Created in COMSOL Multiphysics 5.6

Elastoplastic 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.

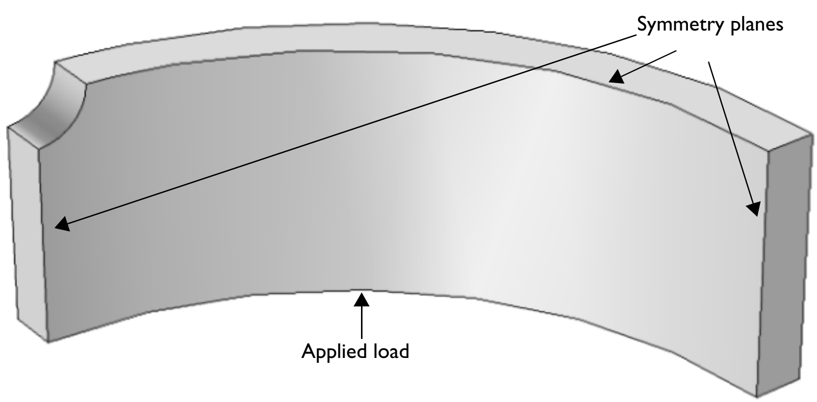


Figure 1: Model geometry with constrained and loaded faces.

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:
  - $\sigma_f' = 1323$ MPa
  - $b = -0.097$
  - $\varepsilon_f' = 0.375$
  - $c = -0.60$

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 parameter 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 equivalent plastic strain history is shown in Figure 7.

