Shaft with Fillet

Introduction

This benchmark model is based on the example found in section 5.4.3 of Ref. 1. It shows how to perform a high-cycle fatigue analysis for non-proportional loading using critical plane methods.

Model Definition

The geometry is a circular shaft with two different diameters, 10 mm and 16 mm. At the transition between the two diameters there is a fillet with a radius of 2 mm.

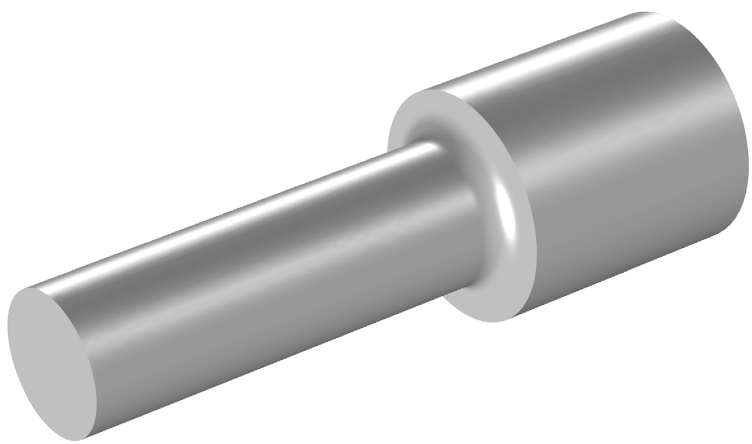

Figure 1: The notched shaft

Two time-dependent loads are applied at the small end of the shaft: a transverse force and a twisting moment. The force varies between 0 and 1.94 kN and the torque varies between −28.7 and +28.7 Nm. Figure 2 shows the history of one loading cycle.

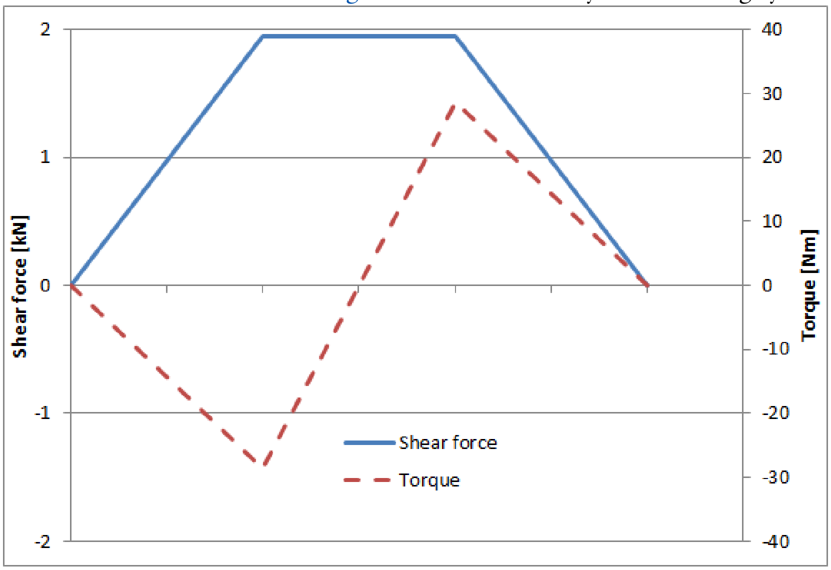

Figure 2: Load history.

The big end of the shaft is fixed. The material is Elastic with E = 100 GPa and $\nu = 0$.

In Ref. 1 it is stated that the fatigue limit for completely reversed axial tension is 700 MPa, while the fatigue limit for pure torsion is 560 MPa. These values are the stress amplitudes.

In pure tension, the Findley criterion can be written as

$$\sqrt{\left(\frac{\Delta\sigma}{2}\right)^2 + (k \cdot \sigma_{max})^2} + k \cdot \sigma_{max} = 2f \tag{1}$$

This means that you have to solve the simultaneous equations

$$\begin{aligned} \sqrt{700^2 + (k \cdot 700)^2} + k \cdot 700 &= 2f \\ \sqrt{560^2 + (k \cdot 1120)^2} + k \cdot 1120 &= 2f \end{aligned} \tag{2}$$

to get the Findley parameters f and k. The result is f = 440 MPa and k = 0.23.

