GPa
E_2	-	-	25 MPa
ν_1	-	-	0.45
ν_2	-	-	0.4
G	-	-	4 MPa

Table 2 collects the orientation of the cords in each belt with respect to the azimuthal direction. The orientation of the cords, along with the first principal direction of the transversely isotropic material used for the carcass are represented in Figure 2.

The tire is inflated to reach an inner pressure of 2 bar.

TABLE 2: ORIENTATION OF CORDS.

Belt 1	**Belt 2**	**Belt 3**
+20°	-20°	0°

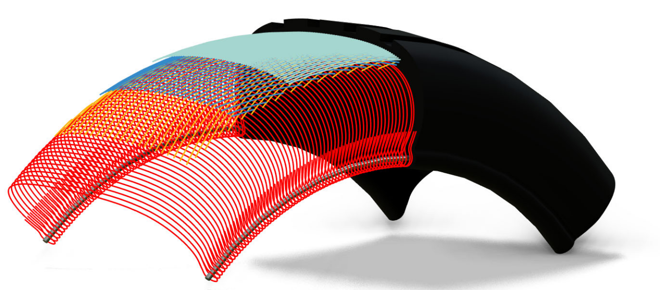

Figure 2: Cords orientation.

Results and Discussion

Figure 3 shows the twist displacement in the tire, and Figure 4 shows the von Mises stress in the cords and in the bead wires.

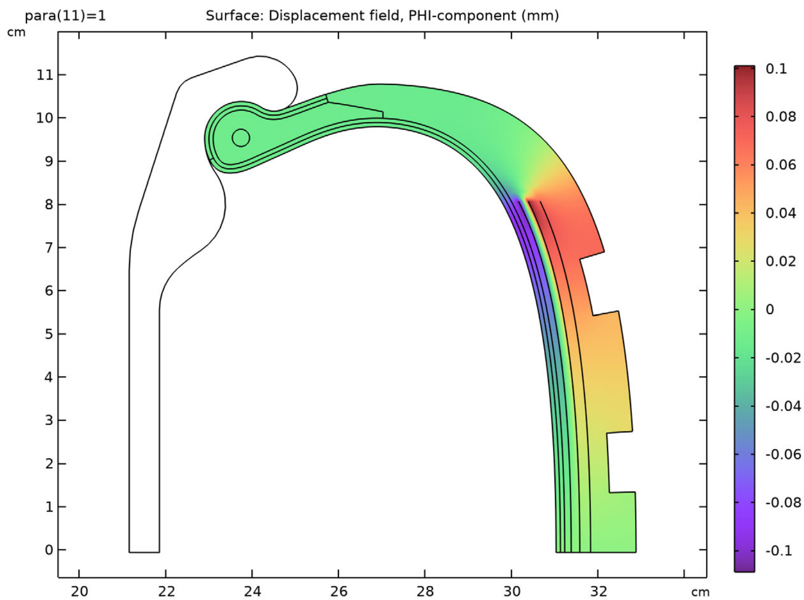

Figure 3: Azimuthal displacement.

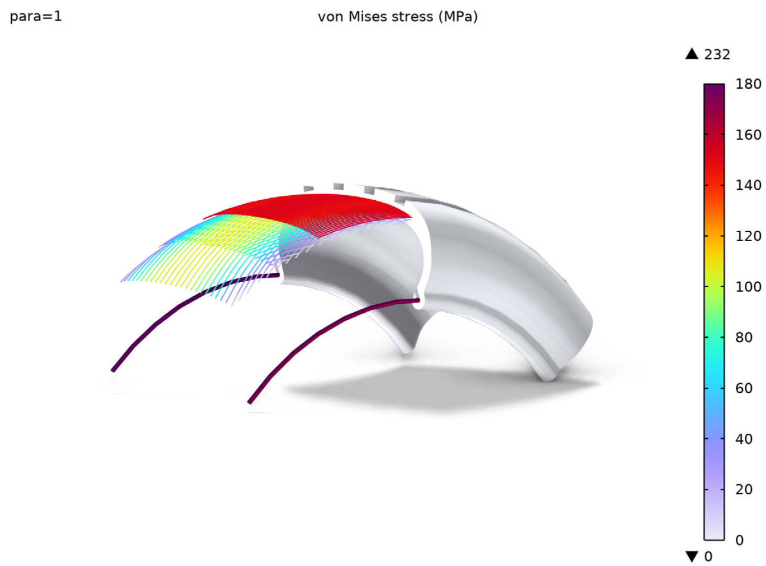

Figure 4: The von Mises stress at the end of inflation.

Notes About the COMSOL Implementation

- You find the **Include circumferential displacement** check box in the **Axial Symmetry Approximation** section in the settings window of the Solid Mechanics interface. This allows you to enable the 2D axisymmetric formulation that includes the twist degree of freedom.
- You can add a **Initial Stress and Strain** node under **Linear Elastic Material** to model the prestress in the bead wires, resulting after the mounting of the tire on the rim.
- You can use a **Curvilinear Coordinates** interface to compute the direction of anisotropy in the carcass.
- To model the belts without explicitly drawing their thickness, you can simply add a **Thin Layer** and add a **Hyperelastic Material** to it. Use the **Solid** approximation to create a slit for the displacement variable at the selected boundary.
- Add three **Fiber** nodes to the **Hyperelastic Material** under the **Thin Layer** in order to model the three different orientations of the cords. You can directly specify the orientation

with respect to the boundary coordinate system without further computations. Choose a **Linear elastic** material model for the cords, to model steel.

- You find the **Antisymmetry** option under the **Circumferential Condition** section when you add the **Symmetry Plane**.

References

1. T. Király, P. Primusz and C. Tóth, "Simulation of Static Tyre-Pavement Interaction Using Two FE Models of Different Complexity," *Appl. Sci.*, vol. 12, no. 5, p. 2388, 2022.

2. H.S. Aldhufairi, O. Olatunbosun, and K. Essa, "Determination of a Tyre's Rolling Resistance Using Parallel Rheological Framework," *SAE Technical Paper*, no. 2019-01-5069, 2019.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/tire_inflation`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

First, import the geometry of the tire cross section and the geometry of the rim.

TIRE SECTION

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts>2D Part**.

