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 cylindrical 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.

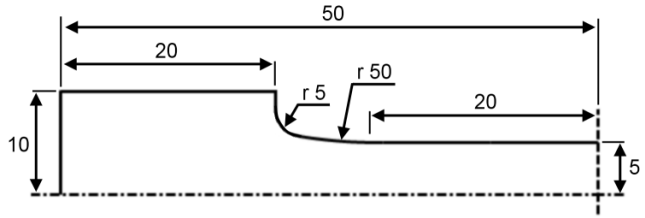

Figure 1: Geometry of the test specimen.

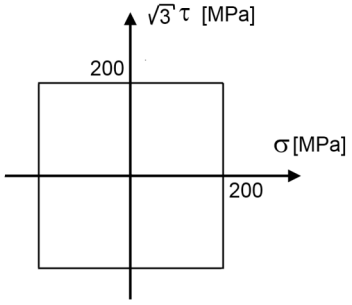

Figure 2: Stress history at the outer radius in the central part of the specimen.

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. Material parameters are calculated from uniaxial fatigue tests. Results of two tests are available. In the first one material is tested in reversed tension-compression, which means that the load amplitude is alternating around zero stress, and the endurance limit is $\sigma_{R=-1} = 350$ MPa. In the second one material is tested in pure tension, where load is pulsating between zero to two times the load amplitude, and the endurance limit is $\sigma_{R=0} = 288$ MPa. The denominator of the endurance limit shows the R -value of the test.

Findley parameters are related to the above fatigue data via

$$f_{\mathrm{F}} = \frac{\sigma_{R=-1}}{2}(k_{\mathrm{F}} + \sqrt{1 + k_{\mathrm{F}}^2})$$
$$f_{\mathrm{F}} = \frac{\sigma_{R=0}}{2}(2k_{\mathrm{F}} + \sqrt{1 + 4k_{\mathrm{F}}^2}) \quad (1)$$

and are $f_{\mathrm{F}} = 213$ MPa and $k_{\mathrm{F}} = 0.20$. Matake parameters are related to the above fatigue data via

$$f_{\mathrm{M}} = \frac{\sigma_{R=-1}}{2}(1 + k_{\mathrm{M}})$$
$$f_{\mathrm{M}} = \frac{\sigma_{R=0}}{2}(1 + 2k_{\mathrm{M}}) \quad (2)$$

and are $f_{\mathrm{M}} = 223$ MPa and $k_{\mathrm{F}} = 0.27$. In the normal stress model the stress limit equals twice the stress amplitude of the fatigue test. The model does not take into account the R -value dependence and therefor two different limits are obtained. In order to give conservative results the lower limit, $\sigma_{R=0}$,is chosen and thus the model parameter is $f_{\mathrm{N}} = 576$ MPa.

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.

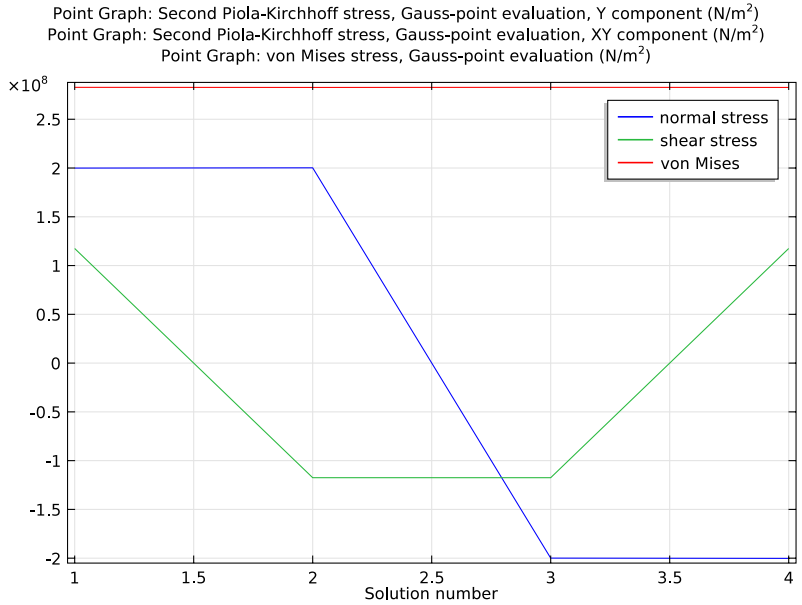

Figure 3: Resulting stresses at the outer radius of the smallest section of the specimen.

