Created in COMSOL Multiphysics 5.6

Fatigue Analysis of a Wheel Rim

Introduction

During the development of safety critical components like a car wheel rim, making sure fatigue cracks do not occur is one of the most important tasks. When a final prototype is available this is ensured by testing, but prototype production and testing are time consuming and expensive activities. Good predictions from simulations can keep down the number the number of prototypes to a minimum.

In this example, you do a fatigue evaluation on a model of a wheel rim, subjected to the load history from a simulated test.

Model Definition

For a definition of geometry, loads and boundary conditions, see the documentation for the model *Submodel in a Wheel Rim* in the Structural Mechanics Module Application Library.

The fatigue limit (in terms if the stress amplitude) is known for two cases with pure axial loading. For pure tension it is 95 MPa, and for fully reversed loading it is 125 MPa. In this model, you use the Findley criterion, so the Findley parameters have to be derived from these data.

In pure tension, the Findley criterion can be written as

$$\sqrt{\left(\frac{\Delta\sigma}{2}\right)^2+(k\cdot\sigma_{max})^2}+k\cdot\sigma_{max} = 2f$$

This means that you have to solve the simultaneous equations

$$\sqrt{95^2+(k\cdot 190)^2}+k\cdot 190 = 2f$$
$$\sqrt{125^2+(k\cdot 125)^2}+k\cdot 125 = 2f$$

to get the Findley parameters f and k. The result is $f = 84$ MPa and $k = 0.30$.

Results and Discussion

The fatigue usage factor distribution is shown in Figure 1. The maximum value is about 0.67, which should indicate that the design is good when taking into account that the required safety factor has been included in the load. In Figure 2 the stress histories in the critical point are displayed. The loading is slightly nonproportional, and has a compressive mean stress, which is captured by the Findley criterion.

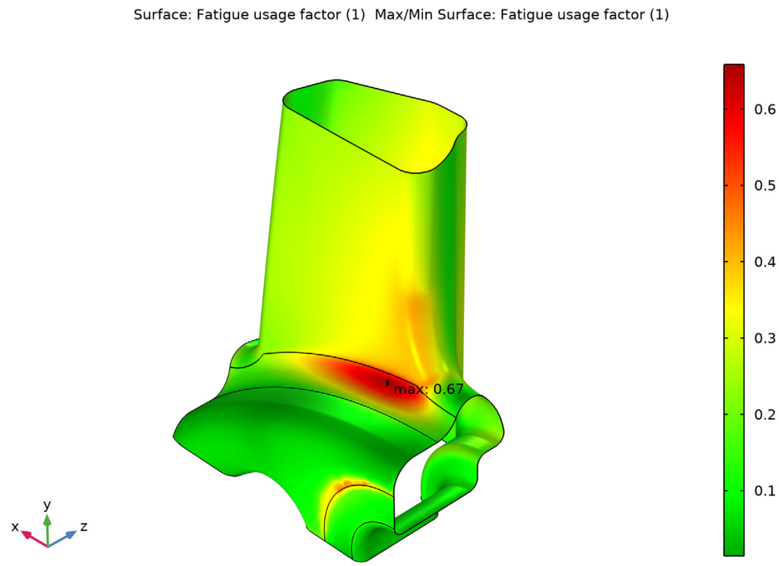

Figure 1: Fatigue usage factor using the Findley criterion.

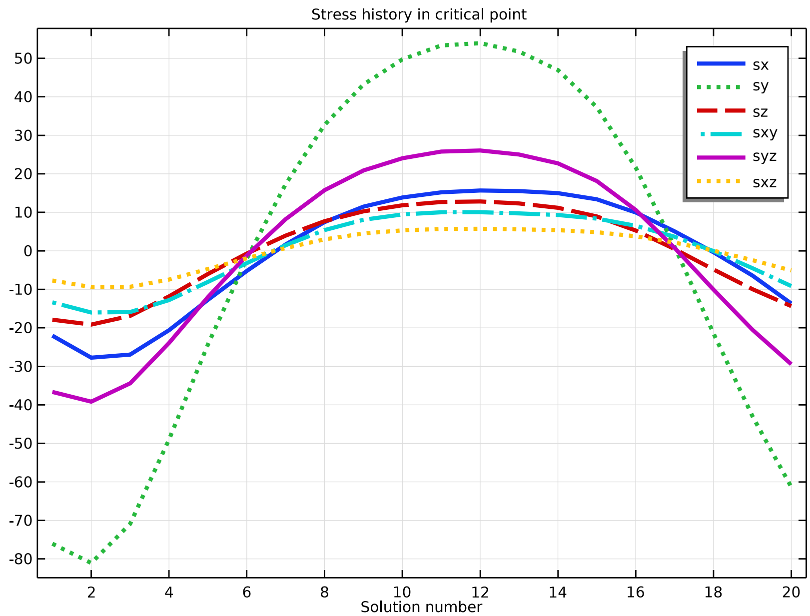

Figure 2: Stress histories in the critical point.

