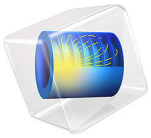


Model created in COMSOL Multiphysics 6.4

# Wheel Rim — Stress Optimization with Fatigue Evaluation

## *Introduction*

This model demonstrates how to improve fatigue properties of a wheel rim using shape optimization. The fatigue properties are not optimized directly. Instead a heuristic methodology is applied where an approximate value of the maximum stress is minimized subject to constraints on the mass and stiffness.

## *Model Definition*

The model is based on the model Fatigue Analysis of a Wheel Rim in the Fatigue Module Application Library. The initial stiffness as well as fatigue properties can thus be evaluated immediately. The fatigue analysis indicates that failure is likely to occur near a small fillet as illustrated in Figure 1.

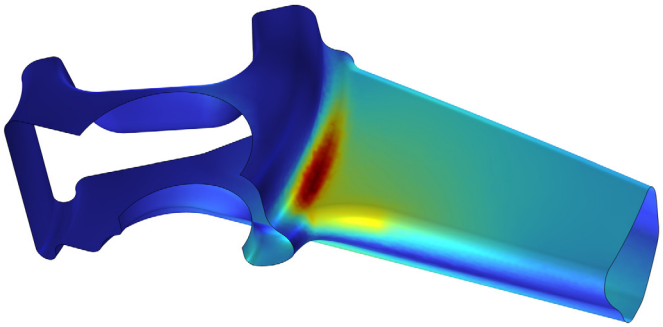


*Figure 1: The initial fatigue properties.*

The **Free Shape Domain**, **Free Shape Boundary**, and **Symmetry/Roller** shape optimization features are used to allow modification of the fillet details where the maximum stress occurs. COMSOL does not support optimization of the fatigue properties directly, so a heuristic approach is applied. The method is not guaranteed to improve the fatigue properties and therefore it is critical that these are evaluated before and after optimization.

The heuristic approach involves using the p-norm of the von Mises stress as the objective function, φ,

$$\varphi = \left[\int_{\Omega} (\sigma_{mises}/\sigma_{max})^p d\Omega / \int_{\Omega} d\Omega\right]^{1/p}$$

For large values of $p$ the objective is a good approximation of the maximum stress, but too large values can cause numerical problems for the optimization solver, so the model uses $p = 20$.The objective is scaled with the initial value, so (strictly speaking) the maximum stress, $\sigma_{max}$, does not play a role; it only serves to prevent unit warnings. The setup of this objective function is simplified by the use of the **P-Norm** feature. COMSOL Multiphysics comes with built-in variables for the volume and the stiffness, so it is easy to specify these constraints. The number of optimization iterations is limited to 20 and an iterative solver is used for the structural mechanics to save computation time. The reference model uses a submodeling approach to capture the stresses accurately without having to model the entire wheel in detail for every load cases, but the boundary conditions for the submodel would have to be recomputed in every optimization iteration, which could be done, but for simplicity the optimization model just uses a coarse mesh for the spokes where the stress is not evaluated; see Figure 2.

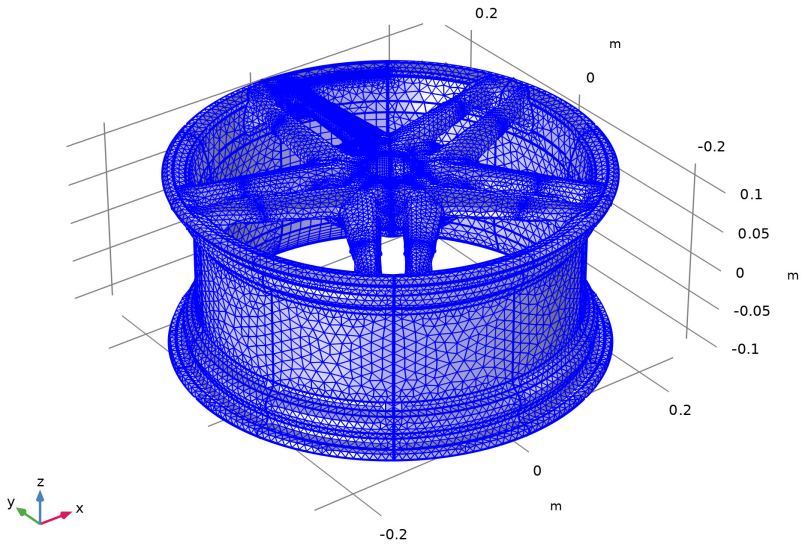



*Figure 2: The mesh is coarse for all but one of the ten spokes.*

Finally, the reference model considers six load cases, but only the two worst load cases are considered in the optimization, and one can see that this is sufficient based on the verification analysis which considers all six load cases.

## *Results and Discussion*

As one might expect the optimization increases the fillet to reduce the maximum stress. This can be seen in Figure 3.

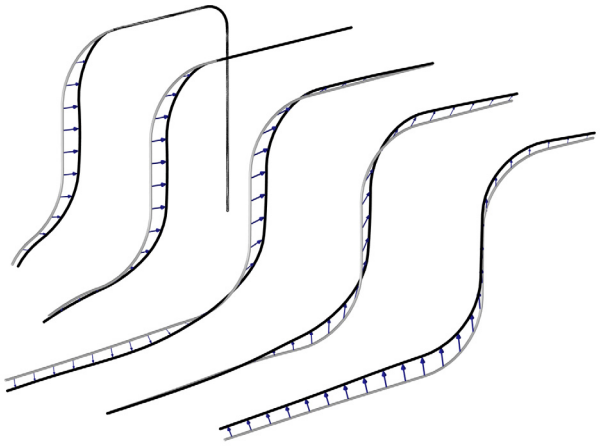


*Figure 3: Line slices are plotted for the initial (gray) and optimization geometry (black) indicating that the shape optimization increases the fillet radius at the point of maximum stress.*

To verify that the optimization has indeed improved the fatigue properties, a fatigue analysis is performed on the optimized design, and the result can be seen in Figure 4.