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 0.99 against 0.95 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.

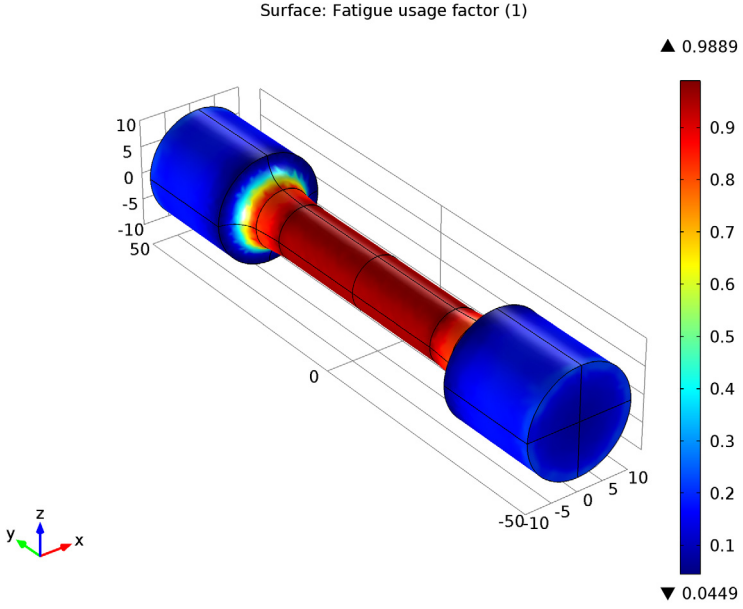

Figure 4: Fatigue usage factor Findley criterion.

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 9. The analysis of the test specimen predicts a fatigue usage factor in the range from 0.78 to 0.98 in

the central part, as compared to 1.03 at some points at the transition from the thin cylinder to the notch.

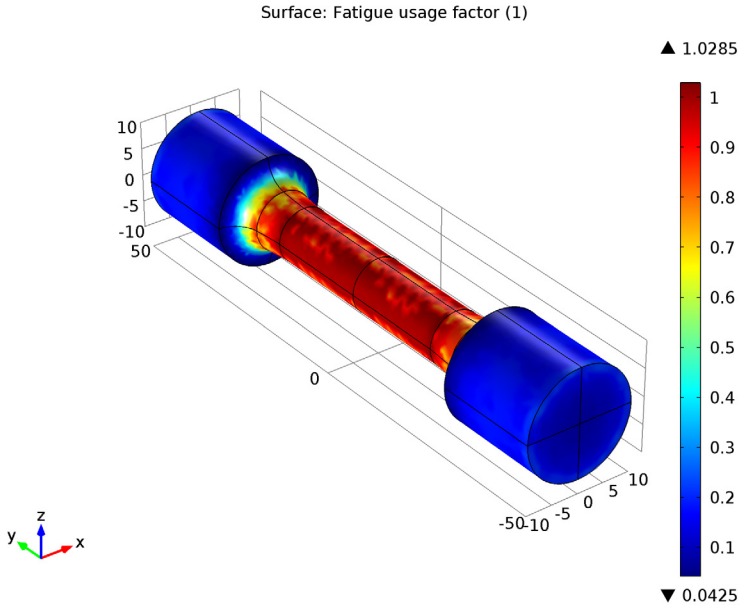

Figure 5: Fatigue usage factor Matake criterion.

The Normal Stress criterion predicts yet different 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, 0.93, as compared to the center of the specimen, where it is 0.88.