2 In the **Settings** window for **Part**, type `Tire Section` in the **Label** text field.

Import 1 (imp1)

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

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `tire_inflation.mphbin`.

5 In the **Home** toolbar, click **Build All**.

TIRE SECTION

In the **Model Builder** window, collapse the **Global Definitions>Geometry Parts>Tire Section** node.

RIM

1 In the **Model Builder** window, under **Global Definitions** right-click **Geometry Parts** and choose **3D Part**.

2 In the **Settings** window for **Part**, type `Rim` in the **Label** text field.

3 Locate the **Advanced** section. From the **Geometry representation** list, choose **CAD kernel**.

4 In the **Home** toolbar, click **Import**.

1 In the **Settings** window for **Import**, locate the **Import** section.

2 Click **Browse**.

3 Browse to the model's Application Libraries folder and double-click the file `wheel_rim.x_b`.

4 In the **Home** toolbar, click **Build All**.

RIM

In the **Model Builder** window, collapse the **Global Definitions>Geometry Parts>Rim** node.

Generate the 2D axisymmetric geometry.

2D AXISYMMETRIC [TIRE]

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

2 In the **Settings** window for **Component**, type `2D Axisymmetric [Tire]` in the **Label** text field.

GEOMETRY 1

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)** click **Geometry 1**.

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

3 From the **Length unit** list, choose **cm**.

Tire Section 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Tire Section**.

2 In the **Settings** window for **Part Instance**, click **Build Selected**.

In order to add the rim to the 2D axisymmetric component, you need to create a cross section of the 3D geometry.

ADD COMPONENT

Right-click **Tire Section 1 (pi1)** and choose **3D**.

3D [RIM]

In the **Settings** window for **Component**, type `3D [Rim]` in the **Label** text field.

GEOMETRY 2

1 In the **Model Builder** window, under **3D [Rim] (comp2)** click **Geometry 2**.

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

3 From the **Length unit** list, choose **cm**.

4 Locate the **Advanced** section. From the **Geometry representation** list, choose **CAD kernel**.

Rim 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Rim**.

2 In the **Settings** window for **Part Instance**, click **Build Selected**.

Set the plane of the cross section.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 Click **Build Selected**.

GEOMETRY 1

In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)** click **Geometry 1**.

Cross Section 1 (cro1)

1. In the **Geometry** toolbar, click **Cross Section**.
2. In the **Settings** window for **Cross Section**, click **Build Selected**.
3. Click the **Zoom Extents** button in the **Graphics** toolbar.

Delete Entities 1 (del1)

1. In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
2. In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. On the object **cro1**, select Domains 1–3 only.
5. Click **Build Selected**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

Only half of the domain will be included in the study by adopting a symmetry boundary condition. Modify the geometry accordingly.

Line Segment 1 (ls1)

1. In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2. On the object **pi1**, select Point 80 only.
3. In the **Settings** window for **Line Segment**, locate the **Endpoint** section.
4. From the **Specify** list, choose **Coordinates**.
5. In the **r** text field, type `20`.
6. In the **z** text field, type `-7E-2`.
7. Click **Build Selected**.

Partition Objects 1 (par1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.
2. Select the objects **del1** and **pi1** only.
3. In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
4. Click to select the **Activate Selection** toggle button for **Tool objects**.
5. Select the object **ls1** only.
6. Click **Build Selected**.

Delete Entities 2 (del2)

1. Right-click **Geometry 1** and choose **Delete Entities**.
2. In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Click the **Select Box** button in the **Graphics** toolbar.
5. On the object **par1(1)**, select Domain 1 only.
6. On the object **par1(2)**, select Domains 1, 2, 6, 8, and 10 only.
7. Click **Build Selected**.
8. Click the **Zoom Extents** button in the **Graphics** toolbar.

Add now mesh control edges, needed later on in order to mesh properly the component.

Import 1 (imp1)

1. In the **Geometry** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `tire_inflation_meshcontrol.mphbin`.
5. Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
6. From the **Show in physics** list, choose **Boundary selection**.
7. Click **Build Selected**.
8. Click the **Zoom Extents** button in the **Graphics** toolbar.

Union 1 (uni1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2. Select the objects **del2(2)** and **imp1** only.
3. In the **Settings** window for **Union**, click **Build Selected**.

An assembly is needed between the rim and the tire, in order to set properly the contact between them.

Form Union (fin)

1. In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)>Geometry 1** click **Form Union (fin)**.
2. In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
3. From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Click **Build Selected**.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 In the **Settings** window for **Mesh Control Edges**, locate the **Input** section.

3 From the **Edges to include** list, choose **Import 1**.

4 Click **Build Selected**.

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

Set the contact pair between rim and tire.

DEFINITIONS (COMP1)

Contact Pair 1 (p1)

1 In the **Model Builder** window, expand the **2D Axisymmetric [Tire] (comp1)>Definitions** node.

2 Right-click **2D Axisymmetric [Tire] (comp1)>Definitions** and choose **Pairs>Contact Pair**.

3 Select Boundaries 7, 8, and 10 only.

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

5 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

6 Click to select the **Activate Selection** toggle button.

7 Select Boundaries 30, 31, 44, and 46 only.

Add material properties.

GLOBAL DEFINITIONS

Steel [Bead Wires]

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Steel [Bead Wires]` in the **Label** text field.

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Young's Modulus**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties>Poisson's Ratio**.

6 Click **Add to Material**.

7 In the **Material properties** tree, select **Basic Properties>Density**.

8 Click **Add to Material**.

9 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`210[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`7800[kg/m^3]`	kg/m³	Basic

10 Right-click **Steel [Bead Wires]** and choose **Duplicate**.

Steel [Cords]

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Steel [Bead Wires] 1 (mat2)**.

2 In the **Settings** window for **Material**, type `Steel [Cords]` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`170[GPa]`	Pa	Basic

Rubber [Sidewall and Tread]

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Rubber [Sidewall and Tread]` in the **Label** text field.

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Hyperelastic Material>Yeoh**.

4 Click **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Model parameters	c1YE	`1[MPa]`	Pa	Yeoh
Model parameters	c2YE	`-0.3[MPa]`	Pa	Yeoh
Model parameters	c3YE	`0.1[MPa]`	Pa	Yeoh

6 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Bulk Modulus and Shear Modulus>Bulk modulus (K)**.