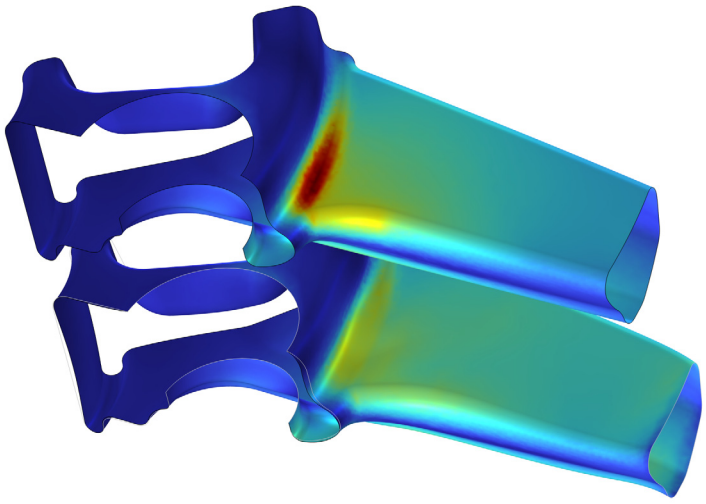


*Figure 4: The shape optimization has removed the stress concentration at the fillet as compared to the initial design.*

## *Notes About the COMSOL Implementation*

When using the MMA optimization solver, note that the (default) globally convergent behavior is disabled. The **Fatigue** study step zeros the shape optimization variables, so the model uses the `withsol` operator to plot the fatigue usage in the deformed frame.

**Application Library path:** `Fatigue_Module/Stress_Based/rim_fatigue_optimization`

## *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 In the **Select Physics** tree, select **Structural Mechanics** > **Fatigue (ftg)**.

5 Click **Add**.

6 Click **Study**.

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

8 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:

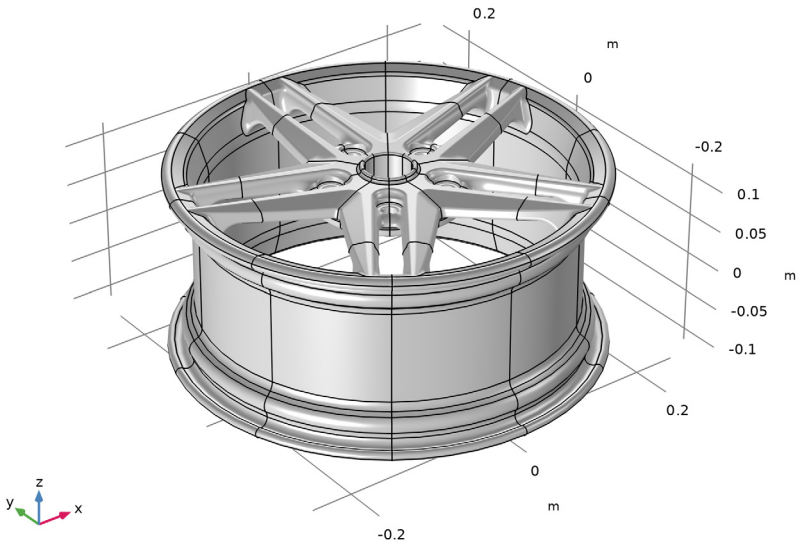
| Name | Expression | Value | Description |
|---|---|---|---|
| pInflation | 2[bar] | 2E5 Pa | Inflation pressure |
| tireLoad | 1120[kg]*g_const | 10983 N | Load on wheel |
| spokeNo | 0 | 0 | Spoke selection |
| spokeAngle | spokeNo*2*pi[rad]/5 | 0 rad | Rotation angle to selected spoke |
| phiLoad | 0[deg] | 0 rad | Peak load angle |
| numLpos | 2 | 2 | Number of load positions in first sector |
| angleStep | 360[deg]/(5*numLpos) | 0.62832 rad | Step in peak load angle [deg] |
| Ws0 | 10.5[mJ] | 0.0105 J | Characteristic elastic energy |
| pExp | 20 | 20 | P-norm exponent |
| sigmaMax | 100[MPa] | 1E8 Pa | Characteristic stress |
| rDesign | 16.5[cm] | 0.165 m | Design domain radius |

| Name | Expression | Value | Description |
|---|---|---|---|
| angleLast | angleStep* (numLpos-1) | 0.62832 rad | Last peak load angle [deg] |
| index | 0 | 0 | |

### GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

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 `rim_fatigue_optimization_geom_sequence.mph`.
3. In the **Geometry** toolbar, click **Build All**.
4. Click the **Zoom Extents** button in the **Graphics** toolbar.
5. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.




The geometry should now look like that in Figure 1.

### MESH 1

*Size 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 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 **Source Critical Fillet Boundary**.

5 Click to expand the **Element Size Parameters** section. Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `5[mm]`.

*Size*

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Curvature factor** text field, type `0.5`.

*Size 2*

1 In the **Model Builder** window, right-click **Mesh 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 **Domain**.

4 From the **Selection** list, choose **Source Design Domain**.

5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `8[mm]`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `1[mm]`.

9 Select the **Curvature factor** checkbox. In the associated text field, type `0.45`.

*Edge 1*

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

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

3 From the **Selection** list, choose **Mapped Edges**.

*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 **Extra fine**.

*Size 2*

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

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

3 From the **Selection** list, choose **Long Mapped Edges**.

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

*Mapped 1*

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

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

3 From the **Selection** list, choose **Mapped Boundaries**.

*Size 1*

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

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

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

*Size 2*

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

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

3 From the **Selection** list, choose **Center Hub Boundaries**.

4 Locate the **Element Size** section. Click the **Custom** button.

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

6 Select the **Maximum element size** checkbox. In the associated text field, type `3[mm]`.

*Edge 2*

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

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

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

*Copy Edge 1*

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Edge**.

2 In the **Settings** window for **Copy Edge**, locate the **Source Edges** section.

3 From the **Selection** list, choose **Spoke Edges Source 0**.

4 Locate the **Destination Edges** section. From the **Selection** list, choose **Spoke Edges Destination 0**.

*Copy Edge 2*

1 Right-click **Copy Edge 1** and choose **Duplicate**.

2 In the **Settings** window for **Copy Edge**, locate the **Source Edges** section.

3 From the **Selection** list, choose **Spoke Edges Source 1**.

4 Locate the **Destination Edges** section. From the **Selection** list, choose **Spoke Edges Destination 1**.

*Copy Edge 3*

1 Right-click **Copy Edge 2** and choose **Duplicate**.

2 In the **Settings** window for **Copy Edge**, locate the **Source Edges** section.

3 From the **Selection** list, choose **Spoke Edges Source 2**.

4 Locate the **Destination Edges** section. From the **Selection** list, choose **Spoke Edges Destination 2**.

*Copy Edge 4*

1 Right-click **Copy Edge 3** and choose **Duplicate**.

2 In the **Settings** window for **Copy Edge**, locate the **Source Edges** section.

3 From the **Selection** list, choose **Spoke Edges Source 1**.

4 Locate the **Destination Edges** section. From the **Selection** list, choose **Spoke Edges Destination 3**.

5 Click **Build Selected**.

*Free Triangular 1*

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

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

3 From the **Selection** list, choose **Unmapped Boundaries**.

*Size 1*

1 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 **Finer**.

