Low-Cycle Fatigue Analysis of Cylinder with a Hole

Introduction

A load carrying component of a structure is subjected to multi-axial cyclic loading during which localized yielding of the material occurs. In this model you perform a low cycle fatigue analysis of the part based on the Smith-Watson-Topper (SWT) model. Due to localized yielding, you can use two methods to obtain the stress and strain distributions for the fatigue evaluation. The first method is a full elastoplastic analysis with kinematic hardening, while the second is a linear elastic analysis with Neuber correction for plasticity. This example explores the first method. In the model Notch Approximation to Low-Cycle Fatigue Analysis of Cylinder with a Hole, the same problem is solved using the elastic approach.

Model Definition

GEOMETRY

A cylinder contains a hole, drilled perpendicularly to its axis. The outer and inner diameters of the cylinder are 200 and 180 mm respectively. Its height is 100 mm. The diameter of the hole is 20 mm. The cylinder is loaded by an axial force which varies in time.

As the structure and loading contains several symmetries, you may model only 1/8 of the cylinder, a shown in Figure 1.

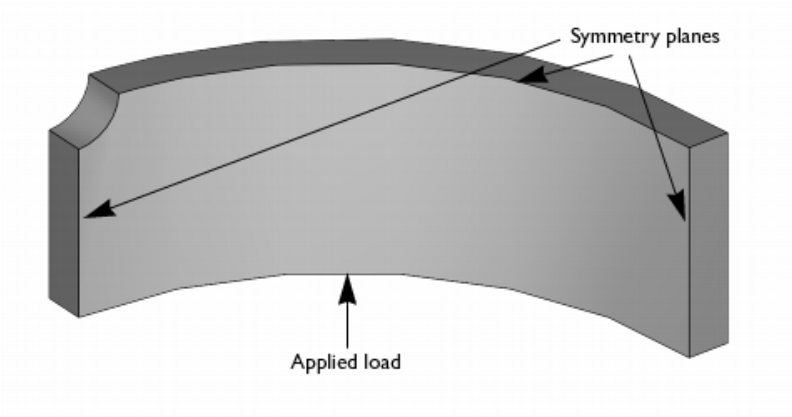

Figure 1: Model geometry with constrained and loaded faces.

MATERIAL PROPERTIES

- Elastic data: Isotropic with E = 210 GPa, ν = 0.3
- Kinematic hardening plasticity data: Yield stress 380 MPa, Tangent modulus 75 GPa
- Fatigue parameters for the SWT equation:
  - σ_f' = 1323 MPa
  - b = •0.097
  - ε_f' = 0.375
  - c = •0.60

CONSTRAINTS

Apply symmetry conditions on the three symmetry sections shown in Figure 1.

LOAD

The loaded boundary of the cylinder is subjected to a pressure of 200 MPa having a sinusoidal variation with time. Since the problem is quasi-static, time is not used explicitly in the problem. Instead you model the load as a function of a parameter, and use the parametric solver to trace the history.

Results and Discussion

The von Mises stress when the maximum load is reached for the first time (at a parametric value of 0.25) is shown in Figure 2. Notice that the yield limit 380 MPa is exceeded by a large factor, and you can therefore expect significant plastic strains.

After completion of the first load cycle, there are no external loads. The internal residual stresses are still high, as can be seen in Figure 3.

For a point located on the face of the hole and in the XY symmetry plane, the highest stresses appear in the Z direction. In Figure 4, you can follow the development of the normal stress and strain in this direction during the first load cycle. The accumulation of plastic strain is shown in Figure 5. This repetitive plastic deformation can be viewed as driving the generation of a fatigue crack.

After having run a second load cycle, the stress-strain loop repeats itself, and you can consider the process to have reached a steady state condition. This is shown in Figure 6. The continuation of the effective strain history is shown in Figure 7.