The Matake criterion is similar to the Findley criterion, with the difference that the critical plane is defined solely by the maximum shear stress. For a pure tensile case, the Matake expression is

$$\frac{\Delta\sigma}{4} + k\sigma_{\max} = f \tag{3}$$

which gives the corresponding system of equations as

$$\begin{aligned} 350 + k \cdot 700 &= f \\ 280 + k \cdot 1120 &= f \end{aligned} \tag{4}$$

The solution is $f = 466$ MPa and $k = 0.17$ as parameters for the Matake case.

Results and Discussion

Figure 3 and Figure 4 show the stress distribution from the two basic load cases. The location for the maximum effective stress is at the surface of the fillet, at a radius slightly larger than the minimum radius of the shaft.

In Figure 5 the effective stress from the combined load case with transverse force and positive torque is shown. It is symmetric with respect to the XY-plane, and is identical also for the case when the torque is reversed.

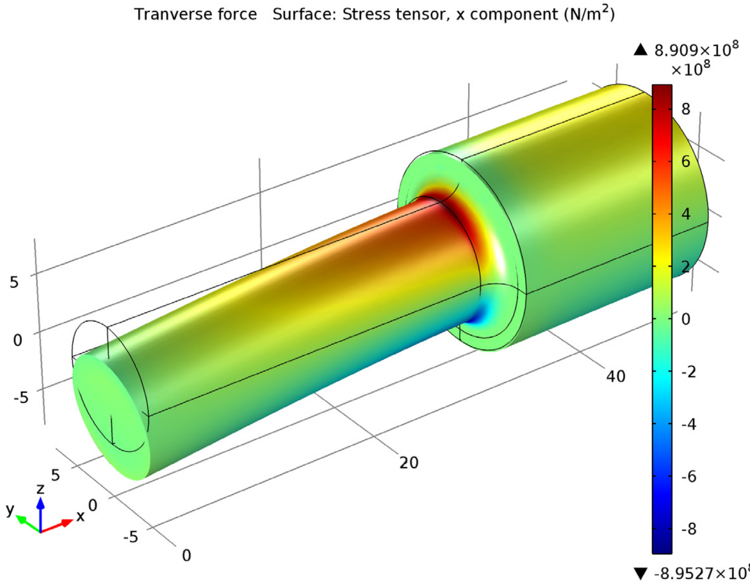

Figure 3: Axial stress from transverse force.

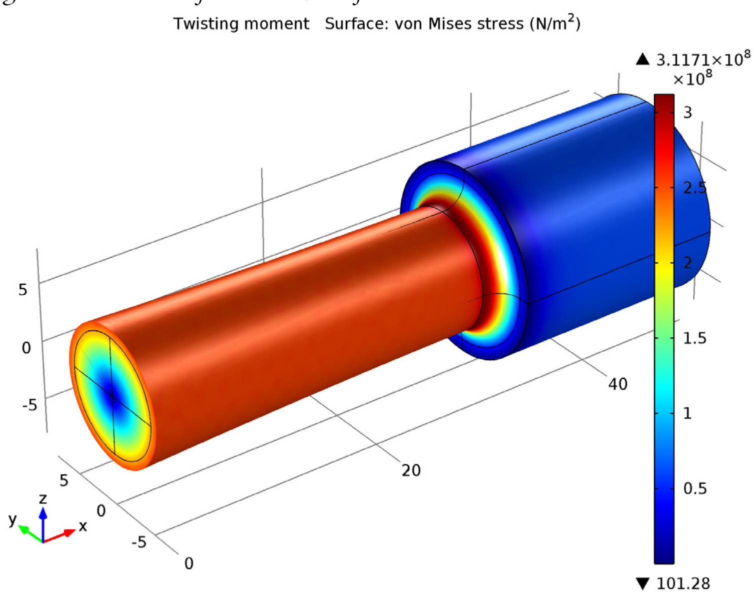

Figure 4: Effective stress from torque.