*Size 3*

1 In the **Model Builder** window, right-click **Mesh 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 **Holes**.

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

*Free Tetrahedral 1*

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

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

3 Right-click **Free Tetrahedral 1** and choose **Plot**.

### 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.

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

### MATERIALS

*Material 3 (mat3)*

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

*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 Boundary**.

*Material 3 (mat3)*

1 In the **Model Builder** window, click **Material 3 (mat3)**.

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

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

4 From the **Selection** list, choose **All boundaries**.

5 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics** > **Fatigue Behavior** > **Stress-Based** > **Findley** > **Limit factor (f_Findley)**.

6 Click **Add to Material**.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Normal stress sensitivity coefficient | k_Findley | 0.30 | 1 | Findley |
| Limit factor | f_Findley | 84[MPa] | Pa | Findley |

8 Right-click **Material 3 (mat3)** and choose **Move Up**.

## 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 **Bead**.

*P-norm 1 (pnorm1)*

1 In the **Definitions** toolbar, click **Physics Utilities** and choose **P-norm**.

2 In the **Settings** window for **P-norm**, type `obj` in the **Name** text field.

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

4 From the **Selection** list, choose **Free Shape Boundaries**.

5 Click **Replace Expression** in the upper-right corner of the **P-norm** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **comp1.solid.mises - von Mises stress - N/m²**.

6 Locate the **P-norm** section. From the *p* list, choose **User defined**.

7 In the **Norm** text field, type `pExp`.

*Analytic 1 (an1)*

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

2 In the **Settings** window for **Analytic**, type `loadDistr` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `(min(abs(atan2(x,y)-z),abs(2*pi+atan2(x,y)-z))<pi/6)*cos(3*(atan2(x,y)-z))`.

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

5 Locate the **Units** section. In the **Function** text field, type `Pa`.

6 In the table, enter the following settings:

| Argument | Unit |
|---|---|
| x | m |
| y | m |
| z | rad |

*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 |

*Cylindrical System 2 (sys2)*

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

## 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 Boundary**.

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

5 In the $d_s$ text field, type `1[cm]`.

*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 **Inflation**.

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

5 In the $p$ text field, type `pInflation`.

*Boundary Load 2*

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 **Bead**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

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 |

## FATIGUE (FTG)

*Stress-Based 1*

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

2 In the **Settings** window for **Stress-Based**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Fatigue Boundaries**.

4 Locate the **Solution Field** section. From the **Physics interface** list, choose **Solid Mechanics (solid)**.

## 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 1 (comp1)**, clear the checkbox for **Fatigue (ftg)**.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **Add**.

6 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| phiLoad (Peak load angle) | `range(0,angleStep, 180[deg])` | deg |

7 From the **Run continuation for** list, choose **No parameter**.

Compute the initial stiffness to get a baseline for the optimization.

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

9 In the **Settings** window for **Study**, type `Study 1: Initial Design` in the **Label** text field.

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

### 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 **Preset Studies for Selected Physics Interfaces** > **Fatigue** > **Fatigue**.

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

### STUDY 2

*Step 1: Fatigue*

1 In the **Settings** window for **Fatigue**, locate the **Values of Dependent Variables** section.

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: Initial Design, Stationary**.

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

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

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

### RESULTS

*Objective and Constraints*

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Objective and Constraints` in the **Label** text field.

*Global Evaluation 1*

1 Right-click **Objective and Constraints** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| obj | N/m^2 | Initial objective |

4 In the **Objective and Constraints** toolbar, click **Evaluate**.

### OBJECTIVE AND CONSTRAINTS

1 Go to the **Objective and Constraints** window.

The evaluation indicates that the stress concentration is worst for the first two angles, so it may be possible to restrict the optimization to these.

### COMPONENT 1 (COMP1)

*Free Shape Domain 1*

1 In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization**.

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

3 From the **Selection** list, choose **Free Shape Domain**.

*Free Shape Boundary 1*

1 In the **Shape Optimization** toolbar, click **Free Shape Boundary**.

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

3 From the **Selection** list, choose **Free Shape Boundaries**.

4 Locate the **Control Variable Settings** section. In the  text field, type `2[mm]`.

*Symmetry/Roller 1*

1 In the **Shape Optimization** toolbar, click **Symmetry/Roller**.

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

3 From the **Selection** list, choose **Symmetry Boundaries**.

*Sector Symmetry 1*

1 In the **Shape Optimization** toolbar, click **Sector Symmetry**.

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

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

### ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

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

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

### STUDY 3

*Step 1: Stationary*

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

2 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Fatigue (ftg)**.

3 Locate the **Study Extensions** section. 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) | `0 angleStep` | deg |

Now verify that the stress concentration is not worse for some of the other angles in the optimized geometry.

*Shape Optimization*

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 In the **Move limits** text field, type `0.2`.

4 In the **Maximum number of iterations** text field, type `10`.

5 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **comp1.obj - P-norm - N/m²**.

6 Locate the **Objective Function** section. From the **Solution** list, choose **Maximum of objectives**.

7 Find the **Objective settings** subsection. From the **Objective scaling** list, choose **Initial solution based** to scale the objective. Do the same for the constraints.

8 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Free Shape Domain 1** > **comp1.fsd1.relVolume - Material volume divided by geometry volume - 1**.

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

| Expression | Lower bound | Upper bound |
|---|---|---|
| `comp1.fsd1.relVolume` | | 1 |
| `comp1.solid.Ws_tot/withsol('sol1', comp1.solid.Ws_tot,setval(phiLoad, phiLoad))` | 1 | |

Initialize the solution to generate a default plot that can be shown while optimizing.

10 In the **Study** toolbar, click **Get Initial Value**.

11 In the **Model Builder** window, click **Shape Optimization**.

12 In the **Settings** window for **Shape Optimization**, click to expand the **Output** section.

13 Select the **Plot** checkbox.

14 From the **Plot group** list, choose **Shape Optimization**.

*Solution 3 (sol3)*

1 In the **Model Builder** window, expand the **Study 3** > **Solver Configurations** node.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 3** > **Solver Configurations** > **Solution 3 (sol3)** > **Optimization Solver 1** > **Stationary Solver 1** > **Segregated 1** node, then click **Solid Mechanics**.

