Created in COMSOL Multiphysics 6.1

Standing Contact Fatigue

Introduction

In the vicinity of a point where two parts are in contact, a surface crack has been observed in a component. There is no relative movement between the two contact interfaces; however, the contact pressure varies in time. After a long operating time, a surface crack has been observed. A further analysis of the failed component reveals that multiple cracks are present on the subsurface level. In order to gain understanding of the failure mechanism, an analysis of standing contact fatigue is performed.

The component is surface hardened. Through this procedure the near surface layer significantly changes its material properties. Both the strength, hardening and the fatigue properties are affected. Moreover a residual stress is introduced throughout the depth.

Model Definition

In a standing contact fatigue test a spherical indenter of a hard material is pressed against the tested material. The failed component is made of surface hardened chromoly steel. Through the process of surface hardening the material has undergone metallurgical changes and can be considered to consist of three distinct layers, as shown in Figure 1.

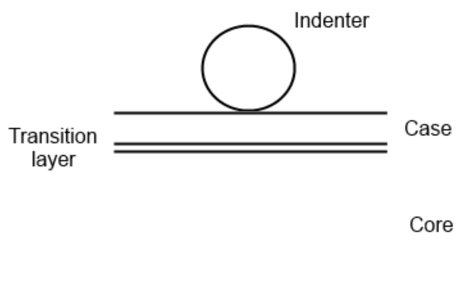


Figure 1: Standing contact fatigue material testing.

The harder surface layer is commonly called the case and the unmodified material inside the component is called the core. Between these two layers there is a thin layer called the transition layer where the material properties varies strongly through the depth.

In the test the indenter is pressed and released so that the maximum contact pressure varies between 2.7 GPa and 0 GPa.

MATERIAL PROPERTIES

The elastic properties are the constant throughout the depth, with Young’s modulus and Poisson's ratio being E = 206 GPa and ν = 0.31, respectively.

The plastic properties varies in the three layers. The hardening in the case can be considered linear and isotropic defined by

$$\sigma_{ys} = \sigma_{ys0} + \frac{E_{Tiso}}{1 - \frac{E_{Tiso}}{E}} \varepsilon_{ep} \tag{1}$$

where σ_{ys} is the yield stress, and ε_{ep} is the equivalent plastic strain. The material constants initial yield stress and isotropic tangent modulus are set to $\sigma_{ys0} = 570$ MPa and $E_{Tiso} = 69$ GPa, respectively.

The core follows an isotropic exponential hardening function, also called Ludwik hardening, given by the expression

$$\sigma_{ys} = \sigma_{ys0} + k(\varepsilon_{ep})^{n}$$

where the material constants initial yield stress, strength coefficient, and hardening exponent are given by $\sigma_{ys0} = 515$ MPa, $k = 2.47$ GPa and $n = 0.55$.

Plastic data are not directly available for the transition layer. However since the material behavior varies significantly in the transition layer it must be dependent on the distance from the surface. Moreover at the interface with the core the plastic law should follow the ore hardening function and at the interface with the case it should follow the case hardening function. In order to use a single hardening function in the transition layer, the linear isotropic hardening in the case is transformed into a Ludwik hardening with following material constants: $\sigma_{ys0} = 570$ MPa, $k = 103$ GPa, and $n = 1.0$. The variation of the parameters in the transition layer is assumed to be

- linear in the initial yield stress,
- linear in the logarithm of the strength coefficient,
- linear in the hardening exponent.

This assumption lead to following expression for the material parameters:

$$\sigma_{ys0} = 570 - 55s \quad \text{MPa}$$

$$k = 10^{(11 - 1 \cdot 62s)}$$

$$n = 1 - 0 \cdot 45s$$

where s is a local coordinate in the transition layer which varies linearly with the depth and is 0 at case interface and 1 at core interface. The stress-strain curve at different levels in the transition layer is shown in Figure 2.

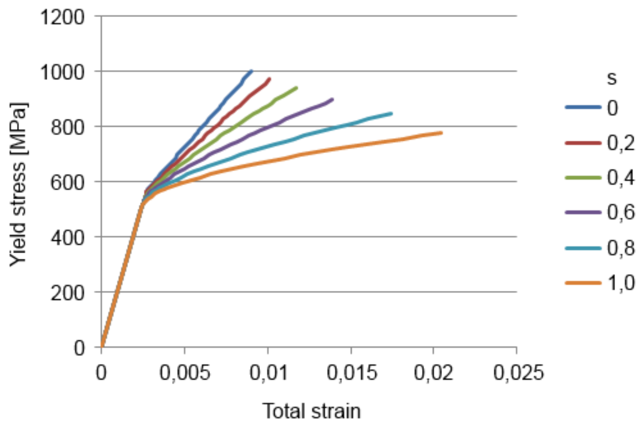


Figure 2: Hardening functions in the transition layer. $s = 0$ *denotes the interface with the case and* $s = 1$ *denotes the interface with the core.*

RESIDUAL STRESS

The hardening process introduces a compressive stress in the case and a tensile stress in the core. The two in-plane components of the direct stress in the case are –500 MPa and in the core 100 MPa. The profile of the residual stress in the transition layer is assumed to vary linearly.