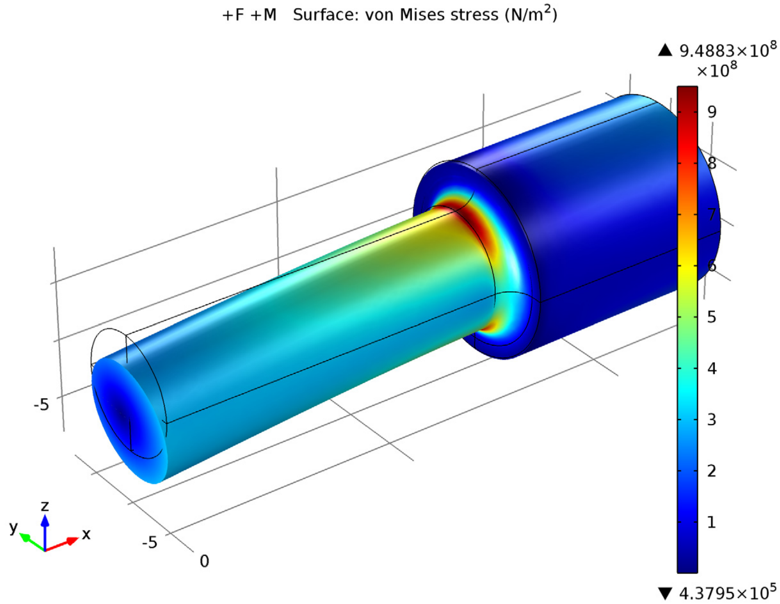

Figure 5: Effective stress distribution for one of the combined load cases

The results from the fatigue evaluation is shown in Figure 6 and Figure 7. With the Findley criterion, the fatigue usage factor is computed to 0.98, in perfect agreement with Ref. 1.

There is a large difference in the fatigue usage factor between the top and bottom side of the bar, even though the effective stress is the same at both positions. This shows how the criterion captures the difference between the predominantly tensile stress states at the critical spot, and the compressive stress states on the other side.

Using the Matake criterion the fatigue usage factor decreases to 0.73, which shows that there can be large differences between results from seemingly similar models. The critical plane computed in the Matake model differs from the one used in the Findley model. As a consequence, the maximum normal stress on the critical plane is significantly lower in the Matake case.

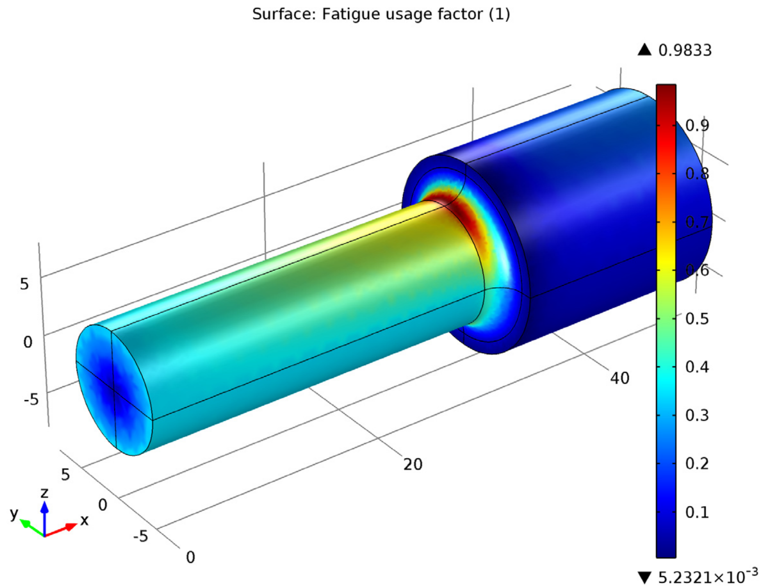

Figure 6: Fatigue usage factor using the Findley criterion.

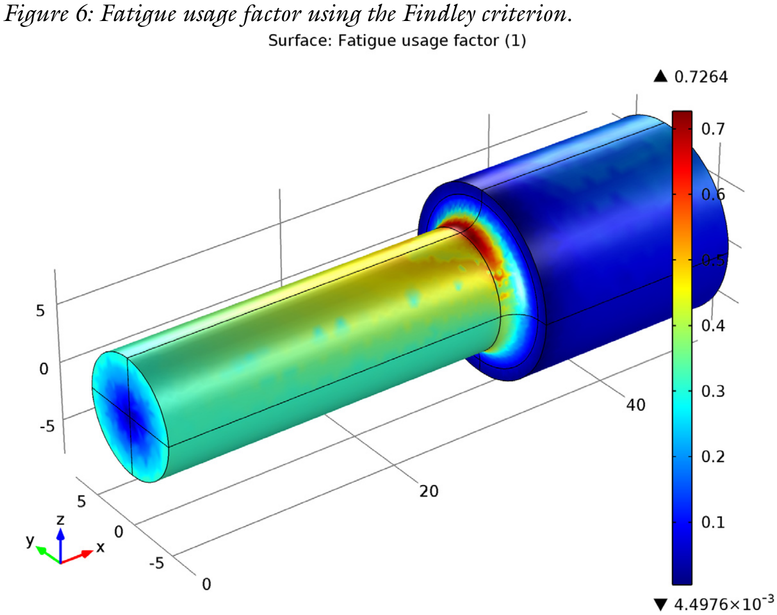

Figure 7: Fatigue usage factor using the Matake criterion.