4 In the **Settings** window for **Segregated Step**, locate the **General** section.

5 From the **Linear solver** list, choose **Suggested Iterative Solver (solid)** to reduce the memory consumption.

6 In the **Model Builder** window, expand the **Study 3** > **Solver Configurations** > **Solution 3 (sol3)** > **Optimization Solver 1** > **Stationary Solver 1** > **Suggested Iterative Solver (solid)** node, then click **Multigrid 1**.

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

8 From the **Solver** list, choose **Smoothed aggregation AMG**.

## STUDY 1: INITIAL DESIGN

*Step 1: Stationary*

1 In the **Model Builder** window, under **Study 1: Initial Design** 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 1 (comp1)**, clear the checkbox for **Deformed Geometry**.

### STUDY 3: OPTIMIZATION

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3: Optimization` in the **Label** text field.

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

### 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 **Fatigue (ftg)**.

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

### STUDY 4

*Step 1: Stationary*

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

2 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Shape Optimization** and **Deformed Geometry**.

3 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**.

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

5 From the **Study** list, choose **Study 3: Optimization, Stationary**.

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

7 Click **Add**.

8 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| phiLoad (Peak load angle) | `range(0,angleStep, 180[deg])` | `deg` |

9 In the **Model Builder** window, click **Study 4**.

10 In the **Settings** window for **Study**, type `Study 4: Verification` in the **Label** text field.

11 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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

### ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Fatigue** > **Fatigue**.

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

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

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

### STUDY 5

*Step 1: Fatigue*

1 In the **Settings** window for **Fatigue**, locate the **Values of Dependent Variables** section.

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 4: Verification, Stationary**.

5 In the **Model Builder** window, click **Study 5**.

6 In the **Settings** window for **Study**, type `Study 5: Optimized Fatigue` in the **Label** text field.

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

### RESULTS

*Shape Optimization 1*

In the **Model Builder** window, under **Results** right-click **Shape Optimization 1** and choose **Delete**.

*Global Evaluation 2*

1 In the **Model Builder** window, under **Results** > **Objective and Constraints** right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 4: Verification/Solution 4 (sol4)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `obj` | `N/m^2` | `Optimized objective` |
| `solid.Ws_tot/withsol('sol1', solid.Ws_tot,setval(phiLoad, phiLoad))` | `1` | `Optimized relative compliance` |

5 In the **Objective and Constraints** toolbar, click **Evaluate**.

The two first angles still correspond to the worst objectives, and the compliance has not increased for any of the angles, so it was sufficient to consider the first two angles in the optimization.

*Fatigue Comparison*

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Fatigue Comparison` in the **Label** text field.

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

*Surface Maximum 1*

1 Right-click **Fatigue Comparison** and choose **Maximum** > **Surface Maximum**.

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

3 From the **Selection** list, choose **Free Shape Boundaries**.

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Fatigue** > **ftg.fus - Fatigue usage factor - 1**.

*Surface Maximum 2*

1 Right-click **Surface Maximum 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 5: Optimized Fatigue/Solution 5 (sol5)**.

4 In the **Fatigue Comparison** toolbar, click **Evaluate**.

*Fatigue Comparison*

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

2 In the **Settings** window for **3D Plot Group**, type `Fatigue Comparison` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

*Surface 1*

1 Right-click **Fatigue Comparison** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Fatigue** > **ftg.fus - Fatigue usage factor - 1**.

3 Locate the **Expression** section. In the **Expression** text field, type `withsol('sol2', ftg.fus)`.

*Selection 1*

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

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

3 From the **Selection** list, choose **Free Shape Boundaries**.

*Line 1*

1 In the **Model Builder** window, right-click **Fatigue Comparison** and choose **Line**.

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

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

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

*Selection 1*

1 Right-click **Line 1** and choose **Selection**.

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

3 From the **Selection** list, choose **External Free Shape Edges**.

*Surface 2*

1 In the **Model Builder** window, under **Results** > **Fatigue Comparison** right-click **Surface 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `withsol('sol5',ftg.fus)`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

*Transformation 1*

1 Right-click **Surface 2** and choose **Transformation**.

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

3 In the **Z** text field, type `0.05`.

*Surface 2*

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

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

3 From the **Dataset** list, choose **Study 3: Optimization/Solution 3 (sol3)**.

*Line 2*

1 In the **Model Builder** window, under **Results** > **Fatigue Comparison** right-click **Line 1** and choose **Duplicate**.

2 In the **Settings** window for **Line**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3: Optimization/Solution 3 (sol3)**.

*Transformation 1*

In the **Model Builder** window, under **Results** > **Fatigue Comparison** > **Surface 2** right-click **Transformation 1** and choose **Copy**.

*Transformation 1*

In the **Model Builder** window, right-click **Line 2** and choose **Paste Transformation**.

*Line 3*

1 Right-click **Line 2** and choose **Duplicate**.

2 In the **Model Builder** window, click **Line 3**.

3 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

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

*Deformation 1*

1 Right-click **Line 3** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Frames** > **Material** > **material.dX,...,material.dZ - Material mesh displacement (geometry frame)**.

3 Locate the **Expression** section. In the **X-component** text field, type `-material.dX`.

4 In the **Y-component** text field, type `-material.dY`.

5 In the **Z-component** text field, type `-material.dZ`.

6 Locate the **Scale** section.

7 Select the **Scale factor** checkbox. In the associated text field, type `1`.

8 In the **Fatigue Comparison** toolbar, click **Plot**.

*STL Export*

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

2 In the **Settings** window for **3D Plot Group**, type `STL Export` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Optimization/ Solution 3 (sol3)**.

*Volume 1*

1 Right-click **STL Export** and choose **Volume**.

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

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

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

6 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

7 From the **Smoothing** list, choose **None**.

*Filter 1*

1 Right-click **Volume 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `centroid((pi/2-pi/5<atan2(Y, X))*(atan2(Y,X)<pi/2))`.

