Model created in COMSOL Multiphysics 6.4

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.

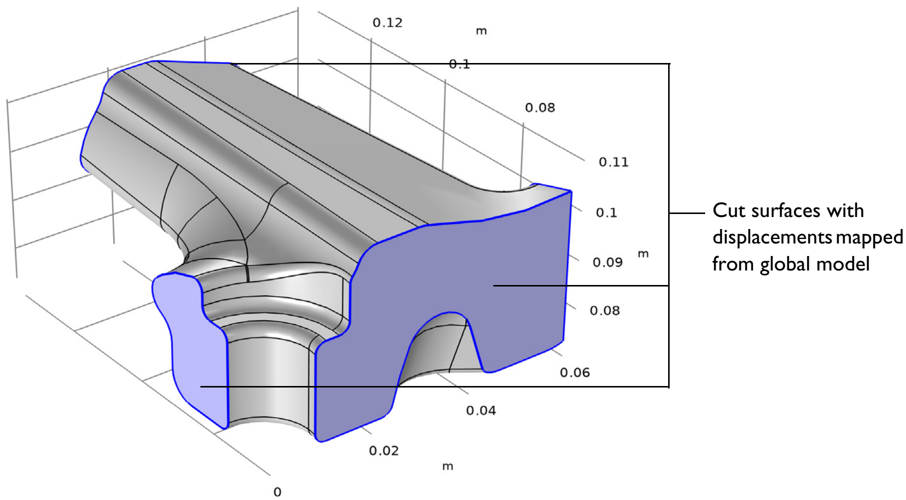

Figure 1: The submodel geometry.

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 modeled using a spring foundation. The stiffness is approximated as steel of 1 cm thickness.

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.

- 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.

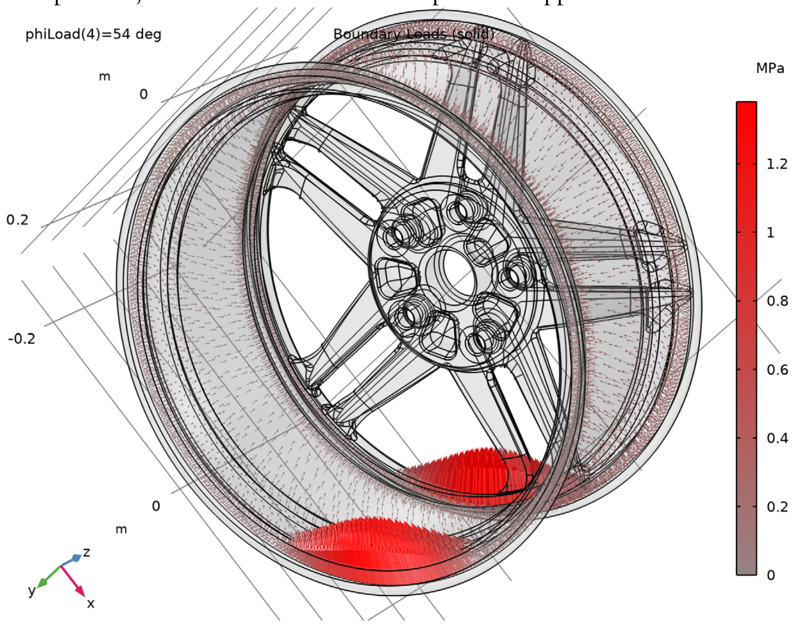

Figure 2: Pressure and tire load when rotated 54° from the center of the first pair of spokes.

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 98 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.

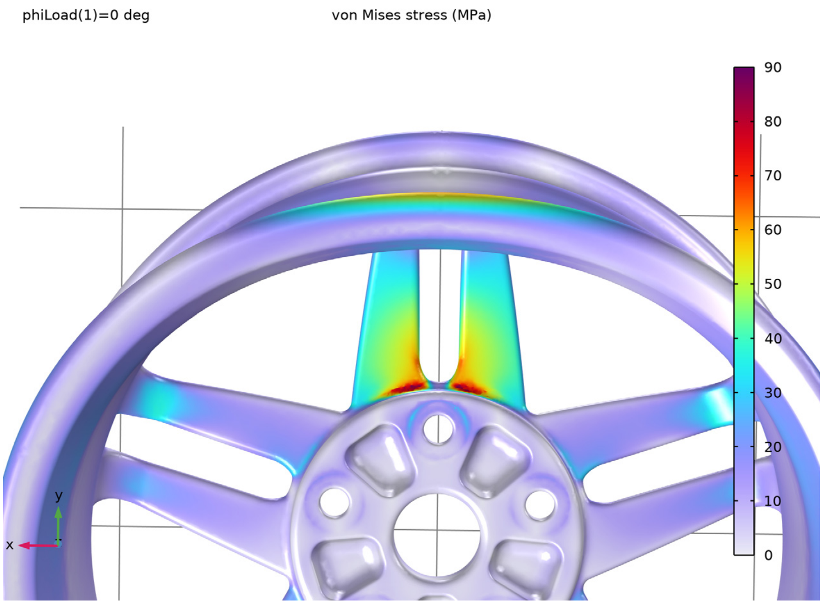

Figure 3: Stresses in the global model.

