Cylindrical Test Specimen

Introduction

This model presents verifications of the stress based fatigue models in COMSOL using an example of a circular specimen subjected to normal and torsional loading.

Model Definition

A circular test specimen with a geometry according to Figure 1 is subjected to fatigue testing. Both normal load and twisting moment are applied in such a way that the stress scenario shown in Figure 2 is affecting in the central thin part of the specimen. The maximum shear stress is experienced on the outer radius of the bar.

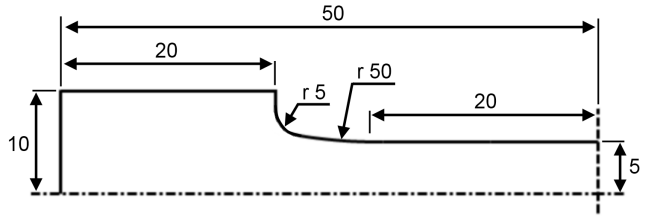

Figure 1: Geometry of the test specimen.

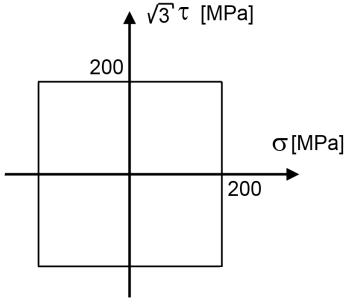

Figure 2: Stress history at the outer radius in the central part of the specimen.

This stress state can be simulated with cyclic normal force of 15708 N and cyclic twisting moment of 22.672 Nm.

The specimen is made of mild steel with Young's modulus E of 210 GPa and a Poisson's ratio ν of 0.3.

The examined fatigue criteria are Findley, Matake, and Normal stress. In all three models, the limit factor f is set to 500 MPa, and the stress sensitivity coefficient k is set to 0.2.

Results and Discussion

The load cycle is obtained in four load steps. The resulting stress at the outer radius of the thin section follows the specification, see Figure 3.

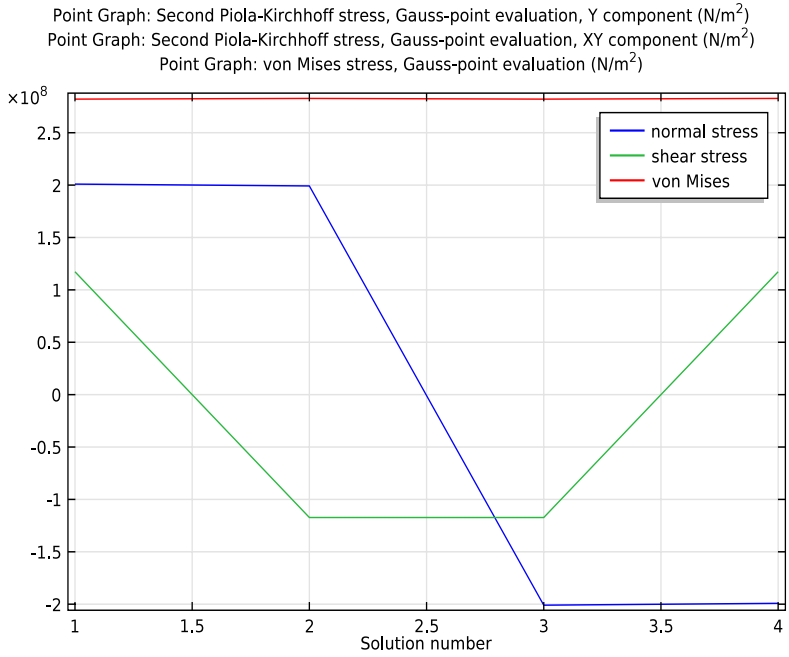

Figure 3: Resulting stresses at the outer radius of the smallest section of the specimen.

The three fatigue criteria evaluated on the boundary of the specimen are presented in Figure 4, Figure 5, and Figure 6.

In this example, it is the central part of the specimen which is in focus for the evaluation, even though the maximum fatigue usage is slightly higher in the fillets.

The Findley model shows that the highest fatigue usage factor is found at the transition between the thin cylinder and the fillet, with the value of 0.42 against 0.41 in the center. A smooth transition in the results indicates that the specified search resolution for the critical plane is sufficient to correctly capture the fatigue response.