7 Click **Add to Material**.

8 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk modulus	K	`100[MPa]`	N/m^2	Bulk modulus and shear modulus

9 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Density**.

10 Click **Add to Material**.

11 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1100[kg/m^3]`	kg/m^3	Basic

12 Right-click **Rubber [Sidewall and Tread]** and choose **Duplicate**.

Rubber [Bead]

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Rubber [Sidewall and Tread] 1 (mat4)**.

2 In the **Settings** window for **Material**, type `Rubber [Bead]` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Model parameters	c1YE	`1.6[MPa]`	Pa	Yeoh
Model parameters	c2YE	`-1.4[MPa]`	Pa	Yeoh
Model parameters	c3YE	`0.9[MPa]`	Pa	Yeoh

Reinforced Rubber [Carcass]

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Reinforced Rubber [Carcass]` in the **Label** text field.

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Transversely Isotropic**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties>Density**.

6 Click **Add to Material**.

7 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1750[kg/m^3]`	kg/m³	Basic
Young's modulus	{Evect1, Evect2}	`{7[GPa], 25[MPa]}`	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	`{0.45, 0.4}`	1	Transversely isotropic
Shear modulus	Gvect1	`4[MPa]`	N/m²	Transversely isotropic

Assign material properties to domains.

MATERIALS

Material Link: Steel [Bead Wires]

1. In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)** right-click **Materials** and choose **More Materials>Material Link**.
2. In the **Settings** window for **Material Link**, type `Material Link: Steel [Bead Wires]` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. Click **Clear Selection**.
4. Select Domain 7 only.
5. Click **Create Selection**.
6. In the **Create Selection** dialog box, type `Bead Wires` in the **Selection name** text field.
7. Click **OK**.
8. In the **Settings** window for **Material Link**, click to expand the **Appearance** section.
9. From the **Material type** list, choose **Steel**.

Material Link: Rubber [Sidewall and Tread]

1. Right-click **Materials** and choose **More Materials>Material Link**.
2. In the **Settings** window for **Material Link**, locate the **Link Settings** section.
3. From the **Material** list, choose **Rubber [Sidewall and Tread] (mat3)**.
4. In the **Label** text field, type `Material Link: Rubber [Sidewall and Tread]`.
5. Select Domains 2, 3, and 5 only.
6. Locate the **Geometric Entity Selection** section. Click **Create Selection**.

7 In the **Create Selection** dialog box, type `Sidewall and Tread` in the **Selection name** text field.

8 Click **OK**.

9 In the **Settings** window for **Material Link**, click to expand the **Appearance** section.

10 From the **Material type** list, choose **Rubber**.

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

Material Link: Rubber [Bead]

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, type `Material Link: Rubber [Bead]` in the **Label** text field.

3 Select Domain 6 only.

4 Locate the **Link Settings** section. From the **Material** list, choose **Rubber [Bead] (mat4)**.

5 Locate the **Geometric Entity Selection** section. Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Bead Rubber` in the **Selection name** text field.

7 Click **OK**.

8 In the **Settings** window for **Material Link**, click to expand the **Appearance** section.

9 From the **Material type** list, choose **Rubber**.

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

Material Link: Reinforced Rubber [Carcass]

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, type `Material Link: Reinforced Rubber [Carcass]` in the **Label** text field.

3 Select Domain 4 only.

4 Locate the **Link Settings** section. From the **Material** list, choose **Reinforced Rubber [Carcass] (mat5)**.

5 Locate the **Geometric Entity Selection** section. Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Carcass` in the **Selection name** text field.

7 Click **OK**.

8 In the **Settings** window for **Material Link**, click to expand the **Appearance** section.

9 From the **Color** list, choose **Gray**.

DEFINITIONS (COMP1)

Hyperelastic Domains

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Hyperelastic Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Sidewall and Tread** and **Bead Rubber**.

5 Click **OK**.

Exclude the rim from **Solid Mechanics**, since it is modeled as a rigid boundary.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 2–7 only.

Assign the hyperelastic material model to domains made of rubber.

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Hyperelastic Domains**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Yeoh**.

5 From the **Compressibility** list, choose **Compressible, uncoupled**.

Assign a transversely isotropic linear elastic material model to the tire carcass.

Linear Elastic Material [Carcass]

1 In the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 In the **Settings** window for **Linear Elastic Material**, type `Linear Elastic Material [Carcass]` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Carcass**.

4 Locate the **Linear Elastic Material** section. From the **Material symmetry** list, choose **Orthotropic**.

5 Select the **Transversely isotropic** check box.

Linear Elastic Material [Bead Wires]

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.
2 In the **Settings** window for **Linear Elastic Material**, type `Linear Elastic Material [Bead Wires]` in the **Label** text field.

Add a **Curvilinear Coordinates** physics interface in order to compute principal directions of anisotropy for the reinforced rubber.

ADD PHYSICS

1 In the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Mathematics>Curvilinear Coordinates (cc)**.
4 Click **Add to 2D Axisymmetric [Tire]** in the window toolbar.
5 In the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

CURVILINEAR COORDINATES (CC)

1 In the **Settings** window for **Curvilinear Coordinates**, locate the **Domain Selection** section.
2 From the **Selection** list, choose **Carcass**.
3 Locate the **Settings** section. Select the **Create base vector system** check box.

Diffusion Method 1

Right-click **2D Axisymmetric [Tire] (comp1)>Curvilinear Coordinates (cc)** and choose **Diffusion Method**.

Inlet 1

1 In the **Physics** toolbar, click **Attributes** and choose **Inlet**.
2 Select Boundary 21 only.

Diffusion Method 1

In the **Model Builder** window, click **Diffusion Method 1**.

Outlet 1

1 In the **Physics** toolbar, click **Attributes** and choose **Outlet**.
2 Select Boundary 17 only.

SOLID MECHANICS (SOLID)

Linear Elastic Material [Carcass]

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)> Solid Mechanics (solid)** click **Linear Elastic Material [Carcass]**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Curvilinear System (cc) (cc_cs)**.

Define geometric parameters of the belts' cord reinforcements.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`dcord_belts`	`0.5[mm]`	5E-4 m	`Diameter of belt cords`
`spcord_belts`	`1.16[mm]`	0.00116 m	`Spacing between belt cords`
`alpha_belt`	`70[deg]`	1.2217 rad	`Angle of belt cords`

Add the belts.

