Created in COMSOL Multiphysics 6.1

Notch Approximation to Low-Cycle Fatigue Analysis of Cylinder with a Hole

Introduction

A load carrying component of a structure is subjected to multiaxial cyclic loading during which a 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 an elastoplastic analysis with linear kinematic hardening, while the second is a linear elastic analysis with Neuber correction for plasticity, based on the Ramberg-Osgood model. This example explores the second method. In the model Elastoplastic Low-Cycle Fatigue Analysis of Cylinder with a Hole, the same problem is solved using the full elastoplastic 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 that varies in time.

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

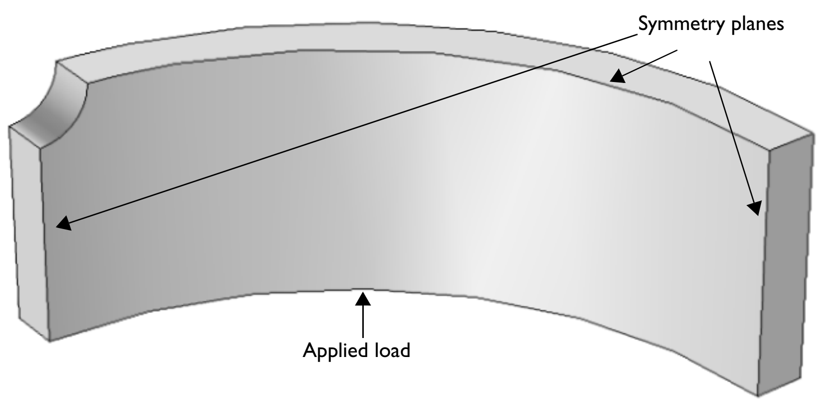


Figure 1: Model geometry with constrained and loaded faces.

MATERIAL PROPERTIES

- Elastic data: Isotropic with E = 210 GPa, ν = 0.3

- Cyclic Ramberg-Osgood plasticity data: K' = 1550 MPa, n' = 0.16
- 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 varying between +200 MPa and -200 MPa.

Results and Discussion

The von Mises stress distribution at maximum load is shown in Figure 2. Notice that the yield limit(380 = MPa) is extensively exceeded, so in reality significant plastic strains can be expected.

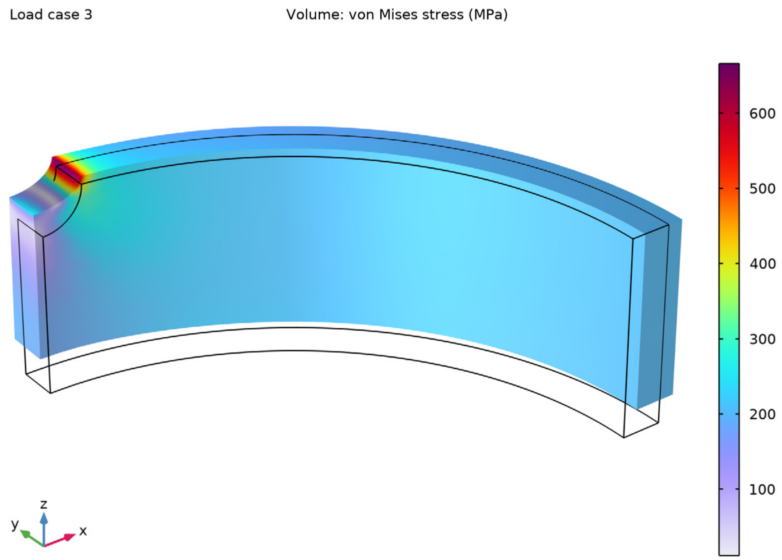


Figure 2: von Mises stress level at the first maximum load

The computed number of cycles to fatigue is shown in Figure 3. It is slightly below 5000 cycles.

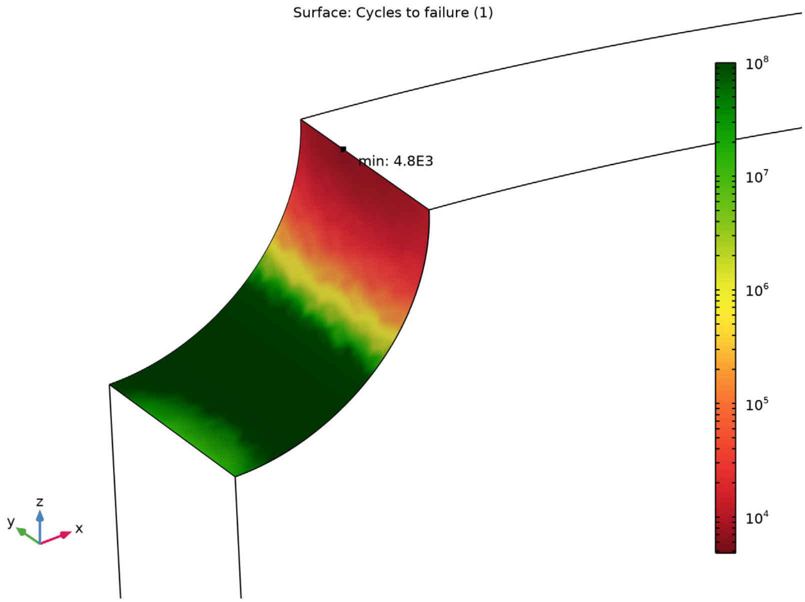


Figure 3: The distribution of expected number of cycles at the hole.