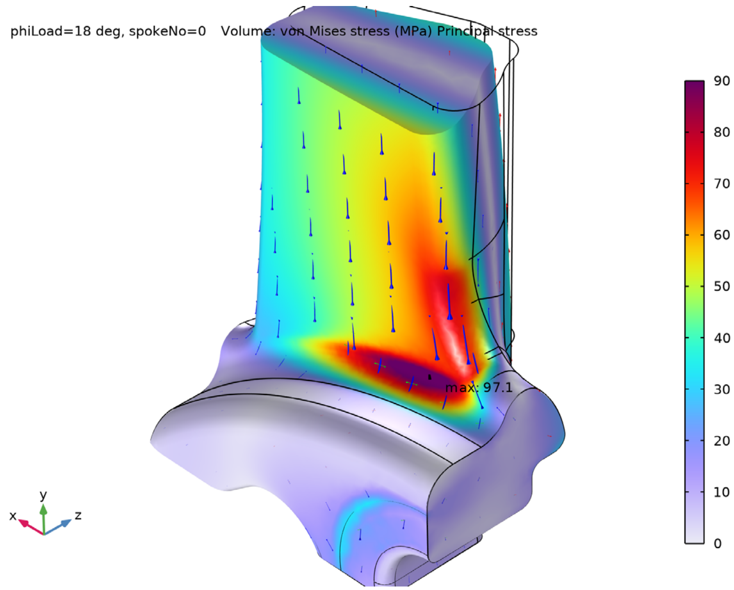

Figure 4: Stresses in the submodel at the load position giving the peak von Mises stress.

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.

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[deg]	0 rad	Peak load angle
numLpos	4	4	Number of load positions in first sector
angleStep	360[deg]/(5*numLpos)	0.31416 rad	Step in peak load angle [deg]
angleLast	angleStep*(numLpos-1)	0.94248 rad	Last peak load angle [deg]

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `wheel_rim_geom_sequence.mph`.

Import 1 (imp1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Import 1 (imp1)**.

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

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

Create selections to help with the physics setup. When building the **Rotate 1** node, the selections will propagate to the full geometry.

Fixed to Hub

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Fixed to Hub` in the **Label** text field.

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

4 On the object **imp1**, select Boundaries 10 and 137 only.

Tire Attachment

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Tire Attachment` in the **Label** text field.

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

4 On the object **imp1**, select Boundaries 47–50 only.

Pressure Surface

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Pressure Surface` in the **Label** text field.

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

4 On the object **imp1**, select Boundaries 36–38, 41, 42, and 45–50 only.

Fixed to Hub (sel1), Pressure Surface (sel3), Tire Attachment (sel2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, Ctrl-click to select **Fixed to Hub (sel1)**, **Tire Attachment (sel2)**, and **Pressure Surface (sel3)**.

2 Right-click and choose **Group**.

Selections

In the **Settings** window for **Group**, type `Selections` in the **Label** text field.

Remove Details 1 (rmd1)

1 In the **Model Builder** window, click **Remove Details 1 (rmd1)**.

2 In the **Settings** window for **Remove Details**, click **Build Selected**.

ADD MATERIAL

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

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

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

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

5 In the **Materials** 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, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

3 Click **OK**.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Structural steel**.

3 Click the **Add to Component** button in the window toolbar.

MATERIALS

Structural steel (mat2)

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

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

3 From the **Selection** list, choose **Fixed to Hub**.

SOLID MECHANICS (SOLID)

Spring Foundation 1

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

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

3 From the **Selection** list, choose **Fixed to Hub**.

4 Locate the **Spring** section. From the **Spring type** list, choose **Use material data**.

5 In the d_s text field, type `0.01`.

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 **Pressure Surface**.

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 **Definitions** toolbar, click **Analytic**.

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

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

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

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

Argument	Unit
x	m
y	m
phi	rad

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 **Tire Attachment**.

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 **Tire Attachment**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
loadAmpl	tireLoad/intop1(loadDistr(X,Y,0)*cos(atan2(X,Y)))		Load amplitude

MESH 1

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

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

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

4 Click **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** checkbox.

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)`	`deg`

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 **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Suggested Iterative Solver (solid)** and choose **Enable**.

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

Set default units for result presentation.

