Submodel in a Wheel Rim

Introduction

In stress analysis, it is common that the regions with high stresses are small when compared to the whole structure. Sometimes it is not feasible to have a mesh that at the same time captures the global behavior and resolves the stress concentrations with high accuracy. This is especially true in nonlinear or dynamic problems.

You can cope with these types of problems with a technique known as *submodeling*. First you solve the complete model with a mesh which is sufficient to capture the stiffness of the structure. In a second analysis you create a local model (submodel) of the region around the stress concentration with a fine mesh, and solve it using the displacements from the global model as boundary conditions.

There are some underlying assumptions when using submodels:

- The global model is accurate enough to give correct displacements on the boundary to the submodel.
- The improvements introduced in the submodel are so small that they do not introduce significant changes in stiffness on the global level. Given this, it could still be possible to introduce a nonlinear material locally in the submodel.

This example shows how to perform submodel analysis in COMSOL Multiphysics.

Model Definition

The wheel rim for this analysis has a ten-spoke design, such that the elements of the geometry cause the finite element mesh to become quite large. The loading on the tire is composed of both the tire pressure and a load transferred from the road via the tire to the rim.

In the submodel (shown in Figure 1), you cut out a small region around the hotspot using an intersection between the rim geometry and a 70 mm-by-70 mm-by-60 mm block.
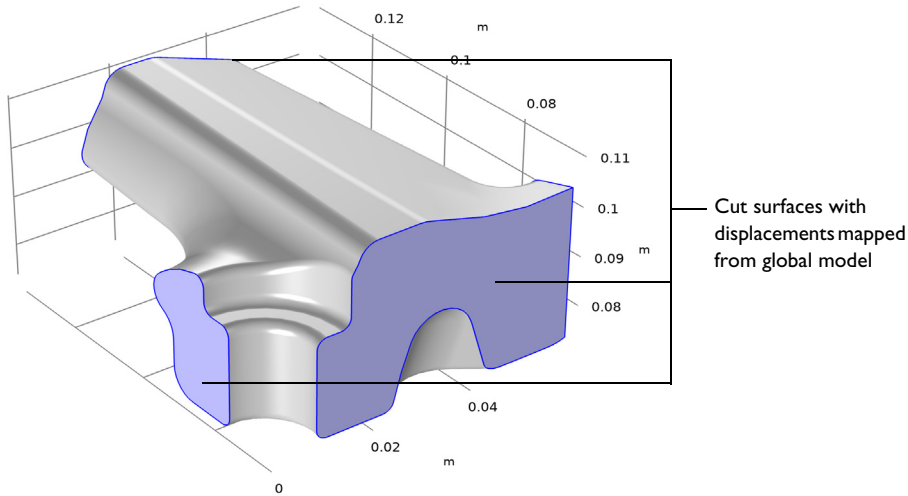

Figure 1: The submodel geometry.

MATERIAL

Aluminum with E = 70 GPa, ν = 0.33.

CONSTRAINTS

- A region around each bolt hole where the wheel rim is attached to the wheel hub is fixed.

LOADS

- Tire pressure: 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 peak of the load. Four different load cases are analyzed, where the center of the peak load is rotated 18° each time. In this way the whole load cycle for the rotating wheel can be covered. The pressure load and the load distribution carried by the wheel are shown in Figure 2.
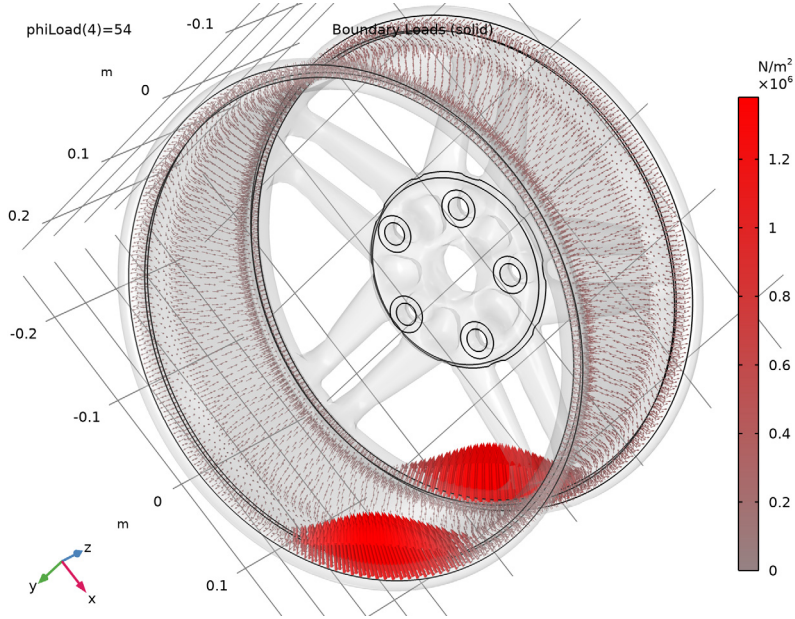
Figure 2: Pressure and tire load when rotated 54° from the center of the first pair of spokes