Notes About the COMSOL Implementation

In this example, you perform the fatigue analysis as an additional study step in a model which already contains the results from a stress analysis. Since the critical points for fatigue crack initiation is on the free surface of the body, it is sufficient to do the fatigue evaluation on the boundary, and not in the domain. This reduces CPU and memory requirements significantly.

Application Library path: `Fatigue_Module/Stress_Based/rim_fatigue`

Modeling Instructions

ROOT

In this example you will start from an existing model which is an example in the Structural Mechanics Module.

1 From the **File** menu, choose **Open**.
2 From the Application Libraries root, browse to the folder `Structural_Mechanics_Module/Tutorials` and double-click the file `rim_submodel.mph`.

RESULTS

Stress in Submodel

If the model was stored without solutions, you will now have to run **Study 1** and **Study 2** before continuing.

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>Fatigue (ftg)**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Study 1** and **Study 2**.
5 Click **Add to Component 2** in the window toolbar.
6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

FATIGUE (FTG)

Stress-Based 1

1 Right-click **Component 2 (comp2)>Fatigue (ftg)** and choose the boundary evaluation **Stress-Based**.
2 Select Boundaries 2–5 only.
3 In the **Settings** window for **Stress-Based**, locate the **Solution Field** section.
4 From the **Physics interface** list, choose **Solid Mechanics 2 (solid2)**.

MATERIALS

Material 3 (mat3)

1 In the **Model Builder** window, expand the **Component 2 (comp2)>Materials** node.

2 Right-click **Component 2 (comp2)>Materials** and choose **Blank Material**.

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

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

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

6 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

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Solid Mechanics (solid)** and **Solid Mechanics 2 (solid2)**.

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

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

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

STUDY 3

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 2, Stationary**.

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

RESULTS

Max/Min Surface 1

1 In the **Fatigue Usage Factor (ftg)** toolbar, click **More Plots** and choose **Max/ Min Surface**.

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

3 In the **Expression** text field, type `ftg.fus`.

4 Click to expand the **Advanced** section. Locate the **Display** section. From the **Display** list, choose **Max**.

5 Locate the **Text Format** section. In the **Display precision** text field, type `2`.

The maximum value of fatigue usage factor may be outside the expected area because a singularity can be present on the edge of the boundary where the fixed constraint is applied. Add a selection to ensure to display the maximum value of the surface of interest.

Selection 1

1 In the **Fatigue Usage Factor (ftg)** toolbar, click **Selection**.

2 Select Boundary 5 only.

3 In the **Fatigue Usage Factor (ftg)** toolbar, click **Plot**.

Fatigue Usage Factor (ftg)

1 In the **Model Builder** window, click **Fatigue Usage Factor (ftg)**.

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

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

In order to get the location of the point with maximum fatigue usage, zoom in on the maximum marker and click on it in the graphics window. You will then see the value and the coordinates in the Table window. The location will be approximately (0.016, 0.092, 0.088). Use these coordinates to create a Cut Point 3D dataset for detailed evaluation of the stress history in the critical point.

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Locate the **Point Data** section. In the **X** text field, type `0.0164`.

5 In the **Y** text field, type `0.0924`.

6 In the **Z** text field, type `0.0884`.

7 Select the **Snap to closest boundary** check box.

1D Plot Group 5

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

Point Graph 1

1 Right-click **1D Plot Group 5** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 3D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)>Solid Mechanics 2>Stress>Stress tensor (spatial frame) - N/m²>solid2.sx - Stress tensor, x component**.

5 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

7 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **All solutions**.

8 Locate the **Coloring and Style** section. In the **Width** text field, type `3`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
sx

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid2.sy`.

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

Legends
sy

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid2.sz`.

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

Legends
sz

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid2.sxy`.

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

Legends
sxy

Point Graph 5

1 Right-click **Point Graph 4** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid2.syz`.

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

Legends
syz

Point Graph 6

1 Right-click **Point Graph 5** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid2.sxz`.

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

Legends
sxz

Stress History

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, type `Stress History` in the **Label** text field.

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

4 In the **Title** text area, type `Stress history in critical point`.

5 In the **Stress History** toolbar, click **Plot**.