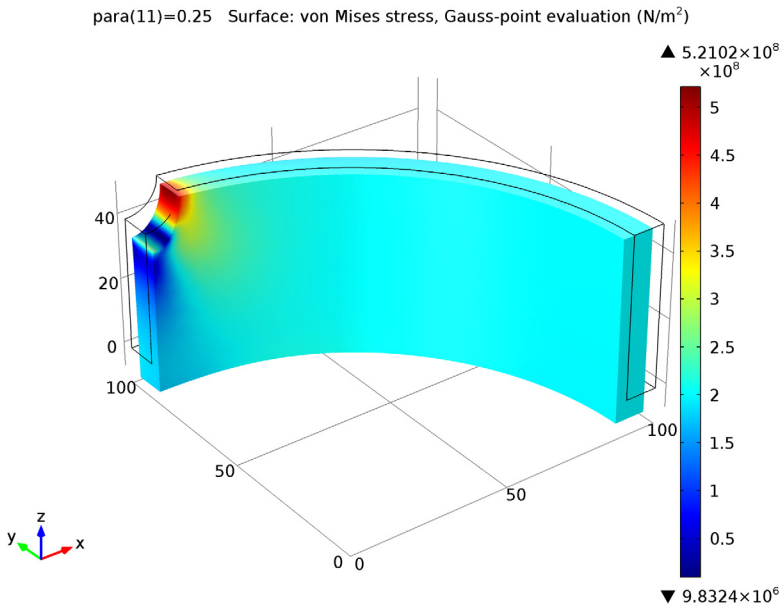

Figure 2: von Mises stress level at the first maximum load

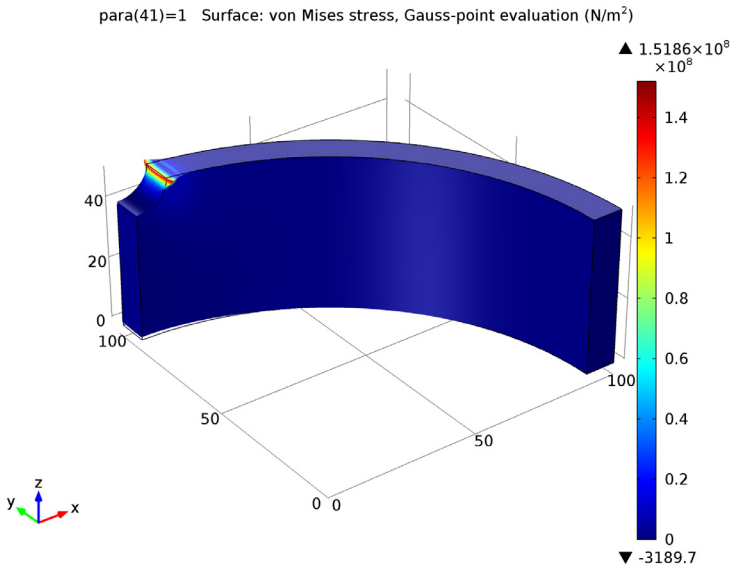

Figure 3: Residual effective stress after completion of first load cycle

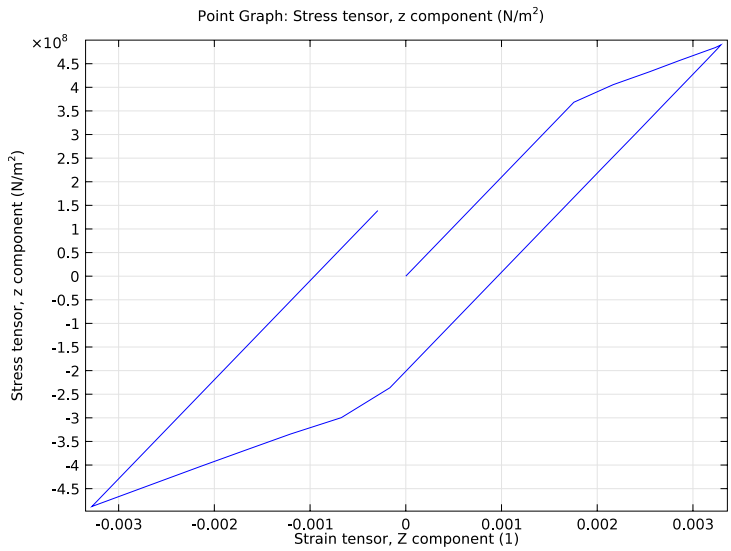

Figure 4: Stress as function of strain during the first load cycle.