SOLID MECHANICS (SOLID)

Thin Layer 1

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

2 Select Boundaries 57–59 only.

3 In the **Settings** window for **Thin Layer**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Belts` in the **Selection name** text field.

6 Click **OK**.

7 In the **Settings** window for **Thin Layer**, locate the **Boundary Properties** section.

8 In the L_{th} text field, type `dcord_belts`.

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Attributes** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Belts**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Yeoh**.

5 From the **Compressibility** list, choose **Compressible, uncoupled**.

MATERIALS

Material Link: Rubber [Sidewall and Tread] (matlnk2)

In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)>Materials** right-click **Material Link: Rubber [Sidewall and Tread] (matlnk2)** and choose **Duplicate**.

Material Link: Rubber [Belts]

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)>Materials** click **Material Link: Rubber [Sidewall and Tread] 1 (matlnk5)**.

2 In the **Settings** window for **Material Link**, type `Material Link: Rubber [Belts]` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Belts**.

SOLID MECHANICS (SOLID)

Hyperelastic Material 1

In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)> Solid Mechanics (solid)>Thin Layer 1** click **Hyperelastic Material 1**.

Fiber [Cords, Belt 1]

1 In the **Physics** toolbar, click **Attributes** and choose **Fiber**.

2 In the **Settings** window for **Fiber**, type `Fiber [Cords, Belt 1]` in the **Label** text field.

3 Locate the **Fiber Model** section. From the **Material model** list, choose **Linear elastic**.

4 From the **Material** list, choose **Steel [Cords] (mat2)**.

5 Locate the **Orientation** section. From the **a** list, choose **User defined**.

6 Specify the **Direction** vector as

`cos(alpha_belt)`	t1
`sin(alpha_belt)`	to

7 In the d_{fiber} text field, type `dcord_belts`.

8 In the s_{fiber} text field, type `spcord_belts`.

9 Locate the **Boundary Selection** section. Click **Clear Selection**.

10 Select Boundary 57 only.

11 Click **Create Selection**.

12 In the **Create Selection** dialog box, type `Belt 1` in the **Selection name** text field.

13 Click **OK**.

14 Right-click **Fiber [Cords, Belt 1]** and choose **Duplicate**.

Fiber [Cords, Belt 2]

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)> Solid Mechanics (solid)>Thin Layer 1>Hyperelastic Material 1** click **Fiber [Cords, Belt 1] 1**.

2 In the **Settings** window for **Fiber**, type `Fiber [Cords, Belt 2]` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Select Boundary 58 only.

5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Belt 2` in the **Selection name** text field.

7 Click **OK**.

8 In the **Settings** window for **Fiber**, locate the **Orientation** section.

9 Specify the **Direction** vector as

`-cos(alpha_belt)`	t1

10 Right-click **Fiber [Cords, Belt 2]** and choose **Duplicate**.

Fiber [Cords, Belt 3]

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)> Solid Mechanics (solid)>Thin Layer 1>Hyperelastic Material 1** click **Fiber [Cords, Belt 2] 1**.

2 In the **Settings** window for **Fiber**, type `Fiber [Cords, Belt 3]` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Select Boundary 59 only.

5 Click **Create Selection**.

6 In the **Create Selection** dialog box, type `Belt 3` in the **Selection name** text field.

7 Click **OK**.

8 In the **Settings** window for **Fiber**, locate the **Orientation** section.

9 From the **a** list, choose **Second axis**.

Thin Layer 1

In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)>Solid Mechanics (solid)** click **Thin Layer 1**.

Roller 1

1 In the **Physics** toolbar, click **Attributes** and choose **Roller**.

2 Select Points 46, 48, and 49 only.

Because of the orientation of the fibers, the circumferential displacement will not be zero.

3 In the **Model Builder** window, click **Solid Mechanics (solid)**.

4 In the **Settings** window for **Solid Mechanics**, locate the **Axial Symmetry Approximation** section.

5 Select the **Include circumferential displacement** check box.

Set the prestress in the bead wires.

Linear Elastic Material [Bead Wires]

In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material [Bead Wires]**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the S_0 table, enter the following settings:

0	300[MPa]	0

Contact 1

1 In the **Model Builder** window, under **2D Axisymmetric [Tire] (comp1)> Solid Mechanics (solid)** click **Contact 1**.

2 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.

3 From the **Penalty factor control** list, choose **Manual tuning**.

4 In the f_p text field, type 2.

Set up the inner pressure.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`p0`	`200[kPa]`	2E5 Pa	`Inflation pressure`
`para`	`0`	0	`Continuation parameter for inflation`

SOLID MECHANICS (SOLID)

Inner Pressure

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

2 In the **Settings** window for **Boundary Load**, type `Inner Pressure` in the **Label** text field.

3 Select Boundaries 30, 45, and 48 only.

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the p text field, type `p0*para`.

Set up the symmetry plane.

Symmetry Plane 1

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

2 Click the **Select Box** button in the **Graphics** toolbar.

3 Select Boundaries 20–23, 25, and 26 only.

4 In the **Settings** window for **Symmetry Plane**, locate the **Circumferential Condition** section.

5 From the list, choose **Antisymmetry**.

Set up the mesh.

MESH 1

Free Quad 1

1 In the **Mesh** toolbar, click **Free Quad**.

2 In the **Settings** window for **Free Quad**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–4, 8, and 19 only.

5 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution 1

1 Right-click **Free Quad 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `1`.

4 Select Boundaries 15, 95, and 96 only.

Distribution 2

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

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

3 From the **Distribution type** list, choose **Predefined**.

4 Select Boundaries 55, 56, and 70 only.

5 In the **Number of elements** text field, type `6`.

6 In the **Element ratio** text field, type `1.2`.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 3

1 Right-click **Free Quad 1** and choose **Distribution**.

2 Select Boundary 45 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `15`.

6 In the **Element ratio** text field, type `1.4`.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

9 Right-click **Distribution 3** and choose **Duplicate**.

Distribution 4

1 In the **Model Builder** window, click **Distribution 4**.

2 Select Boundary 71 only.

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

4 In the **Element ratio** text field, type `1.6`.

5 Clear the **Reverse direction** check box.

6 Right-click **Distribution 4** and choose **Duplicate**.

Distribution 5

1 In the **Model Builder** window, click **Distribution 5**.

2 Select Boundary 72 only.

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

4 In the **Element ratio** text field, type `1.8`.

Distribution 6

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

2 Select Boundary 63 only.

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

4 In the **Number of elements** text field, type `6`.

5 Select Boundaries 42–44, 51, 52, and 63 only.

Distribution 7

1 Right-click **Free Quad 1** and choose **Distribution**.

2 Select Boundaries 34 and 37 only.

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

4 In the **Number of elements** text field, type `2`.

Distribution 8

1 Right-click **Free Quad 1** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Element ratio** text field, type `1.5`.