*Volume 1*

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

2 In the **STL Export** toolbar, click **Plot**.

*Plot 1*

1 Right-click **Results** > **STL Export** > **Volume 1** and choose **Add Plot Data to Export**.

2 In the **Settings** window for **Plot**, locate the **Output** section.

3 From the **File type** list, choose **STL binary file (*.stl)**.

4 In the **Filename** text field, type `rim_fatigue_optimization.stl`.

## *Geometry Modeling Instructions*

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

**NEW**

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

## 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 | 2 | 2 | Number of load positions in first sector |
| angleStep | 360[deg]/(5*numLpos) | 0.62832 rad | Step in peak load angle [deg] |
| Ws0 | 10.5[mJ] | 0.0105 J | Characteristic elastic energy |
| pExp | 20 | 20 | P-norm exponent |
| sigmaMax | 100[MPa] | 1E8 Pa | Characteristic stress |
| rDesign | 16.5[cm] | 0.165 m | Design domain radius |
| angleLast | angleStep*(numLpos-1) | 0.62832 rad | Last peak load angle [deg] |
| index | 0 | 0 | |

## ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

## GEOMETRY 1

*Import 1 (imp1)*

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

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

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

4 From the **Source** list, choose **COMSOL Multiphysics file**.

5 Click **Browse**.

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

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 Locate the **Source** section. Click **null**.

*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 In the **z-coordinate** text field, type `-15[cm]`.

*Work Plane 1 (wp1) > Plane Geometry*

In the **Model Builder** window, click **Plane Geometry**.

*Work Plane 1 (wp1) > Circle 1 (c1)*

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

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

3 In the **Sector angle** text field, type 36.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type 54.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

*Extrude 1 (ext1)*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** checkbox.

*Objects to Intersect*

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

2 In the **Settings** window for **Complement Selection**, type `Objects to Intersect` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

*Intersection 1 (int1)*

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

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

3 From the **Input objects** list, choose **Objects to Intersect**.

*Cylinder 1 (cyl1)*

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

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

3 In the **Radius** text field, type `2[cm]`.

4 In the **Height** text field, type `1.5[cm]`.

5 Locate the **Position** section. In the **y** text field, type `6.7[cm]`.

6 In the **z** text field, type `7[cm]`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 Click **Build Selected**.

*Cylinder 2 (cyl2)*

1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `rDesign`.

4 In the **Height** text field, type `1[m]`.

5 Locate the **Position** section. In the **y** text field, type `0`.

6 In the **z** text field, type `0`.

7 Click **Build Selected**.

*Tools Objects*

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

2 In the **Settings** window for **Union Selection**, type `Tools Objects` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Cylinder 1** and **Cylinder 2**.

6 Click **OK**.

*Partition Objects 1 (par1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

3 From the **Objects to partition** list, choose **Objects to Intersect**.

4 Click to select the **Activate Selection** toggle button for **Tool objects**.

5 From the **Tool objects** list, choose **Tools Objects**.

6 Click **Build Selected**.

*Symmetry Boundary 1*

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

2 In the **Settings** window for **Box Selection**, type `Symmetry Boundary 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x maximum** text field, type `eps`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

*Symmetry Boundary 2*

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

2 In the **Settings** window for **Cylinder Selection**, type `Symmetry Boundary 2` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

5 In the **Start angle** text field, type `53`.

6 In the **End angle** text field, type `55`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Outer Boundaries*

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

2 In the **Settings** window for **Cylinder Selection**, type `Outer Boundaries` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

4 In the **Inner radius** text field, type `rDesign*0.99`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

6 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

*Inner Boundaries*

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

2 In the **Settings** window for **Cylinder Selection**, type `Inner Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `3.6[cm]`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Fixed Boundary*

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

2 In the **Settings** window for **Cylinder Selection**, type `Fixed Boundary` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.81[cm]`.

5 In the **Inner radius** text field, type `1.09[cm]`.

6 In the **Top distance** text field, type `8[cm]`.

7 Locate the **Position** section. In the **y** text field, type `6.7[cm]`.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

9 Click **Build Selected**.

*Nondesign Boundaries 1*

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

2 In the **Settings** window for **Cylinder Selection**, type `Nondesign Boundaries 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.5[cm]`.

5 Locate the **Position** section. In the **y** text field, type `6.7[cm]`.

6 Click **Build Selected**.

*Nondesign Boundaries 2*

1 Right-click **Nondesign Boundaries 1** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Nondesign Boundaries 2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Outer radius** text field, type `2.01[cm]`.

4 In the **Inner radius** text field, type `1.79[cm]`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

6 Click **Build Selected**.

*Critical Fillet Boundary*

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

2 In the **Settings** window for **Cylinder Selection**, type `Critical Fillet Boundary` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `9.5[cm[`.

5 Clear the **Inner radius** text field.

6 In the **Outer radius** text field, type `9.5[cm]`.

7 In the **Inner radius** text field, type `9.4[cm]`.

8 In the **Top distance** text field, type `9[cm]`.

9 In the **Start angle** text field, type `70`.

10 In the **End angle** text field, type `80`.

*Merge Candidates*

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

2 In the **Settings** window for **Cylinder Selection**, type `Merge Candidates` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

5 In the **Start angle** text field, type `70`.

6 In the **End angle** text field, type `91`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Mirror 1 (mir1)*

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 In the **Settings** window for **Mirror**, locate the **Input** section.

3 From the **Input objects** list, choose **Import 1**.

4 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type `1`.

5 In the **z** text field, type `0`.

*Nonsymmetry Boundaries*

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

2 In the **Settings** window for **Complement Selection**, type `Nonsymmetry Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to invert** list, choose **Symmetry Boundary 1** and **Symmetry Boundary 2**.

6 Click **OK**.

*Mirror Boundaries*

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

2 In the **Settings** window for **Cylinder Selection**, type `Mirror Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Nonsymmetry Boundaries** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, locate the **Size and Shape** section.

9 In the **Outer radius** text field, type `Inf`.

10 In the **Start angle** text field, type `89`.

11 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Nonmirror Domains*

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

2 In the **Settings** window for **Cylinder Selection**, type `Nonmirror Domains` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.01*rDesign`.

4 In the **End angle** text field, type `91`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

6 Click **Build Selected**.

*Mirror 1 (mir1)*

1 In the **Model Builder** window, click **Mirror 1 (mir1)**.

2 In the **Settings** window for **Mirror**, locate the **Input** section.

3 Click to select the **Activate Selection** toggle button for **Input objects**.

4 Select the **Keep input objects** checkbox.

5 Click **Build Selected**.

*Nonmirror Domains (cylsel10)*

1 In the **Model Builder** window, click **Nonmirror Domains (cylsel10)**.

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

*Mirror Domains*

1 Right-click **Component 1 (comp1)** > **Geometry 1** > **Nonmirror Domains (cylsel10)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Mirror Domains` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `89`.