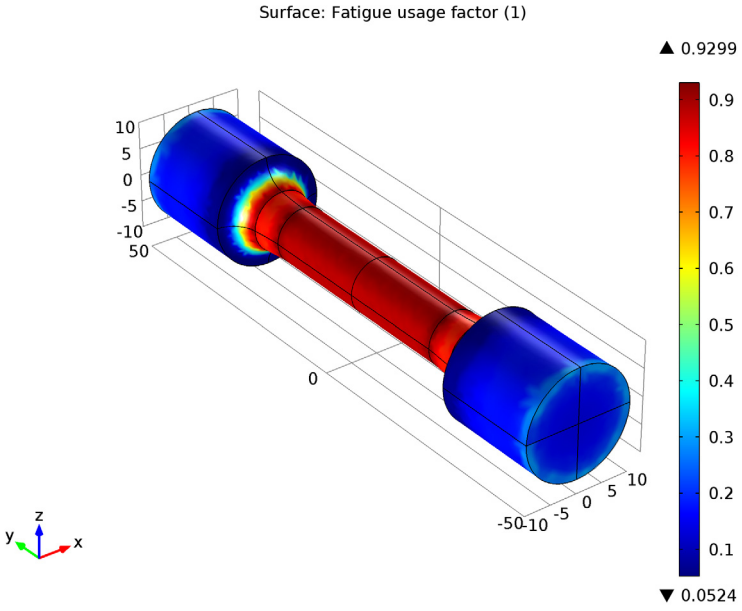

Figure 6: Fatigue usage factor Normal stress criterion.

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

$$\sigma(\varphi) = \sigma_a \cos\varphi^2 + \sigma_b \sin\varphi^2 + \tau_{ab} \sin\varphi\cos\varphi$$
$$\tau(\varphi) = \frac{\sigma_b - \sigma_a}{2} \sin 2\varphi + \tau_{ab} \cos 2\varphi \qquad (3)$$

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 the 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 3, 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)
0	115	-115	-115	115	230
9	79	-141	-79	141	281
18	35	-152	-35	152	304
27	-13	-149	13	149	298
36	-59	-131	59	131	262
45	-100	-100	100	100	200
54	-131	-59	131	59	262
63	-149	-13	149	13	298
72	-152	35	152	-35	304
81	-141	79	141	-79	281
90	-115	115	115	-115	231
99	-79	141	79	-141	281
108	-35	152	35	-152	304
117	13	149	-13	-149	298
126	59	131	-59	-131	262
135	100	100	-100	-100	200
144	131	59	-131	-59	262
153	149	13	-149	-13	298
162	152	-35	-152	35	304
171	141	-79	-141	79	281

The Findley criterion searches for a critical plane where a combination between the shear stress range and the normal stress is highest. This stress is shown in the second column of Table 3, where it can be seen that 202 MPa is found at planes oriented at 18° and 162°. This results into a usage factor of $202/213 = 0.95$, which is in good agreement with the computed results.

TABLE 3: FATIGUE STRESS

φ (°)	$\Delta\tau/2+0.20\sigma_n$ (MPa)	$\Delta\tau/2+0.27\sigma_n$ (MPa)	$\Delta\sigma_n$(MPa)
0	155	169	400
9	187	203	462
18	202	219	498
27	199	217	504
36	179	196	481
45	143	158	431
54	167	179	358
63	176	185	269
72	170	175	174
81	149	152	81
90	115	115	0
99	149	152	81
108	170	175	174
117	176	185	269
126	167	179	358
135	143	158	431
144	179	196	481
153	199	217	504
162	202	219	498
171	187	203	462

The Matake criterion selects the critical plane as the one with the largest shear stress range. This occurs at orientations 18°, 72°, 108°, and 162° where the range is 304 MPa, see Table 2. The maximum normal stress on those planes is either 87 MPa or 249 MPa, see Table 1 and the Matake stress is either 219 MPa or 175 MPa, see Table 3. Since the Matake criterion does not contain the normal stress in the selection of the critical plane, the fatigue criteria will be calculated with either one of them. In Figure 5, this feature is demonstrated by the non smooth results. The analytical values at the critical planes for the Matake usage factor are 219/223 = 0.98 and 175/223 = 0.78, 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 3 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 504/576 = 0.88, 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

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

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

1 In the **Model Wizard** window, click the **3D** button.

2 In the **Select physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click the **Add** button.

4 Click the **Study** button.

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

6 Click the **Done** button.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 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 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression
Moment	22.672 [N*m]
Force	15708 [N]

GEOMETRY 1

On the **Geometry** toolbar, click **Work Plane**.

Bézier Polygon 1

1 In the **Model Builder** window, under **Component 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 **Weights** subsection. In row **2**, set **xw** to 5.25.

14 In row **2**, set **yw** to 25.

15 In row **3**, set **yw** to 20.

16 In row **3**, set **xw** to 5.

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

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

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

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

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

22 Find the **Weights** subsection. In row **2**, set **xw** to 5.25.

23 In row **2**, set **yw** to -25.

24 In row **3**, set **yw** to -30.

25 In row **3**, set **xw** to 6.01.

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

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

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

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

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

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

32 Click the **Build Selected** button.

Fillet 1

1 On the **Work Plane** toolbar, click **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 On the **Geometry** toolbar, click **Revolve**.