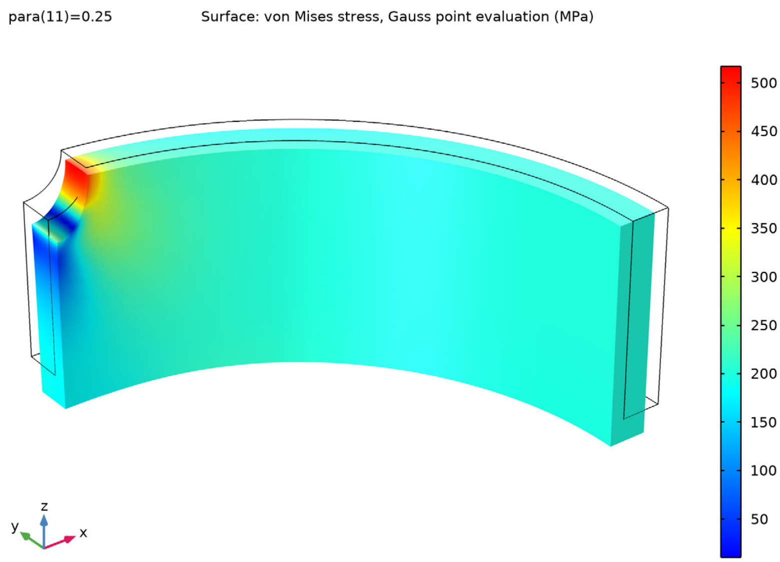


Figure 2: von Mises stress level at the first maximum load

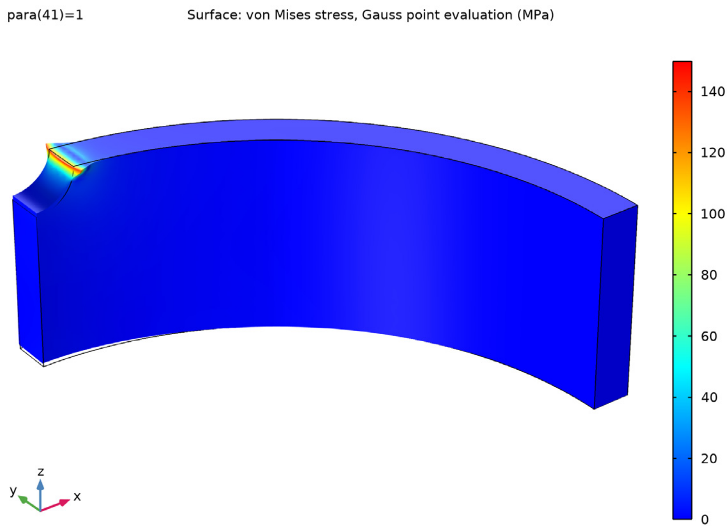


Figure 3: Residual equivalent stress after completion of first load cycle

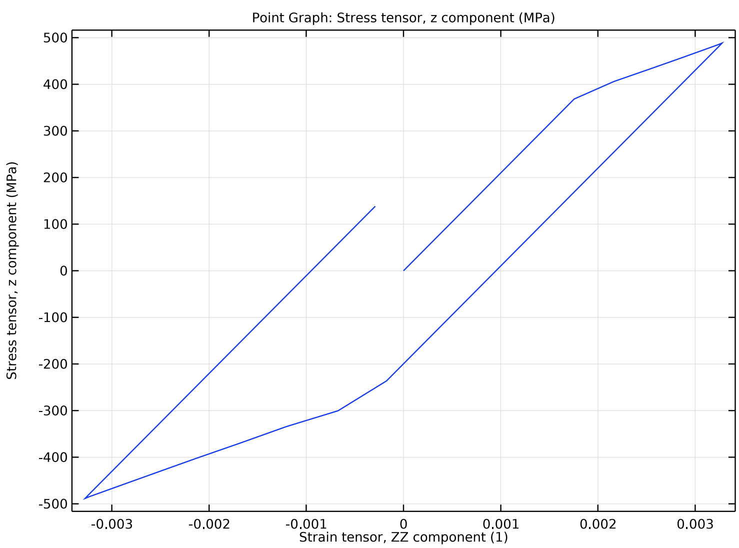


Figure 4: Stress as function of strain during the first load cycle.