4 In the **End angle** text field, type `360`.

*Nonmirror Boundaries*

1 Right-click **Nonmirror Domains (cylsel10)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Nonmirror Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Build Preceding State**.

6 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

7 Locate the **Input Entities** section. Click **Add**.

8 In the **Add** dialog, select **Nonsymmetry Boundaries** in the **Selections** list.

9 Click **OK**.

*Rotate 1 (rot1)*

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Right-click **Rotate 1 (rot1)** and choose **Move Down**.

3 In the **Settings** window for **Rotate**, locate the **Input** section.

4 Click to select the **Activate Selection** toggle button for **Input objects**.

5 From the **Input objects** list, choose **Import 1**.

6 Locate the **Rotation** section. In the **Angle** text field, type `range(72,72,360)`.

7 Click **Build Selected**.

*Design Boundaries*

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

2 In the **Settings** window for **Complement Selection**, type `Design Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to invert** list, choose **Cylinder 1**, **Symmetry Boundary 1**, **Symmetry Boundary 2**, **Outer Boundaries**, **Fixed Boundary**, **Nondesign Boundaries 1**, and **Nondesign Boundaries 2**.

6 Click **OK**.

*Boundaries to Merge*

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

2 In the **Settings** window for **Difference Selection**, type `Boundaries to Merge` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Design Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Inner Boundaries** and **Critical Fillet Boundary**.

10 Click **OK**.

*Free Shape Domain*

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

2 In the **Settings** window for **Adjacent Selection**, type `Free Shape Domain` in the **Label** text field.

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

4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent domains**.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog, select **Design Boundaries** in the **Input selections** list.

7 Click **OK**.

*Nondesign Domains*

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

2 In the **Settings** window for **Cylinder Selection**, type `Nondesign Domains` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

5 In the **Start angle** text field, type `89`.

6 In the **End angle** text field, type `415`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

8 Locate the **Input Entities** section. Click **Add**.

9 In the **Add** dialog, select **Free Shape Domain** in the **Selections** list.

10 Click **OK**.

*Design Domains*

1 Right-click **Nondesign Domains** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Design Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Build Preceding State**.

4 Locate the **Size and Shape** section. In the **Start angle** text field, type `53`.

5 In the **End angle** text field, type `91`.

*Design Sector Boundaries*

1 Right-click **Design Domains** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Design Sector Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Selections** list box, select **Free Shape Domain**.

4 Click **Delete**.

5 From the **Entities** list, choose **All**.

6 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

*Free Shape Boundaries*

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

2 In the **Settings** window for **Intersection Selection**, type `Free Shape Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

*Keep Boundaries*

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

2 In the **Settings** window for **Cylinder Selection**, type `Keep Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.2*rDesign`.

5 In the **Inner radius** text field, type `0.99*rDesign`.

6 In the **Top distance** text field, type `0.1`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

8 Click **Build Selected**.

*Form Union (fin)*

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

*Form Composite Faces 1 (cmf1)*

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 From the **Faces to composite** list, choose **Boundaries to Merge**.

4 Click **Build Selected**.

*Symmetry Boundaries*

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

2 In the **Settings** window for **Union Selection**, type `Symmetry Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Symmetry Boundary 1** and **Symmetry Boundary 2**.

6 Click **OK**.

*Form Composite Faces 2 (cmf2)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Form Composite Faces 1 (cmf1)** and choose **Duplicate**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 Click to select the **Activate Selection** toggle button for **Faces to composite**.

4 From the **Faces to composite** list, choose **Nondesign Boundaries 2**.

5 Click **Build Selected**.

*Boundaries to Merge 2*

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

2 In the **Settings** window for **Difference Selection**, type `Boundaries to Merge 2` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Merge Candidates** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Fixed Boundary**, **Design Boundaries**, **Keep Boundaries**, and **Symmetry Boundaries**.

10 Click **OK**.

11 In the **Settings** window for **Difference Selection**, click **Build Selected**.

*Form Composite Faces 3 (cmf3)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Form Composite Faces 2 (cmf2)** and choose **Duplicate**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 From the **Faces to composite** list, choose **Boundaries to Merge 2**.

4 Click **Build Selected**.

*Destination Boundaries*

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

2 In the **Settings** window for **Difference Selection**, type `Destination Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Design Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Free Shape Boundaries** and **Symmetry Boundaries**.

10 Click **OK**.

11 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

12 In the **Selections to add** list box, select **Design Boundaries**.

13 In the **Selections to subtract** list box, select **Free Shape Boundaries**.

*Free Shape Boundaries (intsel1)*

1 In the **Model Builder** window, click **Free Shape Boundaries (intsel1)**.

2 In the **Settings** window for **Intersection Selection**, locate the **Input Entities** section.

3 Click **Build Preceding State**.

4 Click **Add**.

5 In the **Add** dialog, in the **Selections to intersect** list, choose **Design Boundaries** and **Design Sector Boundaries**.

6 Click **OK**.

*Destination Boundaries (difsel3)*

1 In the **Model Builder** window, click **Destination Boundaries (difsel3)**.

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

*Source Critical Fillet Boundary*

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

2 In the **Settings** window for **Cylinder Selection**, type `Source Critical Fillet Boundary` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Critical Fillet Boundary** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, locate the **Size and Shape** section.

9 In the **Outer radius** text field, type `Inf`.

10 In the **Start angle** text field, type `53`.

11 In the **End angle** text field, type `90`.

12 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

13 Click **Build Selected**.

*Source Design Domain*

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

2 In the **Settings** window for **Cylinder Selection**, type `Source Design Domain` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.01*rDesign`.

4 In the **Start angle** text field, type `51`.

5 In the **End angle** text field, type `91`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Source Critical Fillet Boundary (cylsel17)*

1 In the **Model Builder** window, click **Source Critical Fillet Boundary (cylsel17)**.

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

3 In the **End angle** text field, type `91`.

4 Click **Build Selected**.

*Source Design Domain (cylsel18)*

1 In the **Model Builder** window, click **Source Design Domain (cylsel18)**.

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

*Fatigue Boundaries*

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

2 In the **Settings** window for **Difference Selection**, type `Fatigue Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Design Sector Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the + **Add** button for **Selections to subtract**.

9 In the **Add** dialog, in the **Selections to subtract** list, choose **Outer Boundaries** and **Nondesign Boundaries 1**.