2 In the **Model Builder** window, under **Component 1>Geometry 1** right-click **Revolve Settings** window choose **Build All Objects**.

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

MATERIALS

Material 1

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	210e9	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7800	kg/m³	Basic

SOLID MECHANICS

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 3, 4, 21, and 23 only.

Rigid Connector 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundaries 11, 12, 40, and 41 only.

Applied Force 1

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

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

3 Specify the $\mathbf{F}$ vector as

0	x
Force	y
0	z

Rigid Connector 1

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

Applied Moment 1

1 In the **Model Builder** window, under **Component 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 Specify the $\mathbf{M}$ vector as

0	x
Moment	y
0	z

4 Right-click **Component 1>Solid Mechanics>Rigid Connector 1>Applied Moment 1** and choose **Load Group>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 Locate the **Study Extensions** section. Select the **Define load cases** check box.

4 Click **Add** four times.

5 In the table, enter the following settings:

Define load cases

Load case	lgf	Weight	lgm	Weight
Load case 1	✓	1.0	✓	1.0
Load case 2	✓	1.0	✓	-1.0
Load case 3	✓	-1.0	✓	-1.0
Load case 4	✓	-1.0	✓	1.0

6 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

Verify stresses at the center of the test specimen.

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 On the **1D Plot Group** toolbar, click **Point Graph**.

3 Select Point 20 only.

4 In the **Point Graph** settings window, locate the **y-axis data** section. In the upper-right corner of the section click **Replace Expression** and select **Solid Mechanics>Stress (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

16 On the top of the section click **Plot**.

COMPONENT 1

On the **Home** toolbar, click **Add Physics**.

ADD PHYSICS

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

2 In the **Add Physics** tree, select **Structural Mechanics>Fatigue (ftg)**.

3 Find the **Physics in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	×

4 In the **Add Physics** window, click **Add to Component**.

ROOT

On the **Home** toolbar, click **Add Study**.

ADD STUDY

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

2 Find the **Physics in study** subsection. In the table, enter the following settings:

Physics	Solve
Solid Mechanics (solid)	×

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

4 In the **Add Study** window, click **Add Study**.

FATIGUE

Stress-Based 1

1 In the **Model Builder** window, under **Component 1** right-click **Fatigue** and choose **Stress-Based** boundary evaluation.

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

6 In the **Rename Fatigue** dialog box, type `Fatigue Findley` in the **New name** edit field.

Click **OK**.

ADD PHYSICS

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

2 In the **Add Physics** tree, select **Structural Mechanics>Fatigue (ftg)**.

3 In the **Add Physics** window, click **Add to Component**.

FATIGUE 2

1 In the **Model Builder** window, under **Component 1** right-click **Fatigue 2** and choose **Stress-Based** boundary evaluation.

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

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

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

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

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

7 In the **Rename Fatigue** dialog box, type `Fatigue Matake` in the **New name** edit field.

8 Click **OK**.

ADD PHYSICS

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

2 In the **Add Physics** tree, select **Structural Mechanics>Fatigue (ftg)**.

3 In the **Add Physics** window, click **Add to Component**.

FATIGUE 3

1 In the **Model Builder** window, under **Component 1** right-click **Fatigue 3** and choose **Stress-Based** boundary evaluation.

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

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

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

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

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

7 In the **Rename Fatigue** dialog box, type `Fatigue Normal Stress` in the **New name** edit field.

8 Click **OK**.

MATERIALS

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

Material 2

1 Right-click **Materials** and choose **New 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	Unit	Property group
Normal stress sensitivity coefficient	k_Findley	0.20	1	Findley
Limit factor	f_Findley	213 [MPa]	Pa	Findley
Normal stress sensitivity coefficient	k_Matake	0.27	1	Matake
Limit factor	f_Matake	223 [MPa]	Pa	Matake
Limit factor	f_NormalStress	576 [MPa]	Pa	Normal stress

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	Discretization
Fatigue Matake	×	physics
Fatigue Normal Stress	×	physics

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 Locate the **Values of Dependent Variables** section. 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 On the **Home** toolbar, click **Compute**.

RESULTS

Fatigue Usage Factor (ftg)

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

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**.