Notes About the COMSOL Implementation

When using the elastic approach to low cycle fatigue, it is only the peak loads that are important, not the load cycle as such. The model includes both the maximum and the minimum loads, since some of the criteria are sensitive to the sign of the stresses. It is also necessary to include the “zero” load case, which contains no loads.

Application Library path: `Fatigue_Module/Strain_Based/cylinder_with_hole_elastic`

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. Click to 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. Click to 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**.

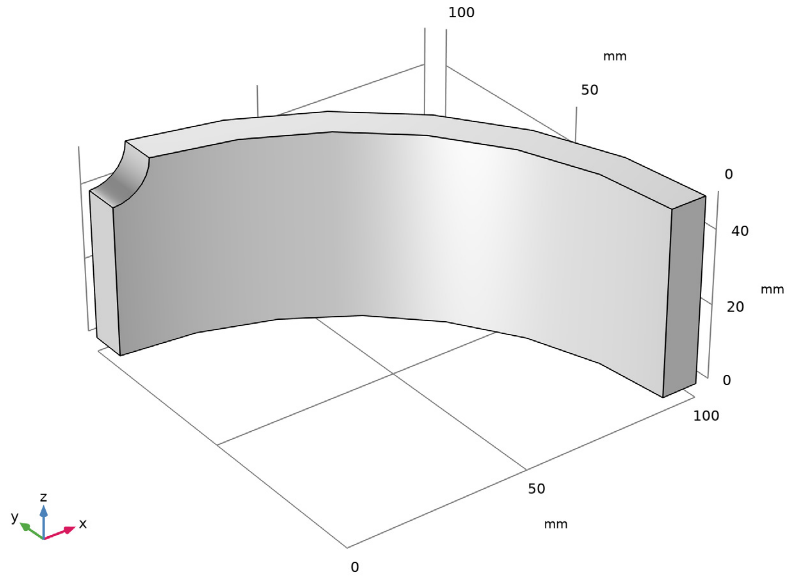
Intersection 1 (int1)

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.



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.

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]	z

5 In the **Physics** toolbar, click **Load Group** and choose **New Load Group**.

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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic

MESH 1

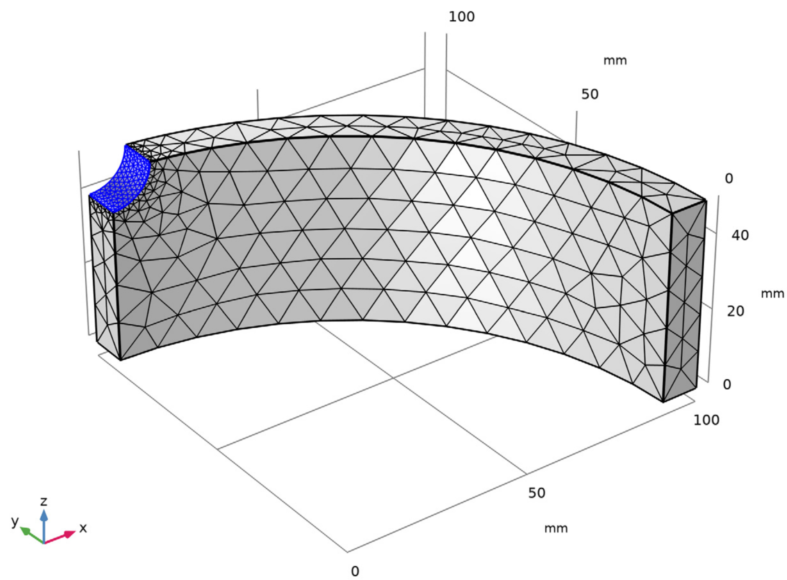
Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size 1

1. 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 **Boundary**.
4. Select Boundary 4 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `1.5`.

8 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 **Define load cases** check box.

In an analysis of this type you should always supply the maximum and the minimum load, and also a zero solution.

4 Click **Add** three times.

5 In the table, enter the following settings:

Load case	lg1	Weight
`Load case 1`	√	`0`
`Load case 2`	√	`1.0`
`Load case 3`	√	`-1.0`

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

RESULTS

Volume 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 In the **Stress (solid)** 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 box for **Study 1**.
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 Select Boundary 4 only.

   As the elastic approach to strain based fatigue is based on a notch assumption, it only makes sense to select boundaries at the hole.
3 In the **Settings** window for **Strain-Based**, locate the **Fatigue Model Selection** section.
4 From the **Solution type** list, choose **Elastic solution with notch assumption**.
5 Locate the **Solution Field** section. From the **Physics interface** list, choose **Solid Mechanics (solid)**.

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	`solid.E`	Pa	Basic
Poisson's ratio	nu	`solid.nu`	1	Basic
Cyclic hardening coefficient	K_ROcyclic	`1550[MPa]`	Pa	Ramberg-Osgood
Cyclic hardening exponent	n_ROcyclic	`0.16`	1	Ramberg-Osgood

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 2

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 1, Stationary**.

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

RESULTS

Cycles to Failure (ftg)

The plot of cycles to failure shown in Figure 3 is created automatically.

1 In the **Model Builder** window, expand the **Cycles to Failure (ftg)** node.

Marker 1

1 In the **Model Builder** window, expand the **Results>Cycles to Failure (ftg)>Surface 1** node, then click **Marker 1**.

2 In the **Settings** window for **Marker**, locate the **Text Format** section.

3 In the **Display precision** text field, type 2.

4 In the **Cycles to Failure (ftg)** toolbar, click **Plot**.