10 Click **OK**.

*External Free Shape Edges*

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

2 In the **Settings** window for **Adjacent Selection**, type `External Free Shape Edges` in the **Label** text field.

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

4 Click + **Add**.

5 In the **Add** dialog, select **Free Shape Boundaries** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

8 From the **Geometric entity level** list, choose **Adjacent edges**.

*Hub Edges YZ*

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

2 In the **Settings** window for **Box Selection**, type `Hub Edges YZ` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-0.001`.

5 In the **x maximum** text field, type `0.001`.

6 In the **y maximum** text field, type `0.1`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

8 Locate the **Box Limits** section. In the **y minimum** text field, type `0`.

*Hub Edges 1*

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

2 In the **Settings** window for **Cylinder Selection**, type `Hub Edges 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `0.1`.

5 In the **Start angle** text field, type `53`.

6 In the **End angle** text field, type `55`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Unmapped and Mapped Boundaries*

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

2 In the **Settings** window for **Cylinder Selection**, type `Unmapped and Mapped Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.5*rDesign`.

5 In the **Inner radius** text field, type `0.9*rDesign`.

6 In the **Bottom distance** text field, type `0.05`.

7 In the **Start angle** text field, type `53`.

8 In the **End angle** text field, type `91`.

9 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Intersection Input 1*

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

2 In the **Settings** window for **Cylinder Selection**, type `Intersection Input 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Inf`.

5 In the **Inner radius** text field, type `1.3*rDesign`.

6 In the **Top distance** text field, type `0.11`.

7 In the **Bottom distance** text field, type `0.05`.

8 In the **Start angle** text field, type `88`.

9 In the **End angle** text field, type `89`.

*Intersection Input 2*

1 Right-click **Intersection Input 1** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Intersection Input 2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `55`.

4 In the **End angle** text field, type `56`.

5 Click **Build Selected**.

*Mapped Boundaries 2*

1 In the **Geometry** toolbar, click **Selections** and choose **Intersection Selection**.
2 In the **Settings** window for **Intersection Selection**, type `Mapped Boundaries 2` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Click **Add**.
5 In the **Add** dialog, in the **Selections to intersect** list, choose **Intersection Input 1** and **Intersection Input 2**.
6 Click **OK**.

*Unmapped Boundaries*

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.
2 In the **Settings** window for **Difference Selection**, type `Unmapped Boundaries` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.
5 In the **Add** dialog, select **Unmapped and Mapped Boundaries** in the **Selections to add** list.
6 Click **OK**.
7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.
8 Click the **Add** button for **Selections to subtract**.
9 In the **Add** dialog, select **Mapped Boundaries 2** in the **Selections to subtract** list.
10 Click **OK**.

*Center Hub Edges*

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Center Hub Edges` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
4 Locate the **Box Limits** section. In the **x minimum** text field, type `-1e-6*rDesign`.
5 In the **x maximum** text field, type `1e-6*rDesign`.
6 In the **y minimum** text field, type `0`.
7 In the **y maximum** text field, type `0.04`.
8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

*Center Rim Edges*

1 Right-click **Center Hub Edges** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Center Rim Edges` in the **Label** text field.

3 Locate the **Box Limits** section. In the **y minimum** text field, type `0.1`.

4 In the **y maximum** text field, type `Inf`.

5 Click **Build Selected**.

*Center Rim Boundaries*

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

2 In the **Settings** window for **Box Selection**, type `Center Rim Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `0.001*rDesign`.

5 In the **x maximum** text field, type `0.002*rDesign`.

6 In the **y minimum** text field, type `0.1`.

*Center Hub Boundaries*

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

2 In the **Settings** window for **Cylinder Selection**, type `Center Hub Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `0.04`.

5 In the **Start angle** text field, type `53`.

6 In the **End angle** text field, type `91`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

*Mapped Boundaries*

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

2 In the **Settings** window for **Difference Selection**, type `Mapped Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Center Rim Boundaries** and **Center Hub Boundaries**.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the + **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Unmapped Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

*Mapped Edges 1*

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

2 In the **Settings** window for **Adjacent Selection**, type `Mapped Edges 1` in the **Label** text field.

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

4 Click + **Add**.

5 In the **Add** dialog, select **Mapped Boundaries 2** in the **Input selections** list.

6 Click **OK**.

7 In the **Model Builder** window, click **Mapped Edges 1 (adjsel3)**.

8 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

9 From the **Geometric entity level** list, choose **Adjacent edges**.

10 Locate the **Input Entities** section. Click + **Add**.

11 In the **Add** dialog, select **Mapped Boundaries** in the **Input selections** list.

12 Click **OK**.

*Mapped Edges*

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

2 In the **Settings** window for **Union Selection**, type `Mapped Edges` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Click + **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Center Hub Edges** and **Center Rim Edges**.

6 Click **OK**.

*Long Mapped Edges*

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

2 In the **Settings** window for **Box Selection**, type `Long Mapped Edges` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Center Rim Edges** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box Selection**, locate the **Box Limits** section.

9 In the **z minimum** text field, type `0`.

10 In the **z maximum** text field, type `0.01*rDesign`.

11 Click **Build Selected**.

*Exterior Boundaries*

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

2 In the **Settings** window for **Adjacent Selection**, type `Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog, select **Import 1** in the **Input selections** list.

5 Click **OK**.

*Hole 1*

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

2 In the **Settings** window for **Cylinder Selection**, type `Hole 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Exterior Boundaries** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, locate the **Size and Shape** section.

9 In the **Outer radius** text field, type `1.5[cm]`.

10 In the **Bottom distance** text field, type `0.08`.

11 In the **Start angle** text field, type `91`.

12 In the **End angle** text field, type `169`.

13 Locate the **Position** section. In the **x** text field, type `6.7[cm]*sin(2*pi/5*0)`.

14 In the **y** text field, type `6.7[cm]*cos(2*pi/5*0)`.

15 Click **Build Selected**.

*Hole 2*

1. Right-click **Hole 1** and choose **Duplicate**.
2. In the **Settings** window for **Cylinder Selection**, type `Hole 2` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Start angle** text field, type `0`.
4. In the **End angle** text field, type `360`.
5. Locate the **Position** section. In the **x** text field, type `6.7[cm]*sin(2*pi/5*1)`.
6. In the **y** text field, type `6.7[cm]*cos(2*pi/5*1)`.

*Hole 3*

1. Right-click **Hole 2** and choose **Duplicate**.
2. In the **Settings** window for **Cylinder Selection**, type `Hole 3` in the **Label** text field.
3. Locate the **Position** section. In the **x** text field, type `6.7[cm]*sin(2*pi/5*2)`.
4. In the **y** text field, type `6.7[cm]*cos(2*pi/5*2)`.