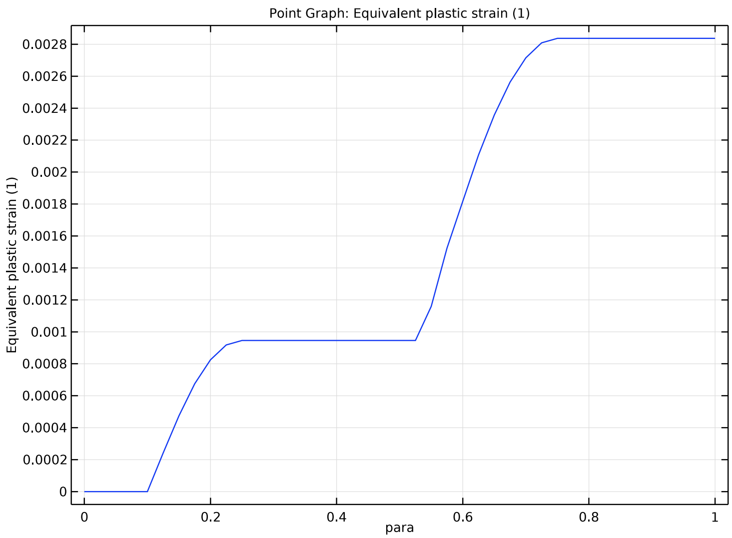


Figure 5: Accumulated equivalent plastic strain during the first load cycle

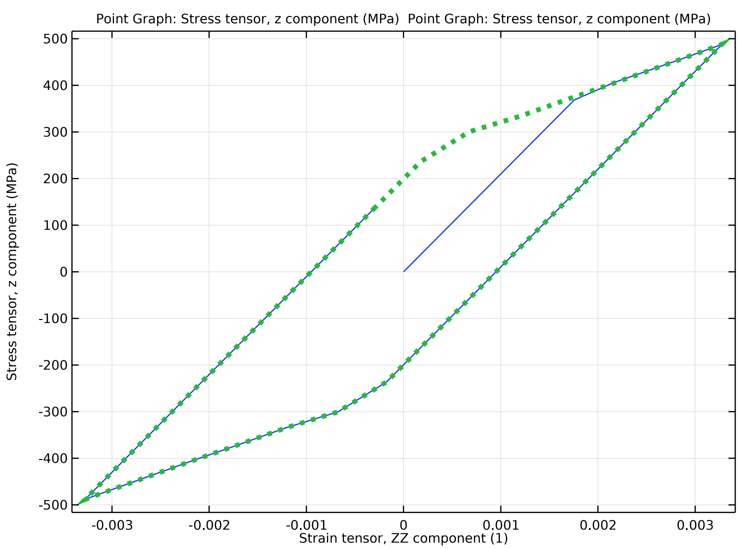


Figure 6: Stress as function of strain after two load cycles

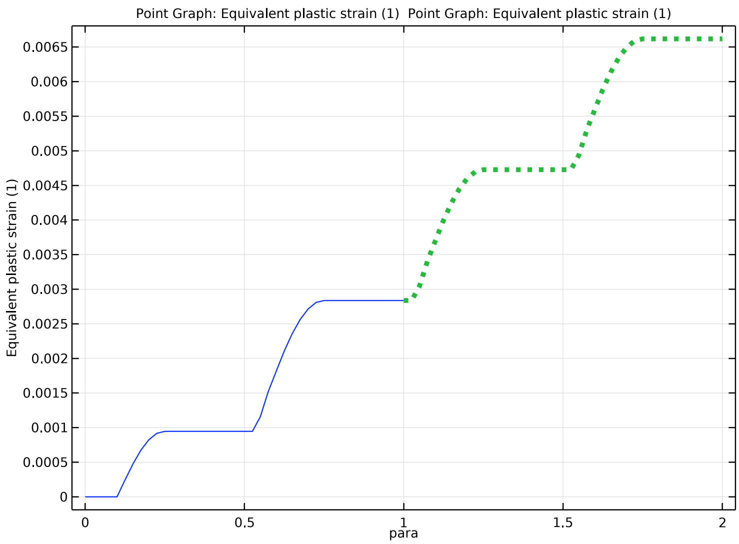


Figure 7: Accumulated equivalent plastic strain during two load cycles

The result of the fatigue evaluation is shown in Figure 8 below. The most critical point has a computed life of approximately 6700 load cycles.

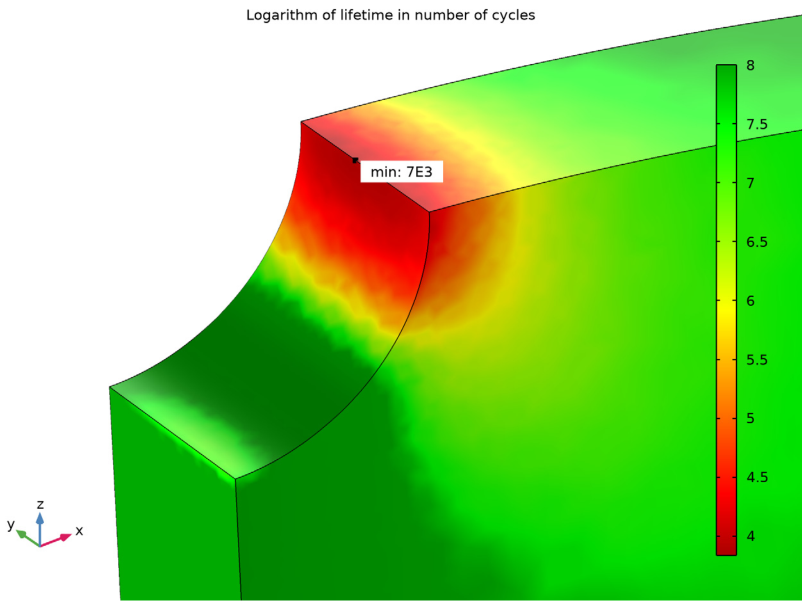


Figure 8: Lifetime plot

Notes About the COMSOL Implementation

Two studies are used 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. This approach is not necessary. since it in the **Fatigue** study step is possible to select part of a study as defining a load cycle.