 ©2012 COMSOL

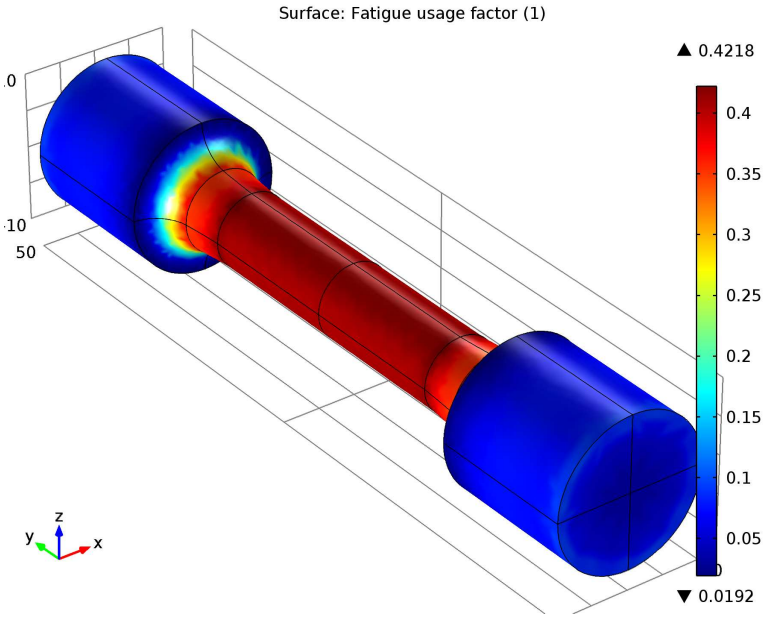

Figure 4: Fatigue usage factor Findley criterion.

The Matake criterion gives less smooth results. The reason can be found in the definition of the criterion, which selects the critical plane using the shear stresses only and then adds the normal stress. This is discussed in more detail on page 7. The analysis of the test specimen predicts a fatigue usage factor in the range from 0.34 to 0.41 in the central part, as compared to 0.42 at some points at the transition from the thin cylinder to the notch.

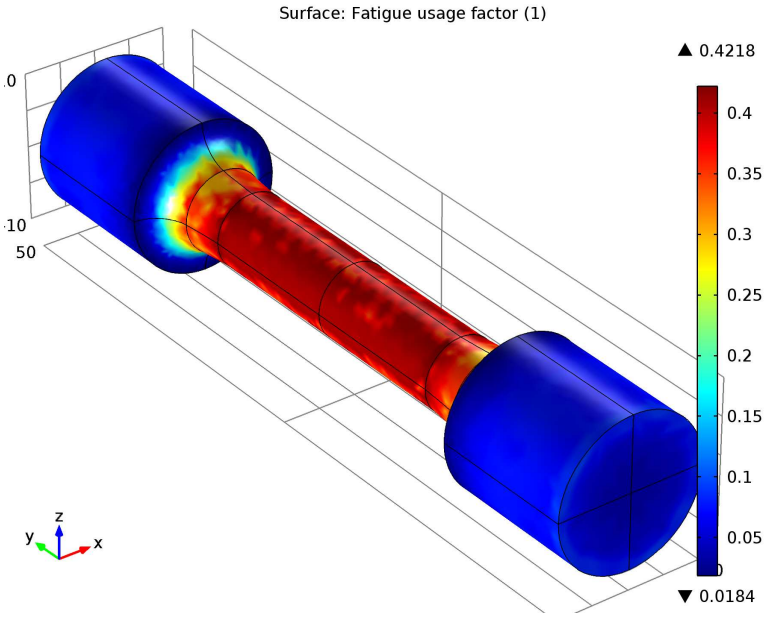

Figure 5: Fatigue usage factor Matake criterion.

The Normal Stress criterion predicts higher fatigue usage factor than the two previous models. Here again the highest fatigue usage factor is found on the transition from the thin section to the notch, 1.08, as compared to the center of the specimen, where it is 1.02.