RESULTS

Preferred Units 1

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

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

3 Click **Add Physical Quantity**.

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

5 Click **OK**.

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

7 In the table, enter the following settings:

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

8 Select the **Apply conversions to expressions with the same dimensions** checkbox.

9 Click **Apply**.

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 (deg))** list, choose **0**.
4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Volume 1

1 In the **Model Builder** window, click **Volume 1**.
2 In the **Settings** window for **Volume**, click to expand the **Range** section.
3 Select the **Manual color range** checkbox.
4 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 **Settings** window for **View**, locate the **View** section.
3 Select the **Lock camera** checkbox.

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**.

Insert a plot showing boundary loads from **Result Templates**. Change its setting slightly to reproduce Figure 2.

RESULT TEMPLATES

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

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

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Applied Loads (solid)** > **Boundary Loads (solid)**.

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

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

RESULTS

Boundary Load 1

1 In the **Model Builder** window, expand the **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** checkbox. In the associated text field, type `4E-2`.

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

ROOT

Start creating the submodel.

ADD COMPONENT

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

GEOMETRY 2

Import 1 (imp1)

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

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

3 From the **Source** list, choose **Any importable file**.

4 Click **Browse**.

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

6 Click **Import**.

7 Click the **Go to Default View** button in the **Graphics** toolbar.

Prescribed Displacement

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

2 In the **Settings** window for **Block**, type `Prescribed Displacement` in the **Label** text field.

3 Locate the **Size and Shape** section. 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`.
8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
9 From the **Show in physics** list, choose **Boundary selection**.

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.
3 In the **Settings** window for **Intersection**, click **Build Selected**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Fixed to Hub

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `Fixed to Hub` in the **Label** text field.
3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
4 On the object **int1**, select Boundary 5 only.
5 Click **Build Selected**.

Finer Size Setting

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 On the object **int1**, select Boundary 14 only.
5 In the **Label** text field, type `Finer Size Setting`.
6 Click **Build Selected**.

Finer Size Setting (sel2), Fixed to Hub (sel1)

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Geometry 2**, Ctrl-click to select **Fixed to Hub (sel1)** and **Finer Size Setting (sel2)**.
2 Right-click and choose **Group**.

Selections

In the **Settings** window for **Group**, type `Selections` in the **Label** text field.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

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

3 Click the **Add to Component** button in the window toolbar.

4 In the tree, select **Built-in** > **Structural steel**.

5 Click the **Add to Component** button in the window toolbar.

MATERIALS

Structural steel (mat4)

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

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

3 From the **Selection** list, choose **Fixed to Hub**.

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 the **Add to Component 2** button in the window toolbar.
5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS 2 (SOLID2)

Spring Foundation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.
2 In the **Settings** window for **Spring Foundation**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Fixed to Hub**.
4 Locate the **Spring** section. From the **Spring type** list, choose **Use material data**.
5 In the d_s text field, type `0.01`.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2 In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Prescribed Displacement**.
4 Locate the **Prescribed Displacement** section. From the **Displacement in x direction** list, choose **Prescribed**.
5 In the u_{0x} text field, type `comp1.from_global(comp1.u*cos(spokeAngle)+comp1.v*sin(spokeAngle))`.
6 From the **Displacement in y direction** list, choose **Prescribed**.
7 In the u_{0y} text field, type `comp1.from_global(comp1.v*cos(spokeAngle)-comp1.u*sin(spokeAngle))`.
8 From the **Displacement in z direction** list, choose **Prescribed**.
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 **Fine**.

4 Locate the **Sequence Type** section. From the 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 From the **Selection** list, choose **Finer Size Setting**.

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

6 Click **Build All**.

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

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

6 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 (deg))** list, choose **Automatic (all solutions)**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

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)`	`deg`

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

12 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	**Output**
Solid Mechanics (solid)	None
Solid Mechanics 2 (solid2)	Physics controlled

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 **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** > **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 box, 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 box, select **Geometry 2**.

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

STUDY 2

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

RESULTS

Volume 1

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

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

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

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

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

Marker 1

1 Right-click **Volume 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 **Precision** text field, type 3.

Stress (solid2)

1. In the **Model Builder** window, under **Results** 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 (deg))** 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 **Volume 1**.
3. Clear the **Arrow scale factor** checkbox.
4. Clear the **Color** checkbox.
5. Clear the **Color and data range** checkbox.

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** checkbox.

   In the **Graphics** window, rotate the model and capture the highest stressed part.
4. Select the **Lock camera** checkbox.

   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 Click the **Play** button in the **Graphics** toolbar.

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 **Solve for** column of the table, under **Component 2 (comp2)**, clear the checkbox for **Solid Mechanics 2 (solid2)**.