Application Library path: `Fatigue_Module/Strain_Based/cylinder_with_hole_plastic`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 100.
4 In the **Height** text field, type 100.

Cylinder 2 (cyl2)

1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 90.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **cyl1** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.
5 Select the object **cyl2** only.
6 Click **Build All Objects**.

Cylinder 3 (cyl3)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type 10.

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

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

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

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

8 Click **Build All Objects**.

Difference 2 (dif2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **dif1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **cyl3** only.

6 Click **Build All Objects**.

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

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

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

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

6 Click **Build All Objects**.

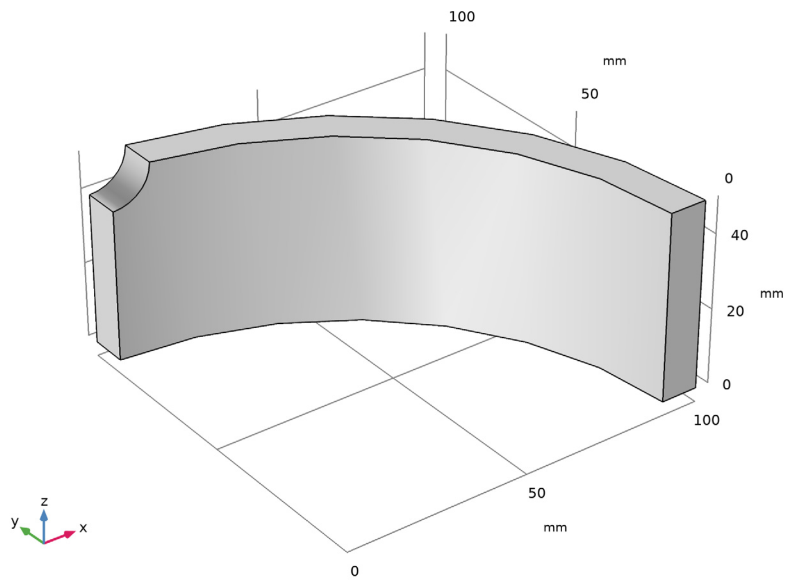
Intersection 1 (int1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Intersection**, click **Build All Objects**.

4 Click the **Zoom Extents** button in 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 (cyl4)

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

2 In the **Settings** window for **Cylinder**, locate the **Axis** section.

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

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

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

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

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

8 Click **Build All Objects**.

Intersection 2 (int2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Intersection**, locate the **Intersection** section.

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

5 Click **Build All Objects**.

Delete Entities 1 (del1)

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

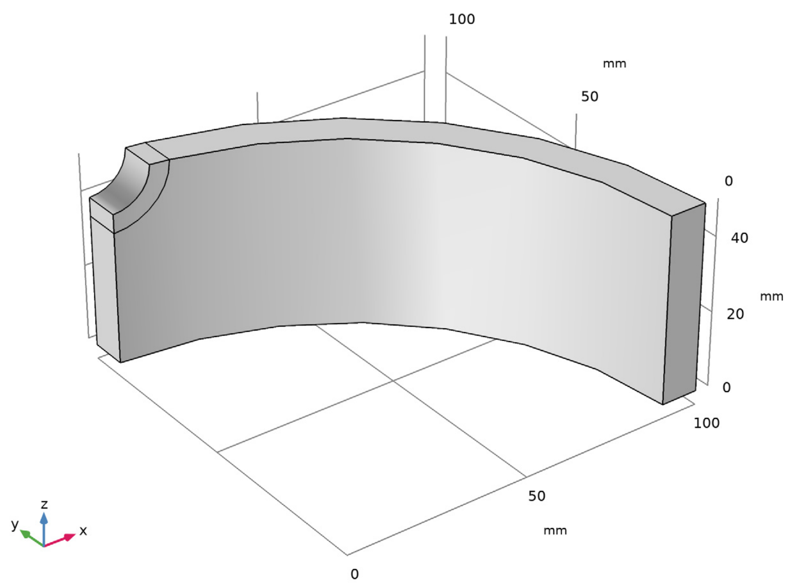
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 Select the object **cyl4** only.