 ©2012 COMSOL

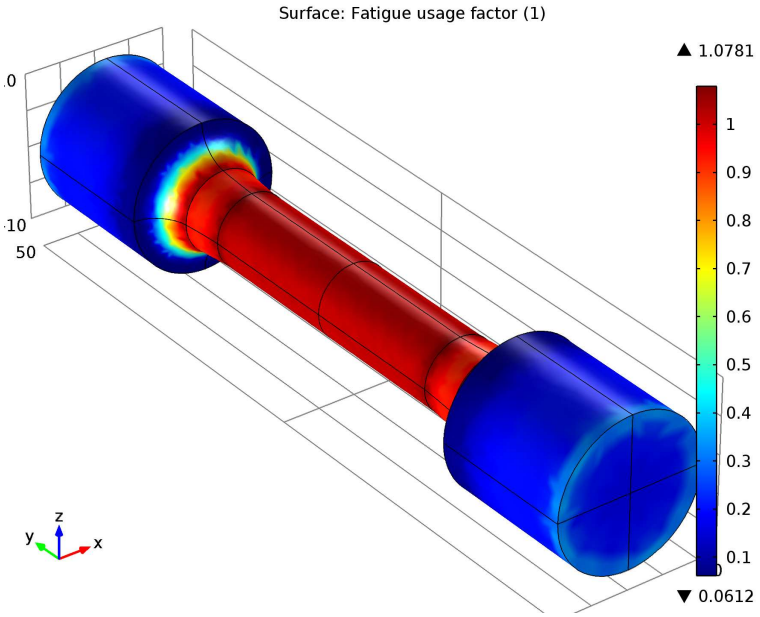

Figure 6: Fatigue usage factor Normal stress criterion.

The stress state on the surface in the central part of the specimen can be evaluated using analytical expressions. As there are no tractions on the boundary, it is in a state of plane stress. Therefore, the search for the critical plane can be reduced to two dimensions. In that case, the normal and shear stresses on a plane can be obtained as

$$\sigma(\varphi) = \sigma_a \cos\varphi^2 + \sigma_b \sin\varphi^2 + \tau_{ab} \sin\varphi\cos\varphi$$
$$\tau(\varphi) = \frac{\sigma_b - \sigma_a}{2} \sin 2\varphi + \tau_{ab} \cos 2\varphi \qquad (1)$$

where σ_a and σ_b are orthogonal normal stresses, and τ_{ab} is the shear stress. The orientation angle φ is calculated from the axis of σ_a towards σ_b. In current example, the σ_a is the axial stress, and τ_{ab} as the shear stress caused by the twisting moment.

In the current example, a search resolution of $Q = 11$ is used. This gives a 9° angular increment in the critical plane evaluation. Using Equation 1, the normal and shear

stresses evaluated on discrete planes are given in Table 1 and Table 2. The number in the subscripts indicates the load case.

TABLE 1: NORMAL STRESS ON DISCRETE PLANES

φ (°)	$\sigma_{n,1}$ **(MPa)**	$\sigma_{n,2}$ **(MPa)**	$\sigma_{n,3}$ **(MPa)**	$\sigma_{n,4}$ **(MPa)**	$\sigma_{n,max}$ **(MPa)**	$\Delta\sigma_n$ **(MPa)**
0	200	200	-200	-200	200	400
9	231	159	-231	-159	231	462
18	249	113	-249	-133	249	498
27	252	65	-252	-65	252	504
36	241	21	-241	-21	241	481
45	215	-15	-215	15	215	431
54	179	-41	-17	41	179	358
63	135	-52	-135	52	135	269
72	87	-49	-87	49	87	174
81	41	-31	-41	31	41	81
90	0	0	0	0	0	0
99	-31	41	31	-41	41	81
108	-49	87	49	-87	87	174
117	-52	135	52	-135	135	269
126	-41	179	41	-179	179	358
135	-15	215	15	-215	215	431
144	21	241	-21	-241	230	481
153	65	252	-65	-252	252	504
162	113	249	-113	-249	249	498
171	159	231	-159	231	231	462

TABLE 2: SHEAR STRESS ON DISCRETE PLANES

φ (°)	τ_1 **(MPa)**	τ_2 **(MPa)**	τ_3 **(MPa)**	τ_4 **(MPa)**	$\Delta\tau$ **(MPA)**	$\Delta\tau/2+0.2\sigma_{n,max}$ **(MPa)**
0	115	-115	-115	115	230	155
9	79	-141	-79	141	281	187
18	35	-152	-35	152	304	202
27	-13	-149	13	149	298	199
36	-59	-131	59	131	262	179
45	-100	-100	100	100	200	143

 ©2012 COMSOL

TABLE 2: SHEAR STRESS ON DISCRETE PLANES

φ (°)	τ_1 (MPa)	τ_2 (MPa)	τ_3 (MPa)	τ_4 (MPa)	$\Delta\tau$ (MPA)	$\Delta\tau/2+0.2\sigma_{n,max}$ (MPa)
54	-131	-59	131	59	262	167
63	-149	-13	149	13	298	176
72	-152	35	152	-35	304	170
81	-141	79	141	-79	281	149
90	-115	115	115	-115	231	115
99	-79	141	79	-141	281	149
108	-35	152	35	-152	304	170
117	13	149	-13	-149	298	176
126	59	131	-59	-131	262	167
135	100	100	-100	-100	200	143
144	131	59	-131	-59	262	179
153	149	13	-149	-13	298	199
162	152	-35	-152	35	304	202
171	141	-79	-141	79	281	187

The Findley criterion searches for a critical plane where a combination between the shear stress range and the normal stress is highest. This range is shown in the last column of Table 2. It can be seen that 401 MPa is found at planes oriented at 18° and 162°. This results into a usage factor of 202/500 = 0.40, which is in good agreement with the computed results.