• In the submodel, the stress history for a full revolution of the wheel is computed. This is possible, since results from different spokes are applied to the submodel sequentially.

Results and Discussion

The highest stresses occur in the fillet where the spoke connects to the hub. In the global model the maximum equivalent stress is mesh dependent, and not reliable. In the submodel, where the resolution is good, the von Mises stress is about **96** MPa. It occurs when the load is rotated **18°** from the reference angle. In a fatigue analysis, where the lifetime could vary as the fifth power of the stress, it is essential to get this level of accuracy in the critical regions.
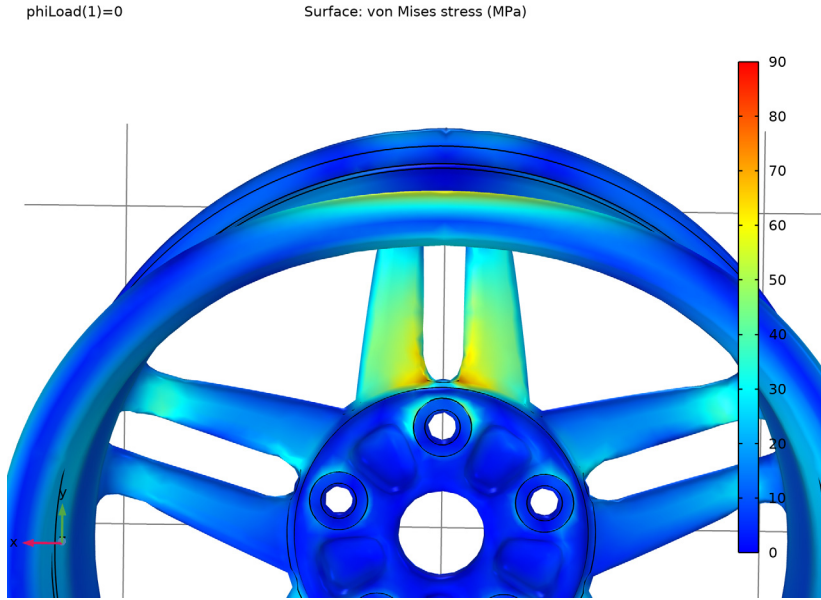
phiLoad(1)=0   Surface: von Mises stress (MPa)

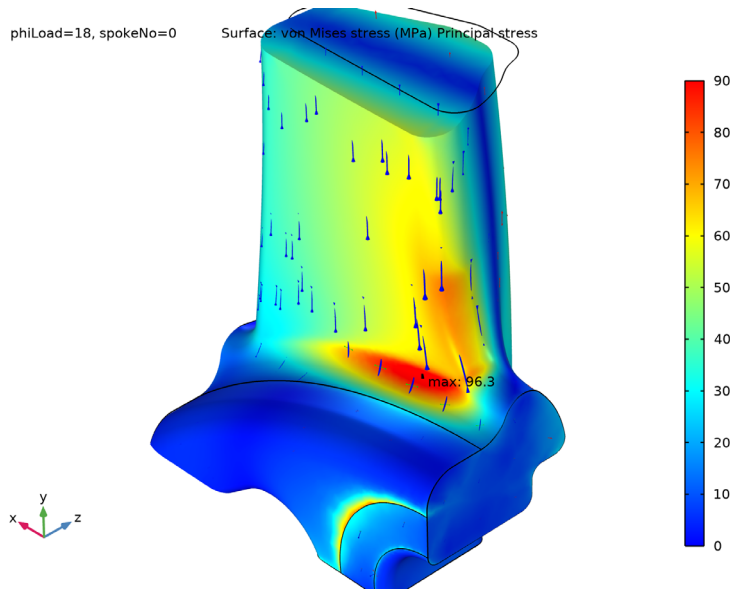*Figure 3: Stresses in the global model.*

Figure 4: Stresses in the submodel at the load position giving the peak von Mises stress

Notes About the COMSOL Implementation

Two different components are used within the same mph file. In the global model, a general extrusion feature is introduced in order to describe the mapping of results from the global model to the submodel. The general extrusion is parameterized so that displacements from different spokes can be applied to the submodel.