*Hole 4*

1. Right-click **Hole 3** and choose **Duplicate**.
2. In the **Settings** window for **Cylinder Selection**, type `Hole 4` in the **Label** text field.
3. Locate the **Position** section. In the **x** text field, type `6.7[cm]*sin(2*pi/5*3)`.
4. In the **y** text field, type `6.7[cm]*cos(2*pi/5*3)`.

*Hole 5*

1. Right-click **Hole 4** and choose **Duplicate**.
2. In the **Settings** window for **Cylinder Selection**, type `Hole 5` in the **Label** text field.
3. Locate the **Position** section. In the **x** text field, type `6.7[cm]*sin(2*pi/5*4)`.
4. In the **y** text field, type `6.7[cm]*cos(2*pi/5*4)`.

*Holes*

1. In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.
2. In the **Settings** window for **Union Selection**, type `Holes` in the **Label** text field.
3. Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4. Locate the **Input Entities** section. Click **Add**.
5. In the **Add** dialog, in the **Selections to add** list, choose **Hole 1**, **Hole 2**, **Hole 3**, **Hole 4**, and **Hole 5**.
6. Click **OK**.

*Inflation 1*

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

2 In the **Settings** window for **Cylinder Selection**, type `Inflation 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `24[cm]`.

5 In the **Inner radius** text field, type `23[cm]`.

6 In the **Top distance** text field, type `-8.8[cm]`.

7 In the **Bottom distance** text field, type `-11[cm]`.

8 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

9 Click **Add**.

10 In the **Add** dialog, select **Exterior Boundaries** in the **Selections** list.

11 Click **OK**.

12 In the **Settings** window for **Cylinder Selection**, click **Build Selected**.

*Inflation 2*

1 Right-click **Inflation 1** and choose **Duplicate**.

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

3 Locate the **Size and Shape** section. In the **Top distance** text field, type `11[cm]`.

4 In the **Bottom distance** text field, type `9[cm]`.

5 Click **Build Selected**.

*Bead*

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Inflation 1** and **Inflation 2**.

6 Click **OK**.

*Inflation 3*

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

2 In the **Settings** window for **Cylinder Selection**, type `Inflation 3` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `25[cm]`.

5 In the **Inner radius** text field, type `21.5[cm]`.

6 In the **Top distance** text field, type `9.5[cm]`.

7 In the **Bottom distance** text field, type `-9[cm]`.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.

9 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

10 Click **Add**.

11 In the **Add** dialog, select **Exterior Boundaries** in the **Selections** list.

12 Click **OK**.

13 In the **Settings** window for **Cylinder Selection**, click **Build Selected**.

*Inflation*

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Inflation 1**, **Inflation 2**, and **Inflation 3**.

6 Click **OK**.

*Edges*

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

2 In the **Settings** window for **Difference Selection**, type `Edges` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **External Free Shape Edges**, **Hub Edges YZ**, and **Hub Edges 1**.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Mapped Edges 1** in the **Selections to subtract** list.

10 Click **OK**.

11 In the **Settings** window for **Difference Selection**, click **Build Selected**.

12 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Edges (difsel7)** and choose **Group**.

*Mesh*

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

*Symmetry Edges*

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.
2 In the **Settings** window for **Adjacent Selection**, type `Symmetry Edges` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent edges**.
5 Clear the **Interior edges** checkbox.
6 Locate the **Input Entities** section. Click **Add**.
7 In the **Add** dialog, in the **Input selections** list, choose **Symmetry Boundary 1** and **Symmetry Boundary 2**.
8 Click **OK**.
9 In the **Settings** window for **Adjacent Selection**, click **Build Selected**.

*Partition Edge*

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.
2 In the **Settings** window for **Cylinder Selection**, type `Partition Edge` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `1.01*rDesign`.
5 In the **Inner radius** text field, type `0.99*rDesign`.
6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside cylinder**.
7 Click **Build Selected**.

*Inner Symmetry Edges*

1 Right-click **Partition Edge** and choose **Duplicate**.
2 In the **Settings** window for **Cylinder Selection**, type `Inner Symmetry Edges` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Inner radius** text field, type `0`.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Click **Add**.

6 In the **Add** dialog, select **Symmetry Edges** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Cylinder Selection**, click **Build Selected**.

*Source and Destination Edges*

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

2 In the **Settings** window for **Union Selection**, type `Source and Destination Edges` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Partition Edge** and **Inner Symmetry Edges**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, click **Build Selected**.

*Spoke Edges Source 0*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Mesh** right-click **Partition Edge (cylsel32)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Spoke Edges Source 0` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `54`.

4 In the **End angle** text field, type `90`.

5 Click **Build Selected**.

*Spoke Edges Destination 0*

1 Right-click **Spoke Edges Source 0** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Spoke Edges Destination 0` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `90`.

4 In the **End angle** text field, type `90+18`.

5 Click **Build Selected**.

*Spoke Edges Source 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Mesh** right-click **Inner Symmetry Edges (cylsel33)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Spoke Edges Source 1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `53`.

4 In the **End angle** text field, type `91+18`.

5 Locate the **Input Entities** section. In the **Selections** list box, select **Symmetry Edges**.

6 Click **Delete**.

7 Click **Add**.

8 In the **Add** dialog, select **Source and Destination Edges** in the **Selections** list.

9 Click **OK**.

10 In the **Settings** window for **Cylinder Selection**, click **Build Selected**.

*Spoke Edges Destination 1*

1 Right-click **Spoke Edges Source 1** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Spoke Edges Destination 1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `89+36`.

4 In the **End angle** text field, type `91+18+2*36`.

5 Click **Build Selected**.

*Spoke Edges Source 2*

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

2 In the **Settings** window for **Union Selection**, type `Spoke Edges Source 2` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Spoke Edges Destination 0**, **Spoke Edges Source 1**, and **Spoke Edges Destination 1**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, click **Build Selected**.

*Spoke Edges Destination 2*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Mesh** right-click **Spoke Edges Destination 1 (cylsel37)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Spoke Edges Destination 2` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `89+3*36`.

4 In the **End angle** text field, type `91+18+6*36`.

5 Click **Build Selected**.

*Spoke Edges Destination 3*

1 Right-click **Spoke Edges Destination 2** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder Selection**, type `Spoke Edges Destination 3` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Start angle** text field, type `89+7*36`.

4 In the **End angle** text field, type `91+18+8*36`.

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

The model geometry is now complete.