5 Select Boundary 30 only.

Distribution 9

1 Right-click **Free Quad 1** and choose **Distribution**.

2 Select Boundaries 32 and 35 only.

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

4 In the **Number of elements** text field, type `3`.

Distribution 10

1 Right-click **Free Quad 1** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type 10.

5 In the **Element ratio** text field, type 1.5.

6 From the **Growth rate** list, choose **Exponential**.

7 Select the **Symmetric distribution** check box.

8 Select Boundaries 31, 33, and 36 only.

Distribution 11

1 Right-click **Free Quad 1** and choose **Distribution**.

2 Select Boundaries 46, 53, and 54 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 10.

6 In the **Element ratio** text field, type 5.

7 From the **Growth rate** list, choose **Exponential**.

8 Click **Build All**.

Refine 1

1 In the **Mesh** toolbar, click **Modify** and choose **Refine**.

2 In the **Settings** window for **Refine**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–4 only.

5 Click **Build Selected**.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 5, 9-18, 21-37, 39-45 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.

8 Clear the **Smooth across removed control entities** check box.

Distribution 1

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

2 Select Boundaries 24, 26, 27, 50, and 59–62 only.

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

4 In the **Number of elements** text field, type 2.

Distribution 2

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

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

3 In the **Number of elements** text field, type 3.

4 Select Boundary 49 only.

Distribution 3

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

2 Select Boundary 47 only.

Distribution 4

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

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

3 In the **Number of elements** text field, type 1.

4 Select Boundaries 101–103 only.

Distribution 5

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

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

3 In the **Number of elements** text field, type 12.

4 Select Boundary 131 only.

Distribution 6

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

2 Select Boundary 130 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 12.

6 In the **Element ratio** text field, type 1.2.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Reverse direction** check box.

9 Click **Build Selected**.

Mapped 2

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 38 only.

5 Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 7 and 20 only.

5 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra coarse**.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

4 Click **Build Selected**.

Free Triangular 2

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 6 only.

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Boundaries 8, 10, and 31 only.

3 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.

4 In the list, select **31**.

5 Click **Remove from Selection**.

6 Select Boundaries 8, 10, and 30 only.

7 In the list, select **30**.

8 Click **Remove from Selection**.

9 Select Boundaries 7, 8, and 10 only.

Size 1

1 Right-click **Edge 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Maximum element size** check box. In the associated text field, type `0.05`.

7 Click **Build Selected**.

Free Triangular 3

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

1 Right-click **Free Triangular 3** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely coarse**.

4 Click **Build All**.

Refine the mesh for the 3D rim to improve visualization.

GEOMETRY 2

In the **Model Builder** window, under **3D [Rim] (comp2)** click **Geometry 2**.

Remove Details 1 (rmd1)

1 In the **Geometry** toolbar, click **Remove Details**.

2 Right-click **Remove Details 1 (rmd1)** and choose **Build All**.

MESH 2

1 In the **Model Builder** window, under **3D [Rim] (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

STUDY: FIBER DIRECTION

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study: Fiber Direction` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Fiber Direction** 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 **2D Axisymmetric [Tire] (comp1)>Definitions>Contact Pair 1 (p1)**.

5 Click **Disable**.

6 In the tree, select **2D Axisymmetric [Tire] (comp1)>Solid Mechanics (solid), Controls spatial frame**.

7 Click **Disable in Model**.

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

RESULTS

Coordinate System Surface 1

1 In the **Model Builder** window, expand the **Coordinate system (cc)** node, then click **Coordinate System Surface 1**.

2 In the **Settings** window for **Coordinate System Surface**, locate the **Positioning** section.

3 From the **Placement** list, choose **Element centers**.

4 In the **Coordinate system (cc)** 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: INFLATION

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study: Inflation` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Inflation** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Curvilinear Coordinates (cc)**.

4 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study: Fiber Direction, Stationary**.

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Continuation parameter for inflation)	`range(0, 0.1, 1)`	

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

Use **Add Predefined Plot** to generate plots of the von Mises stress and the circumferential displacement.

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: Inflation/Solution 2 (sol2)>Solid Mechanics>Stress (solid)**.

4 Click **Add Plot** in the window toolbar.

5 In the tree, select **Study: Inflation/Solution 2 (sol2)>Solid Mechanics>Displacement (solid)**.

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

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

3 Select the **Show maximum and minimum values** check box.

Surface 1

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

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

3 From the **Unit** list, choose **MPa**.

Change the range of the plot to highlight the stress in the carcass.

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

5 In the **Maximum** text field, type `15`.

6 In the **Stress (solid)** toolbar, click **Plot**.

Modify the plot of displacements in order to show the circumferential component.

Out-of-Plane Displacement (solid)

1 In the **Model Builder** window, under **Results** click **Displacement (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Out-of-Plane Displacement (solid)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Out-of-Plane Displacement (solid)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `v`.

4 From the **Unit** list, choose **mm**.

5 In the **Out-of-Plane Displacement (solid)** toolbar, click **Plot**.

Add revolution datasets to plot the stress in 3D.

6 In the **Model Builder** window, click **Results**.

7 In the **Settings** window for **Results**, locate the **Update of Results** section.

8 Select the **Only plot when requested** check box.

Revolution 2D: Tire

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, type `Revolution 2D: Tire` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Inflation/Solution 2 (sol2)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

5 In the **Revolution angle** text field, type `225`.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–6 only.

Mirror 3D: Tire

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D: Tire` in the **Label** text field.