Form Union (fin)

In the **Geometry** toolbar, click **Build All**.



Mesh Control Domains 1 (mcd1)

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

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

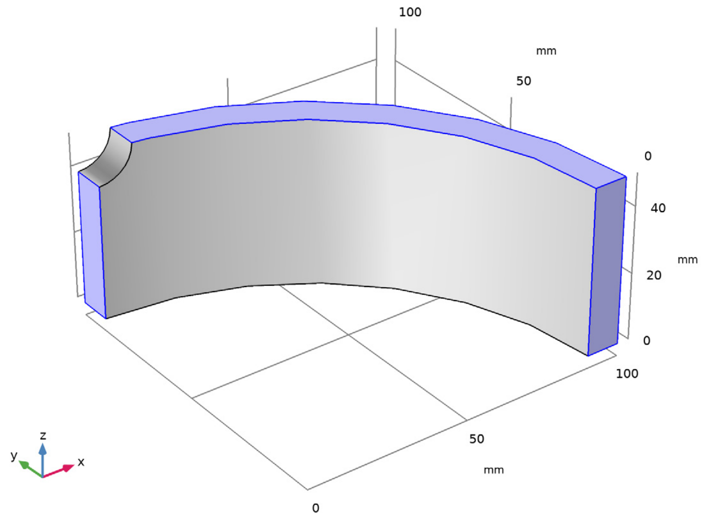
3 In the **Geometry** toolbar, click **Build All**.

SOLID MECHANICS (SOLID)

Symmetry 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **More Constraints>Symmetry**.

2 Select Boundaries 1, 6, and 7 only.



GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`para`	`0`	0	`Load cycle control`

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 Select Boundary 3 only.

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

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

`0`	x

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

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Plasticity**.

2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.

3 Find the **Isotropic hardening model** subsection. From the list, choose **Perfectly plastic**.

4 Find the **Kinematic hardening model** subsection. From the list, choose **Linear**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	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
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 (comp1)** 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 **Mesh** toolbar, click **Clear Mesh**.

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

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

Size 1

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

2 In the **Settings** window for **Size**, 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 text field, type `1.5`.

8 Click **Build All**.

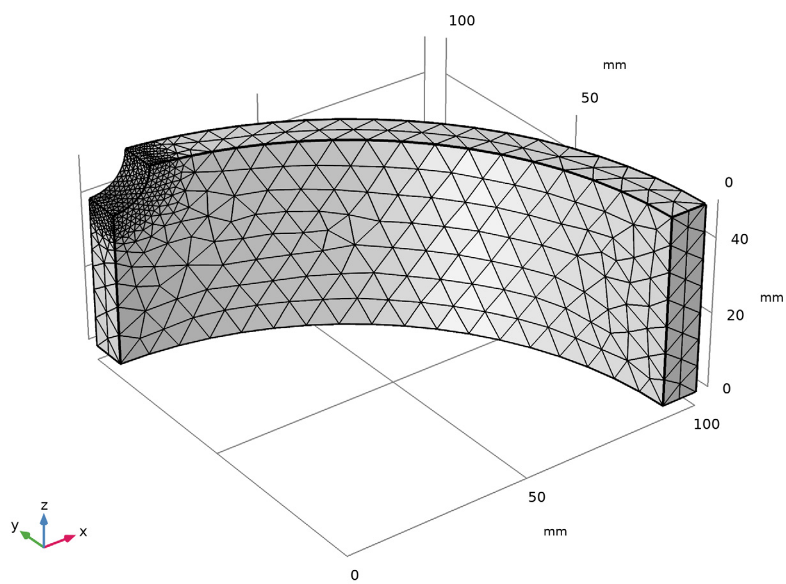
Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