The Matake criterion selects the critical plane as the one having with the largest shear stress range. This occurs at orientations 18°, 72°, 108°, and 162° where the range is 304 MPa. The maximum normal stress on those planes is either 87 MPa or 249 MPa, see Table 1. Due to small numerical differences, either of these four angles may be selected as being the critical one. Since the Matake criterion does not contain the normal stress in the selection of the critical plane, it may in one point choose a plane having the lower normal stress, and in another point a plane with the higher normal stress. In Figure 5, this feature is demonstrated by the non smooth results. The analytical values for the Matake usage factor are 202/500 = 0.40 and 170/500 = 0.34, which is in good agreement with the computed results.

The Normal stress criterion considers the plane with the largest normal stress range. The last column of Table 1 shows that this is found on planes with orientations 27°

and 153° where the range is 504 MPa. This results into a fatigue usage factor of 1.01, which is in good agreement with the computed results.

Notes About the COMSOL Implementation

In the critical plane evaluation, a search resolution Q indicates the number of evaluation point along 90° of a unit circle. The angle between two evaluation points is then at most (90°)/(Q – 1).

Model Library path: `Fatigue_Module/Verification_Models/cylindrical_test_specimen`

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**.

GLOBAL DEFINITIONS

Parameters

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

2 In the **Parameters** settings window, locate the **Parameters** section.

 ©2012 COMSOL

3 In the table, enter the following settings:

Name	Expression
Moment	22.672 [N*m]
Force	15708 [N]
f	500 [MPa]
k	0.2

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Insert Sequence from File**. Browse to the model's Model Library folder and double-click the file `cylindrical_test_specimen.mph`. You can then continue to the Materials section.

GEOMETRY 1

In the **Model Builder** window, under **Model 1** right-click **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 **1**, set **yw** to -50.

5 In row **2**, set **yw** to 50.

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

7 Find the **Control points** subsection. In row **2**, set **xw** to 10.

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

9 Find the **Control points** subsection. In row **2**, set **yw** to 30.

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

11 Find the **Control points** subsection. In row **2**, set **xw** to 6.01.

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

13 Find the **Control points** subsection. In row **2**, set **xw** to 5.25 and **yw** to 25.

14 In row **3**, set **yw** to 20 and **xw** to 5.

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

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

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

18 Find the **Control points** subsection. In row **2**, set **yw** to `-20`.

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

20 Find the **Control points** subsection. In row **2**, set **xw** to `5.25` and **yw** to `-25`.

21 In row **3**, set **yw** to `-30` and **xw** to `6.01`.

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

23 Find the **Control points** subsection. In row **2**, set **xw** to `10`.

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

25 Find the **Control points** subsection. In row **2**, set **yw** to `-50`.

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

27 Find the **Control points** subsection. In row **2**, set **xw** to `0`.

28 Click the **Build Selected** button.

Fillet 1

1 Right-click **Plane Geometry** and choose **Fillet**.

2 On the object **b1**, select Points 6 and 7 only.

3 In the **Fillet** settings window, locate the **Radius** section.

4 In the **Radius** edit field, type `5`.

Revolve 1

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

2 In the **Settings** window, click **Build All**.

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	210e9
Poisson's ratio	nu	0.3
Density	rho	7800

 ©2012 COMSOL

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 3, 4, 21, and 23 only.

Rigid Connector 1

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

2 Select Boundaries 11, 12, 40, and 41 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
Force	y
0	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:

0	X
Moment	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, then click **Load Group 1**.

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

3 In the **Identifier** edit field, type `lgf`.

4 In the **Model Builder** window, under **Global Definitions** click **Load Group 2**.

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

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

Create a load cycle consisting of four load cases.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then 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** four times.

5 In the table, enter the following settings:

lgf	Weight	lgm	Weight
√	(As is)	√	(As is)
√	(As is)	√	-1.0
√	-1.0	√	-1.0
√	-1.0	√	(As is)

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

RESULTS

Verify stresses at the center of the test specimen.

1D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 2** and choose **Point Graph**.