Notes About the COMSOL Implementation

In this model, you use the load case functionality in COMSOL to produce the load cycle. In the first study the two basic load cases are analyzed. This study is not essential for the analysis, but it allows you to inspect the results of the individual basic load cases.

Reference

1. D.F. Socie and G.B. Marquis, Multiaxial Fatigue, SAE, 1999.

Model Library path: `Fatigue_Module/Stress_Based/shaft_with_fillet`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

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

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

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

4 Right-click **Model 1>Geometry 1** and choose **Work Plane**.

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **2**, set **yw** to 5.

5 Find the **Added segments** subsection. Click the **Add Linear** button.

6 Find the **Control points** subsection. In row **2**, set **xw** to 30.

7 Find the **Added segments** subsection. Click the **Add Quadratic** button.

8 Find the **Control points** subsection. In row **3**, set **xw** to 32.

9 In row **3**, set **yw** to 7.

10 In row **2**, set **xw** to 32.

11 In row **2**, set **yw** to 5.

12 Find the **Added segments** subsection. Click the **Add Linear** button.

13 Find the **Control points** subsection. In row **2**, set **yw** to 8.

14 Find the **Added segments** subsection. Click the **Add Linear** button.

15 Find the **Control points** subsection. In row **2**, set **xw** to 50.

16 Find the **Added segments** subsection. Click the **Add Linear** button.

17 Find the **Control points** subsection. In row **2**, set **yw** to 0.

18 Click the **Build Selected** button.

19 Click the **Zoom Extents** button on the Graphics toolbar.

Revolve 1

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

2 In the **Revolve** settings window, locate the **Revolution Axis** section.

3 Find the **Direction of revolution axis** subsection. In the **xw** edit field, type 1.

4 In the **yw** edit field, type 0.

5 Click the **Build Selected** button.

6 Click the **Zoom Extents** button on the Graphics toolbar.

SOLID MECHANICS

Fixed Constraint 1

1 In the **Model Builder** window, under **Model 1** right-click **Solid Mechanics** and choose **Fixed Constraint**.

2 Select Boundaries 22–25 only.

Rigid Connector 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Rigid Connector**.

2 Select Boundaries 1, 3, 5, and 7 only.

Applied Force 1

1 Right-click **Model 1>Solid Mechanics>Rigid Connector 1** and choose **Applied Force**.

2 In the **Applied Force** settings window, locate the **Applied Force** section.

3 In the **F** table, enter the following settings:

0	x
0	y
-1.94[kN]	z

Rigid Connector 1

Right-click **Model 1>Solid Mechanics>Rigid Connector 1>Applied Force 1** and choose **New Load Group**.

Applied Moment 1

1 In the **Model Builder** window, under **Model 1>Solid Mechanics** right-click **Rigid Connector 1** and choose **Applied Moment**.

2 In the **Applied Moment** settings window, locate the **Applied Moment** section.

3 In the **M** table, enter the following settings:

28.7[N*m]	X
0	Y
0	Z

4 Right-click **Model 1>Solid Mechanics>Rigid Connector 1>Applied Moment 1** and choose **New Load Group**.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, expand the **Global Definitions** node.

2 Right-click **Load Group 1** and choose **Rename**.

3 Go to the **Rename Load Group** dialog box and type `Transverse force` in the **New name** edit field.

4 Click **OK**.

5 In the **Load Group** settings window, locate the **Group Identifier** section.

6 In the **Identifier** edit field, type `lgF`.

7 In the **Model Builder** window, under **Global Definitions** right-click **Load Group 2** and choose **Rename**.

8 Go to the **Rename Load Group** dialog box and type `Twisting moment` in the **New name** edit field.

9 Click **OK**.

10 In the **Load Group** settings window, locate the **Group Identifier** section.

11 In the **Identifier** edit field, type `lgM`.

MATERIALS

Material 1

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

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	100[GPa]
Poisson's ratio	nu	0
Density	rho	0

STUDY 1

Step 1: Stationary

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

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

3 Select the **Define load cases** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Load case	lgF
Transverse force	√

6 Click **Add**.

7 In the table, enter the following settings:

Load case	lgM
Twisting moment	√

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

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

4 Click the **Build All** button.

A finer mesh is needed in the fillet to resolve the stress concentration.