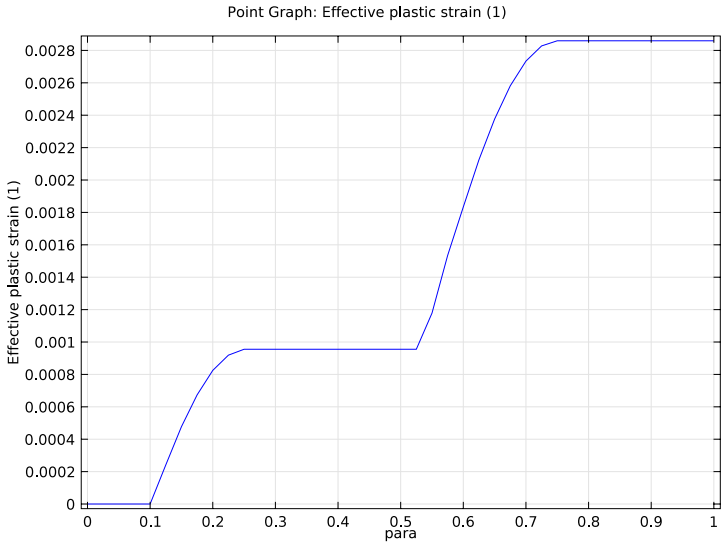

Figure 5: Accumulated effective plastic strain during the first load cycle

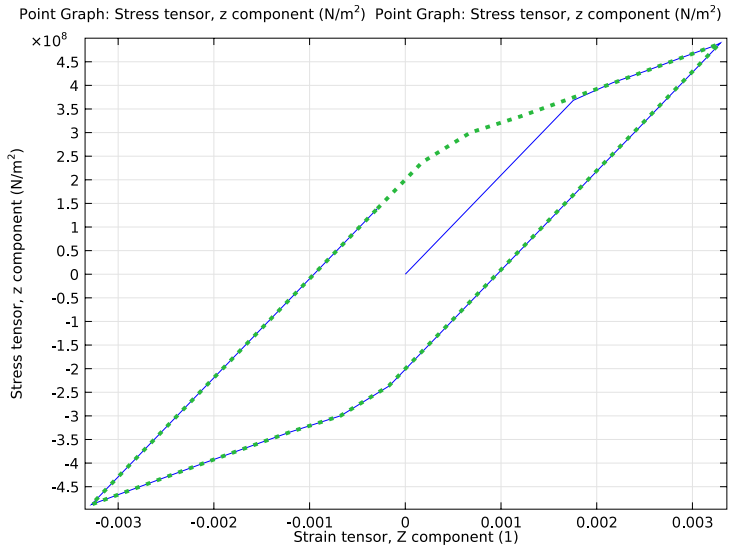

Figure 6: Stress as function of strain after two load cycles

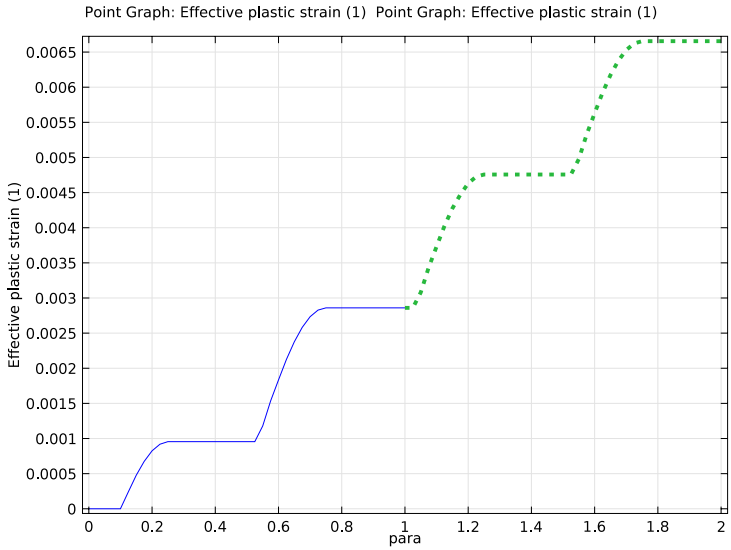

Figure 7: Accumulated effective plastic strain during two load cycles

The result of the fatigue evaluation is shown in Figure 8 below. The most critical point has a computed life of approximately 13000 load cycles.

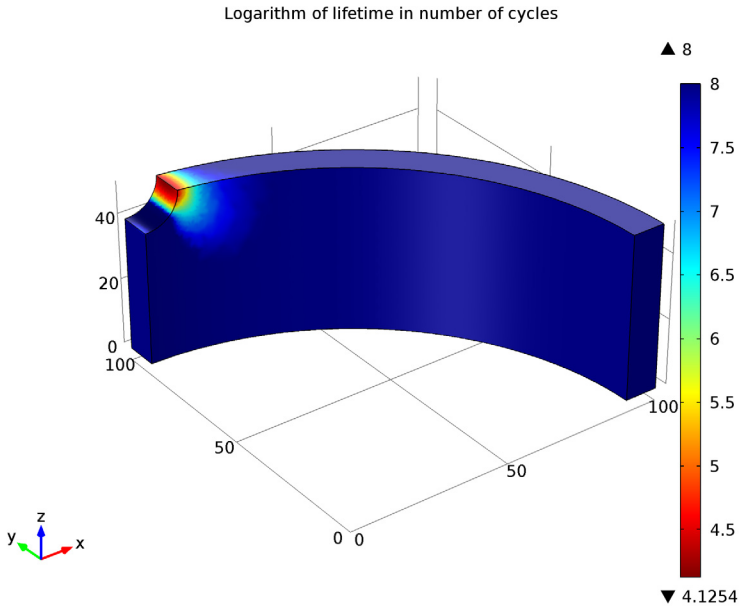

Figure 8: Lifetime plot

Notes About the COMSOL Implementation

You need to use two studies for the elastoplastic analysis. The second study contains the strain cycle that is passed on to the fatigue evaluation. It takes the results from the first study as initial conditions.