3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

4 In the **z-coordinate** text field, type `-7E-2`.

Stress, 3D (solid)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, 3D (solid)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D: Tire**.

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

5 In the **Title** text area, type `von Mises stress (MPa)`.

6 In the **Parameter indicator** text field, type `para=eval(para)`.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

8 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Tire

1 Right-click **Stress, 3D (solid)** and choose **Volume**.

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

3 In the **Expression** text field, type `solid.misesGp`.

4 From the **Unit** list, choose **MPa**.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `180`.

7 Locate the **Coloring and Style** section. Click **Change Color Table**.

8 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

9 Click **OK**.

10 In the **Settings** window for **Volume**, type `Tire` in the **Label** text field.

Deformation 1

1 Right-click **Tire** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

Add a dedicated dataset for the bead wires.

Revolution 2D: Tire

In the **Model Builder** window, under **Results>Datasets** right-click **Revolution 2D: Tire** and choose **Duplicate**.

Revolution 2D: Bead Wires

1 In the **Model Builder** window, under **Results>Datasets** click **Revolution 2D: Tire 1**.

2 In the **Settings** window for **Revolution 2D**, type `Revolution 2D: Bead Wires` in the **Label** text field.

3 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type `275`.

Selection

1 In the **Model Builder** window, expand the **Revolution 2D: Bead Wires** node, then click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Bead Wires**.

Mirror 3D: Tire

In the **Model Builder** window, under **Results>Datasets** right-click **Mirror 3D: Tire** and choose **Duplicate**.

Mirror 3D: Bead Wires

1 In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D: Tire 1**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D: Bead Wires` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D: Bead Wires**.

Tire

In the **Model Builder** window, under **Results>Stress, 3D (solid)** right-click **Tire** and choose **Duplicate**.

Bead Wires

1 In the **Model Builder** window, under **Results>Stress, 3D (solid)** click **Tire 1**.

2 In the **Settings** window for **Volume**, type `Bead Wires` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D: Bead Wires**.

4 From the **Solution parameters** list, choose **From parent**.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Tire**.

Generate dedicated revolution datasets also for the belts.

Revolution 2D: Bead Wires

In the **Model Builder** window, under **Results>Datasets** right-click **Revolution 2D: Bead Wires** and choose **Duplicate**.

Revolution 2D: Belt 1

1 In the **Model Builder** window, under **Results>Datasets** click **Revolution 2D: Bead Wires 1**.

2 In the **Settings** window for **Revolution 2D**, type `Revolution 2D: Belt 1` in the **Label** text field.

3 Locate the **Revolution Layers** section. In the **Start angle** text field, type `135`.

4 In the **Revolution angle** text field, type `40`.

5 From the **Number of layers** list, choose **Fine**.

Selection

1 In the **Model Builder** window, expand the **Revolution 2D: Belt 1** node, then click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Belt 1**.

Mirror 3D: Bead Wires

In the **Model Builder** window, under **Results>Datasets** right-click **Mirror 3D: Bead Wires** and choose **Duplicate**.

Mirror 3D: Belt 1

1 In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D: Bead Wires 1**.

2. In the **Settings** window for **Mirror 3D**, type `Mirror 3D: Belt 1` in the **Label** text field.
3. Click to expand the **Advanced** section. Select the **Define variables** check box.
4. Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D: Belt 1**.

Use a **Streamline Surface** plot to show the cords in the belt.

Stress, 3D (solid)

In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

Belt 1

1. In the **Stress, 3D (solid)** toolbar, click **More Plots** and choose **Streamline Surface**.
2. In the **Settings** window for **Streamline Surface**, type `Belt 1` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D: Belt 1**.
4. From the **Solution parameters** list, choose **From parent**.
5. Locate the **Expression** section. In the **r-component** text field, type `solid.tl1.hmm1.fibt1.aOR*((mir1side*2)-1)`.
6. In the **phi-component** text field, type `solid.tl1.hmm1.fibt1.aOPHI`.
7. In the **z-component** text field, type `solid.tl1.hmm1.fibt1.aOZ*((mir1side*2)-1)`.
8. Select Domain 5 only.
9. Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.
10. In the **Separating distance** text field, type `0.02`.
11. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
12. In the **Tube radius expression** text field, type `solid.tl1.hmm1.fibt1.d/2`.
13. Select the **Radius scale factor** check box. In the associated text field, type `2`.
14. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Tire**.

Color Expression 1

1. Right-click **Belt 1** and choose **Color Expression**.
2. In the **Settings** window for **Color Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.tl1.hmm1.fibt1.mises`.
4. From the **Unit** list, choose **MPa**.

Deformation 1

In the **Model Builder** window, right-click **Belt 1** and choose **Deformation**.

Revolution 2D: Belt 1

In the **Model Builder** window, under **Results>Datasets** right-click **Revolution 2D: Belt 1** and choose **Duplicate**.

Revolution 2D: Belt 2

1. In the **Model Builder** window, under **Results>Datasets** click **Revolution 2D: Belt 1.1**.
2. In the **Settings** window for **Revolution 2D**, type `Revolution 2D: Belt 2` in the **Label** text field.
3. Locate the **Revolution Layers** section. In the **Revolution angle** text field, type `30`.

Selection

1. In the **Model Builder** window, expand the **Revolution 2D: Belt 2** node, then click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Belt 2**.

Mirror 3D: Belt 1

In the **Model Builder** window, under **Results>Datasets** right-click **Mirror 3D: Belt 1** and choose **Duplicate**.

Mirror 3D: Belt 2

1. In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D: Belt 1.1**.
2. In the **Settings** window for **Mirror 3D**, type `Mirror 3D: Belt 2` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D: Belt 2**.

Belt 1

In the **Model Builder** window, under **Results>Stress, 3D (solid)** right-click **Belt 1** and choose **Duplicate**.

Belt 2

1. In the **Model Builder** window, under **Results>Stress, 3D (solid)** click **Belt 1.1**.
2. In the **Settings** window for **Streamline Surface**, type `Belt 2` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D: Belt 2**.
4. Locate the **Expression** section. In the **r-component** text field, type `solid.tl1.hmm1.fibt2.aOR*((mir1side*2)-1)`.
5. In the **phi-component** text field, type `solid.tl1.hmm1.fibt2.aOPHI`.
6. In the **z-component** text field, type `solid.tl1.hmm1.fibt2.aOZ*((mir1side*2)-1)`.
7. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Tube radius expression** text field, type `solid.tl1.hmm1.fibt2.d/2`.