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

4 Click **Build All**.



STUDY 1

Step 1: Stationary

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

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

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

4 Click **Add** to include `para` as continuation parameter.

5 In the table, enter the following settings:

Parameter name	Parameter value list
para (Load cycle control)	`range(0,0.025,1)`

6 In the **Model Builder** window, click **Study 1**.

7 In the **Settings** window for **Study**, type `Study 1 (First Load Cycle)` in the **Label** text field.

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

RESULTS

Stress (solid)

1 In the **Stress (solid)** 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.

Surface 1

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

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

3 From the **Unit** list, choose **MPa**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

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

4 In the associated text field, type `2000`.

5 In the **Stress (solid)** toolbar, click **Plot**.

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

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

Stress (solid)

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

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

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

4 In the **Stress (solid)** toolbar, click **Plot**.

Plastic Strain

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

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

Point Graph 1

1 In the **Plastic Strain** toolbar, click **Point Graph**.

2 Select Point 6 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>solid.epe - Equivalent plastic strain**.

4 In the **Plastic Strain** toolbar, click **Plot**.

Stress vs. Strain

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

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

Point Graph 1

1 In the **Stress vs. Strain** toolbar, click **Point Graph**.

2 Select Point 6 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m²>solid.sz - Stress tensor, z component**.

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

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>Strain tensor (material and geometry frames)>solid.eZZ - Strain tensor, ZZ component**.

6 In the **Stress vs. Strain** toolbar, click **Plot**.

Add another study for the next load cycle.

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 **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list
para (Load cycle control)	`range(1,0.025,2)`

5 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

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

7 From the **Study** list, choose **Study 1 (First Load Cycle), Stationary**.

8 In the **Model Builder** window, click **Study 2**.

9 In the **Settings** window for **Study**, type `Study 2 (Steady State Load Cycle)` in the **Label** text field.

10 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

RESULTS

Point Graph 2

1 In the **Model Builder** window, under **Results>Plastic Strain** right-click **Point Graph 1** and choose **Duplicate**.

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

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

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

5 In the **Width** text field, type 4.

6 In the **Plastic Strain** toolbar, click **Plot**.

Point Graph 2

1 In the **Model Builder** window, under **Results>Stress vs. Strain** right-click **Point Graph 1** and choose **Duplicate**.

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

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

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

5 In the **Width** text field, type 4.

6 In the **Stress vs. Strain** toolbar, click **Plot**.

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 (First Load Cycle)** and **Study 2 (Steady State Load Cycle)**.

5 Click **Add to Component 1** in the window toolbar.

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

FATIGUE (FTG)

Strain-Based 1

1 Right-click **Component 1 (comp1)>Fatigue (ftg)** and choose the boundary evaluation **Strain-Based**.

2 In the **Settings** window for **Strain-Based**, locate the **Boundary Selection** section.

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

4 Locate the **Solution Field** section. From the **Physics interface** list, choose **Solid Mechanics (solid)**.

5 Locate the **Evaluation Settings** section. Find the **Critical plane settings** subsection. In the *Q* text field, type 16.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, 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	Variable	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 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 box for **Solid Mechanics (solid)**.

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 (Steady State Load Cycle), Stationary**.

5 In the **Model Builder** window, click **Study 3**.

6 In the **Settings** window for **Study**, type `Study 3 (Fatigue, SWT)` in the **Label** text field.

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

RESULTS

Fatigue Life

1 In the **Settings** window for **3D Plot Group**, type `Fatigue Life` in the **Label** text field.

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

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

Max/Min Surface 1

1 In the **Model Builder** window, expand the **Fatigue Life** node, then click **Max/Min Surface 1**.

2 In the **Settings** window for **Max/Min Surface**, click to expand the **Advanced** section.

3 Locate the **Text Format** section. In the **Display precision** text field, type `1`.

4 In the **Fatigue Life** toolbar, click **Plot**.