Model Library path: `Fatigue_Module/Strain_Based/cylinder_with_hole_plastic`

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**.

Cylinder 1

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

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

3 In the **Radius** edit field, type 100.

4 In the **Height** edit field, type 100.

Cylinder 2

1 Right-click **Component 1>Geometry 1>Cylinder 1** and choose **Duplicate**.

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

3 In the **Radius** edit field, type 90.

Difference 1

1 On the **Geometry** toolbar, click **Difference**.

2 Select the object **cyl1** only.

3 In the **Difference** settings window, locate the **Difference** section.

4 Select the **Objects to subtract** toggle button.

5 Select the object **cyl2** only.

6 Click the **Build All Objects** button.

Cylinder 3

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

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

3 In the **Radius** edit field, type 10.

4 In the **Height** edit field, type 220.

5 Locate the **Position** section. In the **y** edit field, type -110.

6 In the **z** edit field, type 50.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

8 Click the **Build All Objects** button.

Difference 2

1 On the **Geometry** toolbar, click **Difference**.

2 Select the object **dif1** only.

3 In the **Difference** settings window, locate the **Difference** section.

4 Select the **Objects to subtract** toggle button.

5 Select the object **cyl3** only.

6 Click the **Build All Objects** button.

Block 1

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** edit field, type 100.

4 In the **Depth** edit field, type 100.

5 In the **Height** edit field, type 50.

6 Click the **Build All Objects** button.

Intersection 1

1 On the **Geometry** toolbar, click **Intersection**.

2 Select the objects **dif2** and **blk1** only.

3 Click the **Build All Objects** button.

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

Use an extra domain for controlling the mesh. The mesh should be fine in a region close to the hole.

Cylinder 4

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

2 In the **Cylinder** settings window, locate the **Axis** section.

3 From the **Axis type** list, choose **y-axis**.

4 Locate the **Size and Shape** section. In the **Height** edit field, type 40.

5 In the **Radius** edit field, type `15`.

6 Locate the **Position** section. In the **y** edit field, type `80`.

7 In the **z** edit field, type `50`.

8 Click the **Build Selected** button.

Intersection 2

1 On the **Geometry** toolbar, click **Intersection**.

2 Select the objects **cyl4** and **int1** only.

3 In the **Intersection** settings window, locate the **Intersection** section.

4 Select the **Keep input objects** check box.

5 Click the **Build All Objects** button.

6 In the **Relative repair tolerance** edit field, type `1e-6`.

Delete Entities 1

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

2 In the **Delete Entities** settings window, locate the **Entities or Objects to Delete** section.

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

4 Select the object **cyl4** only.

5 Click the **Build All Objects** button.

Mesh Control Domains 1

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domain 2 only.

3 On the **Home** toolbar, click **Build All**.

SOLID MECHANICS

Symmetry 1

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

2 Select Boundaries 1, 6, and 7 only.

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
para	0

SOLID MECHANICS

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 3 only.

3 In the **Boundary Load** settings window, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
-200[MPa]*sin(2*pi*para)	z

Plasticity 1

1 In the **Model Builder** window, under **Component 1>Solid Mechanics** right-click **Linear Elastic Material 1** and choose **Plasticity**.

2 In the **Plasticity** settings window, locate the **Plasticity Model** section.

3 From the **Hardening model** list, choose **Kinematic**.

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	210[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	0	kg/m³	Basic

Property	Name	Value	Unit	Property group
Initial yield stress	sigmags	380[MPa]	Pa	Elastoplastic material model
Kinematic tangent modulus	Ek	75[GPa]	Pa	Elastoplastic material model

MESH 1

1 In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **Build All**.

The default mesh cannot resolve this problem, so you need to refine it.

Once a mesh is created the mesh control domains are removed. In order to get them back, you need to clear the mesh.

2 In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **Clear Mesh**.

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

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

Size 1

1 In the **Model Builder** window, under **Component 1>Mesh 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 **Domain**.

4 Select Domain 2 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 `1.5`.

8 Click the **Build All** button.

Size

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

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

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

4 Click the **Build All** button.

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 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add** to include **para** as **Auxiliary parameter**.