The pressure distribution from the tire must be adjusted so that its resultant is the intended (about 11 kN). This can be done in different ways. One simple possibility is to run a separate analysis with the tire load as the only load case and use an arbitrary load amplitude. The total reaction force is then computed, and the load amplitude is rescaled based on the result. In this model a more sophisticated method is used. An extra Global Equation is added, in which the integral of the distribution load is set equal to the known force. Thus the amplitude of the load is solved simultaneously with the rest of the problem.

Application Library path: `Structural_Mechanics_Module/Tutorials/`
`rim_submodel`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **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 In the table, enter the following settings:

Name	Expression	Value	Description
pInflation	2[bar]	2E5 Pa	Inflation pressure
tireLoad	1120[kg]*g_const	10983 N	Load on wheel
spokeNo	0	0	Spoke selection
spokeAngle	spokeNo*2*pi[rad]/5	0 rad	Rotation angle to selected spoke
phiLoad	0	0	Peak load angle
numLpos	4	4	Number of load positions in first sector
angleStep	360/(5*numLpos)	18	Step in peak load angle [deg]
angleLast	angleStep*(numLpos-1)	54	Last peak load angle [deg]

1 In the **Geometry** toolbar, click ▦ **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file
   wheel_rim_geom_sequence.mph.

3 In the **Geometry** toolbar, click ▦ **Build All**.

DEFINITIONS

TireAttachment

1 In the **Definitions** toolbar, click 🗂 **Explicit**.

2 In the **Settings** window for **Explicit**, type TireAttachment in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 2–4 and 6 only.

PressureSurface

1 In the **Definitions** toolbar, click 🗂 **Explicit**.

2 In the **Settings** window for **Explicit**, type PressureSurface in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 2–6 only.

FixedToHub

1 In the **Definitions** toolbar, click 🗂 **Explicit**.

2 In the **Settings** window for **Explicit**, type FixedToHub in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8–12 only.

ADD MATERIAL

1 In the **Home** toolbar, click ▦ **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Aluminum**.

4 Click **Add to Component** in the window toolbar.

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

MATERIALS

Aluminum (mat1)

1 Click the 👁 **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

3 Click **OK**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

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

Boundary Load 1

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

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

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

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

5 In the *p* text field, type pInflation.

DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click f⊗ **Functions** and choose **Local>Analytic**.

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

3 In the **Expression** text field, type (abs(atan2(x,y)-z*pi/180)<pi/6)*cos(3* (atan2(x,y)-z*pi/180)).

4 In the **Arguments** text field, type x, y, z.

5 Locate the **Units** section. In the **Arguments** text field, type m,m,1.

6 In the **Function** text field, type Pa.

7 In the **Function name** text field, type loadDistr.

Cylindrical System 2 (sys2)

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

SOLID MECHANICS (SOLID)

Boundary Load 2

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

2 In the **Settings** window for **Boundary Load**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **TireAttachment**.

5 Locate the **Force** section. 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

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**.

SOLID MECHANICS (SOLID)

Global Equations 1

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 (u_t0) (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 box, type force in the text field.

6 Click 🔻 **Filter**.

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

8 Click **OK**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click ✛ **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phiLoad (Peak load angle)	range(0,angleStep, angleLast)	

Solution 1 (sol1)

1 In the **Study** toolbar, click ⊞ **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Because of the model's considerable size, you use an iterative solver that can significantly save on the memory needed for the computations. Use the default GMRES iterative solver with Geometric Multigrid as a preconditioner. On the coarse multigrid level, the solver will lower the order in the discretization of the displacement variables from the default quadratic elements to linear elements.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (solid)** and choose **Enable**.

5 In the **Study** toolbar, click ▬ **Compute**.

RESULTS

Stress (solid)

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

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

3 From the **Parameter value (phiLoad)** list, choose **0**.

Surface 1

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

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

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

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

5 In the **Minimum** text field, type 0.

6 In the **Maximum** text field, type 90.

To get a better view of the region with the highest stresses (compare with Figure 3), use a **View** feature node.