3 Select Point 20 only.

4 In the **Point Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Solid Mechanics>Stress**

 ©2012 COMSOL

(Gauss points)>Second Piola-Kirchhoff stress, Gauss-point evaluation (Material)>Second Piola-Kirchhoff stress, Gauss-point evaluation, Y component (solid.SGpY).

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

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

7 In the table, enter the following settings:

Legends
normal stress

8 Right-click **Results>1D Plot Group 2>Point Graph 1** and choose **Duplicate**.

9 In the **Point Graph** settings window, locate the **y-Axis Data** section.

10 In the **Expression** edit field, type `solid.SGpXY`.

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

Legends
shear stress

12 Right-click **Results>1D Plot Group 2>Point Graph 2** and choose **Duplicate**.

13 In the **Point Graph** settings window, locate the **y-Axis Data** section.

14 In the **Expression** edit field, type `solid.misesGp`.

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

Legends
von Mises

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 **Structural Mechanics>Fatigue (ftg)**.

3 Click **Next**.

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

Physics	Solve for
Solid Mechanics (solid)	×

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

6 Click **Finish**.

FATIGUE

1 In the **Model Builder** window, under **Model 1** right-click **Fatigue** and choose **Rename**.

2 Go to the **Rename Fatigue** dialog box and type `Fatigue Findley` in the **New name** edit field.

3 Click **OK**.

4 Right-click **Model 1>Fatigue** and choose **Stress Based**.

FATIGUE FINDLEY

Stress Based 1

1 In the **Stress Based** settings window, locate the **Boundary Selection** section.

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

3 Locate the **Solution Field** section. From the **Physics** list, choose **Solid Mechanics**.

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 **Structural Mechanics>Fatigue (ftg)**.

3 Click **Finish**.

FATIGUE 2

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

2 Go to the **Rename Fatigue** dialog box and type `Fatigue Matake` in the **New name** edit field.

3 Click **OK**.

4 Right-click **Model 1>Fatigue 2** and choose **Stress Based**.

FATIGUE MATAKE

Stress Based 1

1 In the **Stress Based** settings window, locate the **Boundary Selection** section.

 ©2012 COMSOL

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

3 Locate the **Solution Field** section. From the **Physics** list, choose **Solid Mechanics**.

4 Locate the **Fatigue Model Selection** section. From the **Criterion** list, choose **Matake**.

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 **Structural Mechanics>Fatigue (ftg)**.

3 Click **Finish**.

FATIGUE 3

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

2 Go to the **Rename Fatigue** dialog box and type `Fatigue Normal Stress` in the **New name** edit field.

3 Click **OK**.

4 Right-click **Model 1>Fatigue 3** and choose **Stress Based**.

FATIGUE NORMAL STRESS

Stress Based 1

1 In the **Stress Based** settings window, locate the **Boundary Selection** section.

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

3 Locate the **Solution Field** section. From the **Physics** list, choose **Solid Mechanics**.

4 Locate the **Fatigue Model Selection** section. From the **Criterion** list, choose **Normal stress**.

MATERIALS

In the **Model Builder** window, expand the **Model 1>Materials** node.

Material 2

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 sensitivity coefficient	k_Findley	k
Limit factor	f_Findley	f
Normal stress sensitivity coefficient	k_Matake	k
Limit factor	f_Matake	f
Limit factor	f_NormalStress	f

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then 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 Matake (ftg2)	×
Fatigue Normal Stress (ftg3)	×

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 **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 1, Stationary**.

6 From the **Load case** list, choose **All**.

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

RESULTS

Fatigue Usage Factor (ftg)

1 In the **Model Builder** window, under **Results** right-click **Fatigue Usage Factor (ftg)** and choose **Rename**.

 ©2012 COMSOL

2 Go to the **Rename 3D Plot Group** dialog box and type `Fatigue Usage Factor (Findley)` in the **New name** edit field.

3 Click **OK**.

Fatigue Usage Factor (ftg2)

1 In the **Model Builder** window, under **Results** right-click **Fatigue Usage Factor (ftg2)** and choose **Rename**.

2 Go to the **Rename 3D Plot Group** dialog box and type `Fatigue Usage Factor (Matake)` in the **New name** edit field.

3 Click **OK**.

Fatigue Usage Factor (ftg3)

1 In the **Model Builder** window, under **Results** right-click **Fatigue Usage Factor (ftg3)** and choose **Rename**.

2 Go to the **Rename 3D Plot Group** dialog box and type `Fatigue Usage Factor (Normal Stress)` in the **New name** edit field.

3 Click **OK**.

 ©2012 COMSOL