Color Expression 1

1 In the **Model Builder** window, expand the **Belt 2** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `solid.tl1.hmm1.fibt2.mises`.

4 In the **Stress, 3D (solid)** toolbar, click **Plot**.

Revolution 2D: Belt 2

In the **Model Builder** window, under **Results>Datasets** right-click **Revolution 2D: Belt 2** and choose **Duplicate**.

Revolution 2D: Belt 3

1 In the **Model Builder** window, expand the **Results>Datasets>Revolution 2D: Belt 2.1** node, then click **Revolution 2D: Belt 2.1**.

2 In the **Settings** window for **Revolution 2D**, type `Revolution 2D: Belt 3` in the **Label** text field.

3 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type `20`.

Selection

1 In the **Model Builder** window, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Belt 3**.

Mirror 3D: Belt 2

In the **Model Builder** window, under **Results>Datasets** right-click **Mirror 3D: Belt 2** and choose **Duplicate**.

Mirror 2D: Belt 3

1 In the **Model Builder** window, click **Mirror 3D: Belt 2.1**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D: Belt 3**.

4 In the **Label** text field, type `Mirror 2D: Belt 3`.

Belt 2

In the **Model Builder** window, under **Results>Stress, 3D (solid)** right-click **Belt 2** and choose **Duplicate**.

Belt 3

1 In the **Model Builder** window, under **Results>Stress, 3D (solid)** click **Belt 2.1**.

2 In the **Settings** window for **Streamline Surface**, type `Belt 3` in the **Label** text field.

3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D: Belt 3**.
4. Locate the **Expression** section. In the **r-component** text field, type `solid.tl1.hmm1.fibt3.a0R`.
5. In the **phi-component** text field, type `solid.tl1.hmm1.fibt3.a0PHI`.
6. In the **z-component** text field, type `solid.tl1.hmm1.fibt3.a0Z`.
7. Locate the **Streamline Positioning** section. In the **Separating distance** text field, type `0.008`.
8. From the **Advanced parameters** list, choose **Manual**.
9. In the **Terminating distance factor** text field, type `0.1`.
10. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Tube radius expression** text field, type `solid.tl1.hmm1.fibt3.d/2`.

Color Expression 1

1. In the **Model Builder** window, expand the **Belt 3** node, then click **Color Expression 1**.
2. In the **Settings** window for **Color Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.tl1.hmm1.fibt3.mises`.
4. In the **Stress, 3D (solid)** toolbar, click **Plot**.

Study: Inflation/Solution 2 (sol2)

In the **Model Builder** window, under **Results>Datasets** right-click **Study: Inflation/ Solution 2 (sol2)** and choose **Duplicate**.

Rim geometry

1. In the **Model Builder** window, under **Results>Datasets** click **Study: Inflation/ Solution 2 (3) (sol2)**.
2. In the **Settings** window for **Solution**, type `Rim geometry` in the **Label** text field.
3. Locate the **Solution** section. From the **Component** list, choose **3D [Rim] (comp2)**.

Rim

1. In the **Model Builder** window, right-click **Stress, 3D (solid)** and choose **Surface**.
2. In the **Settings** window for **Surface**, type `Rim` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Rim geometry (sol2)**.
4. Locate the **Expression** section. In the **Expression** text field, type `1`.
5. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
6. Click to expand the **Quality** section. From the **Resolution** list, choose **Fine**.

Material Appearance 1

1 Right-click **Rim** and choose **Material Appearance**.

2 In the **Stress, 3D (solid)** toolbar, click **Plot**.

Stress, 3D (solid)

In the **Model Builder** window, under **Results** right-click **Stress, 3D (solid)** and choose **Duplicate**.

Material and Fiber Direction

1 In the **Model Builder** window, expand the **Results>Stress, 3D (solid) 1** node, then click **Stress, 3D (solid) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Material and Fiber Direction` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **None**.

Tire

1 In the **Model Builder** window, click **Tire**.

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

3 In the **Expression** text field, type `1`.

Material Appearance 1

1 Right-click **Tire** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material** list, choose **Material Link: Rubber [Sidewall and Tread] (matlnk2)**.

4 In the **Material and Fiber Direction** toolbar, click **Plot**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Bead Wires** and choose **Material Appearance**.

2 In the **Material and Fiber Direction** toolbar, click **Plot**.

Color Expression 1

1 In the **Model Builder** window, expand the **Results>Material and Fiber Direction>Belt 1** node.

2 Right-click **Color Expression 1** and choose **Delete**.

Belt 1

1 In the **Model Builder** window, under **Results>Material and Fiber Direction** click **Belt 1**.

2 In the **Settings** window for **Streamline Surface**, locate the **Inherit Style** section.

3 From the **Plot** list, choose **None**.

4 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
5 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
6 Click **Define custom colors**.
7 Set the RGB values to 253, 185, and 19, respectively.
8 Click **Add to custom colors**.
9 Click **Show color palette only** or **OK** on the cross-platform desktop.

Deformation 1

1 In the **Model Builder** window, click **Deformation 1**.
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.

Color Expression 1

1 In the **Model Builder** window, expand the **Results>Material and Fiber Direction>Belt 2** node.
2 Right-click **Color Expression 1** and choose **Delete**.

Belt 2

1 In the **Model Builder** window, under **Results>Material and Fiber Direction** click **Belt 2**.
2 In the **Settings** window for **Streamline Surface**, locate the **Inherit Style** section.
3 From the **Plot** list, choose **None**.
4 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
5 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
6 Click **Define custom colors**.
7 Set the RGB values to 54, 140, and 203, respectively.
8 Click **Add to custom colors**.
9 Click **Show color palette only** or **OK** on the cross-platform desktop.

Deformation 1

1 In the **Model Builder** window, click **Deformation 1**.
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.

Color Expression 1

1. In the **Model Builder** window, expand the **Belt 3** node.
2. Right-click **Color Expression 1** and choose **Delete**.