DEFINITIONS

View 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

In the graphics window, rotate the model and capture the highest stressed part.

2 In the **Model Builder** window, click **View 2**.

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

4 Select the **Lock camera** check box.

RESULTS

Stress (solid)

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

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

3 From the **View** list, choose **View 2**.

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

A plot showing boundary loads is added by default. Change its setting slightly to reproduce Figure 2.

Applied Loads (solid)

In the **Model Builder** window, expand the **Applied Loads (solid)** node.

Boundary Load 1

1 In the **Model Builder** window, expand the **Results>Applied Loads (solid)> Boundary Loads (solid)** node, then click **Boundary Load 1**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

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

4 In the associated text field, type 4E-8.

5 In the **Boundary Loads (solid)** toolbar, click ⊡ **Plot**.

Start creating the submodel.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

GEOMETRY 2

1 In the **Geometry** toolbar, click 🗒 **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file
wheel_rim_geom_sequence.mph.

3 In the **Geometry** toolbar, click 🧱 **Build All**.

Rotate 1 (rot1)

In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** right-click
Rotate 1 (rot1) and choose **Disable**.

Form Composite Domains 1 (cmd1)

1 In the **Model Builder** window, click **Form Composite Domains 1 (cmd1)**.

2 In the **Settings** window for **Form Composite Domains**, click 🧱 **Build Selected**.

Block 1 (blk1)

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

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

3 In the **Width** text field, type 6e-2.

4 In the **Depth** text field, type 7e-2.

5 In the **Height** text field, type 6e-2.

6 Locate the **Position** section. In the **y** text field, type 6.5e-2.

7 In the **z** text field, type 6e-2.

Intersection 1 (int1)

1 In the **Geometry** toolbar, click 🔲 **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Form Composite Domains 1 (cmd1)

1 In the **Model Builder** window, click **Form Composite Domains 1 (cmd1)**.

2 In the **Settings** window for **Form Composite Domains**, click ▦ **Build Selected**.

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

ADD MATERIAL

1 In the **Home** toolbar, click ▦ **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Aluminum**.

4 Click **Add to Component** in the window toolbar.

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

DEFINITIONS (COMP1)

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

General Extrusion 1 (genext1)

1 In the **Definitions** toolbar, click ⌁ **Nonlocal Couplings** and choose **General Extrusion**.

2 In the **Settings** window for **General Extrusion**, type from_global in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

4 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

5 Locate the **Destination Map** section. In the **X-expression** text field, type X*
cos(spokeAngle)-Y*sin(spokeAngle).

6 In the **Y-expression** text field, type Y*cos(spokeAngle)+X*sin(spokeAngle).

7 In the **Z-expression** text field, type Z.

8 Click to expand the **Advanced** section. In the **Extrapolation tolerance** text field, type 0.5.

COMPONENT 2 (COMP2)

In the **Model Builder** window, click **Component 2 (comp2)**.

ADD PHYSICS

1 In the **Home** toolbar, click ⚛ **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component 2** in the window toolbar.

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

SOLID MECHANICS 2 (SOLID2)

Fixed Constraint 1

1 Right-click **Component 2 (comp2)>Solid Mechanics 2 (solid2)** and choose **Fixed Constraint**.

2 Select Boundary 3 only.

Prescribed Displacement 1

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

2 Select Boundaries 1 and 6–8 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type comp1.from_global(comp1.u*cos(spokeAngle)+ comp1.v*sin(spokeAngle)).

6 Select the **Prescribed in y direction** check box.

7 In the u_{0y} text field, type comp1.from_global(comp1.v*cos(spokeAngle)- comp1.u*sin(spokeAngle)).

8 Select the **Prescribed in z direction** check box.

9 In the u_{0z} text field, type comp1.from_global(comp1.w).

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

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

3 From the **Element size** list, choose **Extra fine**.

4 Locate the **Mesh Settings** section. From the **Sequence type** list, choose **User-controlled mesh**.

Size 1

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

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

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

4 Select Boundary 5 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click **Build All**.

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.

5 Click **Add Study** in the window toolbar.

6 In the **Model Builder** window, click the root node.

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

Fetch the displacements from the solution of the global model.