5 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
para	`range(0,0.025,1)`

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

RESULTS

Stress (solid)

1 On the **3D Plot Group** toolbar, click **Plot**.

The default plot is from the last parameter value, which has no external loads. The residual stress and corresponding displacements are shown. You may want to reduce the deformation scale in order to get a better view.

2 In the **Model Builder** window, expand the **Stress (solid)** node.

3 In the **Model Builder** window, expand the **Results>Stress (solid)>Surface 1** node, then click **Deformation**.

4 In the **Deformation** settings window, locate the **Scale** section.

5 Select the **Scale factor** check box.

6 In the associated edit field, type `2000`.

7 On the **3D Plot Group** toolbar, click **Plot**.

Now, set back scaling to default and examine the stress state at maximum tension.

8 Clear the **Scale factor** check box.

9 In the **Model Builder** window, click **Stress (solid)**.

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

11 From the **Parameter value (para)** list, choose **0.25**.

12 On the **3D Plot Group** toolbar, click **Plot**.

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 6 only.

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

5 In the **Expression** edit field, type `solid.epe`.

6 On the **1D Plot Group** toolbar, click **Plot**.

1D Plot Group 3

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 6 only.

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

5 In the **Expression** edit field, type `solid.sz`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** edit field, type `solid.eZ`.

8 On the **1D Plot Group** toolbar, click **Plot**.

Add another study for the next load cycle.

ROOT

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

ADD STUDY

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

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

3 In the **Add study** window, click **Add Study**.

STUDY 2

Step 1: Stationary

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

2 In the **Stationary** settings window, locate the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add** to include **para** as **Auxiliary parameter**.

5 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
para	`range(1,0.025,2)`

6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Select the **Initial values of variables solved for** check box.

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

8 From the **Study** list, choose **Study 1, Stationary**.

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

RESULTS

1D Plot Group 2

1 In the **Model Builder** window, expand the **Results>1D Plot Group 2** node.

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

3 In the **Point Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Solution 2**.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 In the **Width** edit field, type 4.

7 On the **1D Plot Group** toolbar, click **Plot**.

1D Plot Group 3

1 In the **Model Builder** window, expand the **Results>1D Plot Group 3** node.

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

3 In the **Point Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Solution 2**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 In the **Width** edit field, type 4.

7 On the **1D Plot Group** toolbar, 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 In the **Add Physics** window, click **Add to Component**.

FATIGUE

Strain-Based 1

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

2 In the **Strain-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.

MATERIALS

Material 2

1 In the **Model Builder** window, under **Component 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
Fatigue ductility coefficient	epsilonf_CM	0.375	1	Coffin-Manson
Fatigue ductility exponent	c_CM	-0.60	1	Coffin-Manson
Fatigue strength coefficient	sigmaf_Basquin	1323[MPa]	Pa	Basquin
Fatigue strength exponent	b_Basquin	-0.097	1	Basquin
Young's modulus	E	210[GPa]	Pa	Basic

ADD STUDY

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

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

3 In the **Add study** window, click **Add Study**.

STUDY 3

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** 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
Solid Mechanics	×	Physics

4 Locate the **Values of Dependent Variables** section. Select the **Values of variables not solved for** check box.

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

6 From the **Study** list, choose **Study 2, Stationary**.

7 From the **Parameter value (para)** list, choose **All**.

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

RESULTS

Cycles to Failure (ftg)

1 In the **Results>Cycles to Failure (ftg)** settings window, click to expand the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 On the **3D plot group** toolbar, click **Plot**.

4 In the **Title** text area, type `Logarithm of lifetime in number of cycles`.

5 On the **3D plot group** toolbar, click **Plot**.

6 Right-click **Results>Cycles to Failure (ftg)** and choose **Rename**.

7 Go to the **Rename 3D Plot Group** dialog box and type `Fatigue life` in the **New name** edit field.

8 Click **OK**.

Make sure that the two elastoplastic studies do not solve for fatigue, if re-computed.

STUDY 1

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

2 Go to the **Rename Study** dialog box and type `Study 1 (First load cycle)` in the **New name** edit field.

3 Click **OK**.

STUDY 1 (FIRST LOAD CYCLE)

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1 (First load cycle)** 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	×	Physics

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 (Steady state load cycle)` in the **New name** edit field.

3 Click **OK**.

STUDY 2 (STEADY STATE LOAD CYCLE)

1 In the **Model Builder** window, expand the **Study 2 (Steady state load cycle)** 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	×	Physics

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, SWT)` in the **New name** edit field.

3 Click **OK**.