5 From the **Sequence type** list, choose **User-controlled mesh**.

Size 1

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

2 In the **Size** settings window, locate the **Element Size** section.

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

Size 2

1 Right-click **Free Tetrahedral 1** and choose **Size**.

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

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

4 Select Edges 14, 15, 17, and 19 only.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `0.5`.

8 Select the **Maximum element growth rate** check box.

9 In the associated edit field, type `1.2`.

10 Click the **Build All** button.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Stress (solid)

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

2 Right-click **Stress (solid)** and choose **Plot**.

3 In the **3D Plot Group** settings window, locate the **Data** section.

4 From the **Load case** list, choose **Transverse force**.

5 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.
6 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Solid Mechanics>Stress>Stress tensor (Spatial)>Stress tensor, x component (solid.sx)**.
7 Click the **Plot** button.

SOLID MECHANICS

In the **Model Builder** window, expand the **Solid Mechanics** node.

MODEL 1

1 In the **Model Builder** window, expand the **Model 1>Solid Mechanics>Rigid Connector 1** node.
2 Right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 In the **Add physics** tree, select **Structural Mechanics>Fatigue (ftg)**.
3 Click **Finish**.

FATIGUE

Stress Based 1

1 In the **Model Builder** window, under **Model 1** right-click **Fatigue** and choose **Stress Based**.
2 In the **Stress Based** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Locate the **Solution Field** section. From the **Physics** list, choose **Solid Mechanics**.
5 Locate the **Evaluation Settings** section. Find the **Critical plane settings** subsection. In the Q edit field, type 16.

MODEL 1

In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 In the **Add physics** tree, select **Recently Used>Fatigue (ftg)**.
3 Click **Finish**.

FATIGUE 2

1 In the **Model Builder** window, under **Model 1** right-click **Fatigue 2** and choose **Stress Based**.
2 In the **Stress Based** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Locate the **Solution Field** section. From the **Physics** list, choose **Solid Mechanics**.
5 Locate the **Fatigue Model Selection** section. From the **Criterion** list, choose **Matake**.
6 Locate the **Evaluation Settings** section. Find the **Critical plane settings** subsection. In the Q edit field, type 16.

MATERIALS

Because the fatigue model is active only on the boundaries, you need to define a material on the boundaries.

Material 2

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **All boundaries**.
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Normal stress coefficient	k_Findley	0.23
Limit factor	f_Findley	440[MPa]
Normal stress coefficient	k_Matake	0.17
Limit factor	f_Matake	466[MPa]

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

3 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	Solve for
Fatigue (ftg)	×
Fatigue 2 (ftg2)	×

4 Click **Finish**.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

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

3 Select the **Define load cases** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Load case
`No load`

6 Click **Add**.

7 In the table, enter the following settings:

Load case	lgF	lgM	Weight
`+F -M`	√	√	`-1.0`

8 Click **Add**.

9 In the table, enter the following settings:

Load case	lgF	lgM
`+F +M`	√	√

10 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Stress (solid) 1

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

2 Right-click **Stress (solid) 1** and choose **Plot**.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.
3 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	**Solve for**
Solid (solid)	×

4 Click **Finish**.

STUDY 3

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.
2 In the **Stationary** settings window, click to expand the **Values of Dependent Variables** section.
3 Select the **Values of variables not solved for** check box.
4 From the **Method** list, choose **Solution**.
5 From the **Study** list, choose **Study 2, Stationary**.
6 From the **Load case** list, choose **All**.
7 In the **Model Builder** window, right-click **Study 3** and choose **Compute**.

RESULTS

If it should be possible to re-run the studies, they must have the same state as when originally created.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	**Solve for**
Fatigue {ftg}	×
Fatigue 2 {ftg2}	×

Finally, you can rename some of the features, so that the model structure is easier understood.

4 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

5 Go to the **Rename Study** dialog box and type `Study 1 (Basic load cases)` in the **New name** edit field.

6 Click **OK**.

STUDY 2

1 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

2 Go to the **Rename Study** dialog box and type `Study 2 (Combined load cases)` in the **New name** edit field.

3 Click **OK**.

STUDY 3

1 In the **Model Builder** window, right-click **Study 3** and choose **Rename**.

2 Go to the **Rename Study** dialog box and type `Study 3 (Fatigue)` in the **New name** edit field.

3 Click **OK**.