Belt 3

1. In the **Model Builder** window, under **Results>Material and Fiber Direction** click **Belt 3**.
2. In the **Settings** window for **Streamline Surface**, locate the **Inherit Style** section.
3. From the **Plot** list, choose **None**.
4. Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
5. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
6. Click **Define custom colors**.
7. Set the RGB values to 166, 214, and 208, respectively.
8. Click **Add to custom colors**.
9. Click **Show color palette only** or **OK** on the cross-platform desktop.

Deformation 1

1. In the **Model Builder** window, click **Deformation 1**.
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 **Material and Fiber Direction** toolbar, click **Plot**.

Revolution 2D: Belt 1

In the **Model Builder** window, under **Results>Datasets** right-click **Revolution 2D: Belt 1** and choose **Duplicate**.

Revolution 2D: Carcass

1. In the **Model Builder** window, expand the **Results>Datasets>Revolution 2D: Belt 1.1** node, then click **Revolution 2D: Belt 1.1**.
2. In the **Settings** window for **Revolution 2D**, type `Revolution 2D: Carcass` in the **Label** text field.
3. Locate the **Revolution Layers** section. In the **Revolution angle** text field, type `50`.

Selection

1. In the **Model Builder** window, click **Selection**.
2. Select Boundaries 35–37, 42, 52, 53, and 56 only.

Mirror 3D: Belt 1

In the **Model Builder** window, under **Results>Datasets** right-click **Mirror 3D: Belt 1** and choose **Duplicate**.

Mirror 3D: Carcass

1. In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D: Belt 1.1**.
2. In the **Settings** window for **Mirror 3D**, type `Mirror 3D: Carcass` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D: Carcass**.

Belt 1

In the **Model Builder** window, under **Results>Material and Fiber Direction** right-click **Belt 1** and choose **Duplicate**.

Carcass

1. In the **Model Builder** window, under **Results>Material and Fiber Direction** click **Belt 1.1**.
2. In the **Settings** window for **Streamline Surface**, type `Carcass` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D: Carcass**.
4. Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **2D Axisymmetric [Tire] (comp1)>Curvilinear Coordinates>cc.e1R,cc.e1PHI, cc.e1Z - First basis vector**.
5. Locate the **Expression** section. In the **r-component** text field, type `cc.e1R*((mir1side*2)-1)`.
6. In the **z-component** text field, type `cc.e1Z*((mir1side*2)-1)`.
7. Locate the **Surface Selection** section. Click to select the **Activate Selection** toggle button.
8. Select Domain 4 only.
9. Locate the **Streamline Positioning** section. In the **Separating distance** text field, type `0.01`.
10. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Tube radius expression** text field, type `dcord_belts/2`.
11. Find the **Point style** subsection. From the **Color** list, choose **Red**.
12. In the **Material and Fiber Direction** toolbar, click **Plot**.
13. Click **Plot**.
14. In the **Model Builder** window, click **Results**.
15. In the **Settings** window for **Results**, locate the **Update of Results** section.
16. Clear the **Only plot when requested** check box.

Generate a dedicated view for the 3D plots.

View 3D 8

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 3D**.

Camera

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

2 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

3 In the **Model Builder** window, expand the **View 3D 8** node, then click **Camera**.

4 In the **Settings** window for **Camera**, in the **Graphics** window toolbar, click ▼ next to **Scene Light**, then choose **Ambient Occlusion**.

5 In the **Graphics** window toolbar, click ▼ next to **Scene Light**, then choose **Direct Shadows**.

6 In the **Graphics** window toolbar, click ▼ next to **Scene Light**, then choose **Outdoor Environment**.

7 In the **Graphics** window toolbar, click ▼ next to **Scene Light**, then choose **Floor Shadows**.

8 Locate the **Camera** section. In the **Zoom angle** text field, type `5`.

9 Locate the **Position** section. In the **x** text field, type `-263`.

10 In the **y** text field, type `-195`.

11 In the **z** text field, type `373`.

12 Locate the **Target** section. In the **x** text field, type `0.78`.

13 In the **y** text field, type `0`.

14 In the **z** text field, type `-0.04`.

15 Locate the **Up Vector** section. In the **x** text field, type `-0.7`.

16 In the **y** text field, type `0.71`.

17 In the **z** text field, type `-0.12`.

18 Locate the **Center of Rotation** section. In the **x** text field, type `0.78`.

19 In the **y** text field, type `-0.005`.

20 In the **z** text field, type `0.044`.

Material and Fiber Direction

1 In the **Model Builder** window, under **Results** click **Material and Fiber Direction**.

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

3 From the **View** list, choose **View 3D 8**.

4 In the **Material and Fiber Direction** toolbar, click **Plot**.

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

Make the cords unaffected by lighting such that results remain clear to be read.

Visual Effects 1

1 In the **Model Builder** window, right-click **Belt 1** and choose **Visual Effects**.

2 In the **Settings** window for **Visual Effects**, locate the **Visual Effects** section.

3 Clear the **Affected by lighting** check box.

4 Find the **Ambient occlusion** subsection. From the **Mode** list, choose **Manual**.

5 Clear the **Casts shadows** check box.

6 Clear the **Receives shadows** check box.

7 Find the **Direct shadows** subsection. From the **Mode** list, choose **Manual**.

8 Clear the **Casts shadows** check box.

9 Clear the **Receives shadows** check box.

10 Right-click **Visual Effects 1** and choose **Copy**.

Belt 2

In the **Model Builder** window, under **Results>Material and Fiber Direction** right-click **Belt 2** and choose **Paste Visual Effects**.

Belt 3

In the **Model Builder** window, right-click **Belt 3** and choose **Paste Visual Effects**.

Carcass

In the **Model Builder** window, right-click **Carcass** and choose **Paste Visual Effects**.

Visual Effects 1

1 In the **Model Builder** window, click **Visual Effects 1**.

2 In the **Material and Fiber Direction** toolbar, click **Plot**.

Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

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

3 From the **View** list, choose **View 3D 8**.

4 In the **Stress, 3D (solid)** toolbar, click **Plot**.

Belt 1

In the **Model Builder** window, right-click **Belt 1** and choose **Paste Visual Effects**.

Belt 2

In the **Model Builder** window, right-click **Belt 2** and choose **Paste Visual Effects**.

Belt 3

In the **Model Builder** window, right-click **Belt 3** and choose **Paste Visual Effects**.