2 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

4 From the **Study** list, choose **Study 1, Stationary**.

5 From the **Parameter value (phiLoad)** list, choose **All**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 From the **Sweep type** list, choose **All combinations**.

8 Click ✛ **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
spokeNo (Spoke selection)		

10 Click ✛ **Add**.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
spokeNo (Spoke selection)	`range(0,1,4)`	
phiLoad (Peak load angle)	`range(0,angleStep, angleLast)`	

Solution 2 (sol2)

In the **Study** toolbar, click **Show Default Solver**.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)** node.

For the submodel, you also use the default suggested iterative solver.

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

3 Right-click **Suggested Iterative Solver (solid2)** and choose **Enable**.

STUDY 1

Solution 1 (sol1)

In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Suggested Iterative Solver (solid2)** node.

STUDY 2

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Suggested Iterative Solver (solid)** node, then click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Suggested Iterative Solver (solid2)>Multigrid 1**.

2 In the **Settings** window for **Multigrid**, locate the **General** section.

3 In the **Use hierarchy in geometries** list, select **Geometry 1**.

4 Under **Use hierarchy in geometries**, click **Delete**.

STUDY 1

Solution 1 (sol1)

1 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Suggested Iterative Solver (solid)** click **Multigrid 1**.

2 In the **Settings** window for **Multigrid**, locate the **General** section.

3 In the **Use hierarchy in geometries** list, select **Geometry 2**.

4 Under **Use hierarchy in geometries**, click **Delete**.

Avoid saving a lot of duplicate results for the full geometry.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

2 In the **Settings** window for **Field**, locate the **General** section.

3 Clear the **Store in output** check box.

4 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)> Dependent Variables 1** click **State variable loadAmpl (comp1.ODE1)**.

5 In the **Settings** window for **State**, locate the **General** section.

6 Clear the **Store in output** check box.

7 In the **Study** toolbar, click ▬ **Compute**.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid2)** node, then click **Surface 1**.

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

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

4 Locate the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type 0.

6 In the **Maximum** text field, type 90.

Marker 1

1 Right-click **Surface 1** and choose **Marker**.

2 In the **Settings** window for **Marker**, locate the **Display** section.

3 From the **Display** list, choose **Max**.

4 Locate the **Text Format** section. In the **Display precision** text field, type 3.

Stress (solid2)

1 In the **Model Builder** window, click **Stress (solid2)**.

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

3 From the **Parameter value (spokeNo)** list, choose **0**.

4 From the **Parameter value (phiLoad)** list, choose **18**.

Principal Stress Surface 1

In the **Stress (solid2)** toolbar, click **More Plots** and choose **Principal Stress Surface**.

Deformation 1

Right-click **Principal Stress Surface 1** and choose **Deformation**.

Principal Stress Surface 1

1 In the **Settings** window for **Principal Stress Surface**, click to expand the **Inherit Style** section.

2 From the **Plot** list, choose **Surface 1**.

3 Clear the **Arrow scale factor** check box.

4 Clear the **Color** check box.

5 Clear the **Color and data range** check box.

Stress in Submodel

1 In the **Model Builder** window, under **Results** click **Stress (solid2)**.

2 In the **Settings** window for **3D Plot Group**, type Stress in Submodel in the **Label** text field.

Again, create a **View 3D** feature node for the plot

DEFINITIONS (COMP2)

View 4

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Definitions** and choose **View**.

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

3 Clear the **Show grid** check box.

In the graphics window, rotate the model and capture the highest stressed part.

4 In the **Model Builder** window, click **View 4**.

5 Select the **Lock camera** check box.

Apply the view to the submodel plot.

RESULTS

Stress in Submodel

1 In the **Model Builder** window, under **Results** click **Stress in Submodel**.

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

3 From the **View** list, choose **View 4**.

4 In the **Stress in Submodel** toolbar, click ⬛ **Plot**.

Finally, create an animation showing the stress history for a complete revolution of the wheel.

Animation 1

1 In the **Results** toolbar, click ⬛ **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Stress in Submodel**.

4 Locate the **Animation Editing** section. From the **Loop over** list, choose **All solutions**.

5 In the **Parameter values (phiLoad,spokeNo)** list, choose all values.

6 Locate the **Frames** section. From the **Frame selection** list, choose **All**.

7 Right-click **Animation 1** and choose **Play**.

Set the first study so that it computes only the full model.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** 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 **Solid Mechanics 2 (solid2)**.