FATIGUE DATA

The Dang Van model is used to evaluate fatigue. For the case material the Dang Van material parameters are $a = 0.19$ and $b = 282$ MPa. For the core material the Dang Van material parameters are $a = 0.23$ and $b = 248$ MPa.

NUMERICAL MODEL

The tested material volume is large in comparison with the indenter which has a radius, r_{i}, of 7,0 mm. In the model the indenter is replaced with a Hertzian contact pressure distribution obtained from the following set of expressions

$$a_{\mathrm{i}} = \left(\frac{3 \cdot P \cdot r_{\mathrm{i}}}{4 \cdot E_{\mathrm{s}}}\right)^{1/3}$$

$$E_s = \frac{E}{2(1-\nu^2)}$$

where a_i is the contact radius, and P is the contact load. The relation between the contact load and the pressure distribution is given by

$$p_0 = \frac{3 \cdot P}{2 \cdot \pi \cdot a_i^2} = \frac{1}{\pi}\left(\frac{6 \cdot P \cdot E_s}{r_i^2}\right)$$

$$p(r) = p_0\left(1 - \frac{r}{a_i}\right)^{1/2}$$

where p_0 is the peak contact pressure in the contact zone and p is the contact pressure at the radial coordinate r.

In the testing a maximum pressure of 2.7 GPa is applied. This corresponds to a maximum load of 384 N and a maximum contact radius of 0.26 mm. In the model, the size of the geometry must be sufficiently large so that the far boundaries do not affect the stress state at the contact zone.

Since the geometry is axially symmetric, a 2D axial symmetry Solid Mechanics interface is used. The radius and the height of the model are selected to 5 mm. The case is 0.5 mm deep and the transition layer is 0.1 mm. A roller boundary condition is used at the bottom and at the far radial boundary.

Results and Discussion

The stress contours when the load is released after two load cycles, are shown in Figure 3. Residual stresses are present, both from the hardening and from the plastic deformation under the indenter.

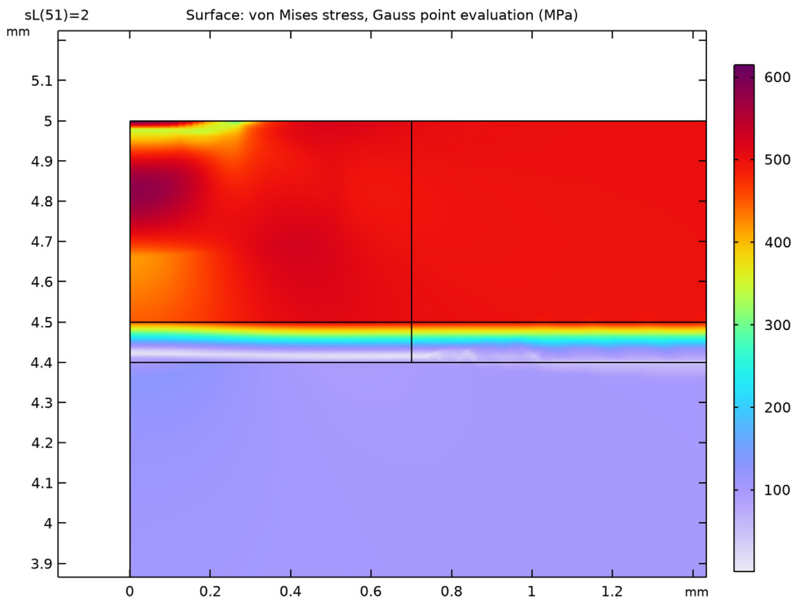


Figure 3: The equivalent stress after two load cycles.

A fatigue analysis requires a stable load cycle. For this purpose the development of the plastic deformation is evaluated. Figure 4 displays where plastic deformation has occurred. Figure 5 displays the magnitude of the equivalent plastic strain. In Figure 6 it can be seen that plasticity develops only during the first load cycle. Every consecutive load cycle is elastic. The second load cycle can therefore be seen as a stable load cycle and is used in the subsequent fatigue study

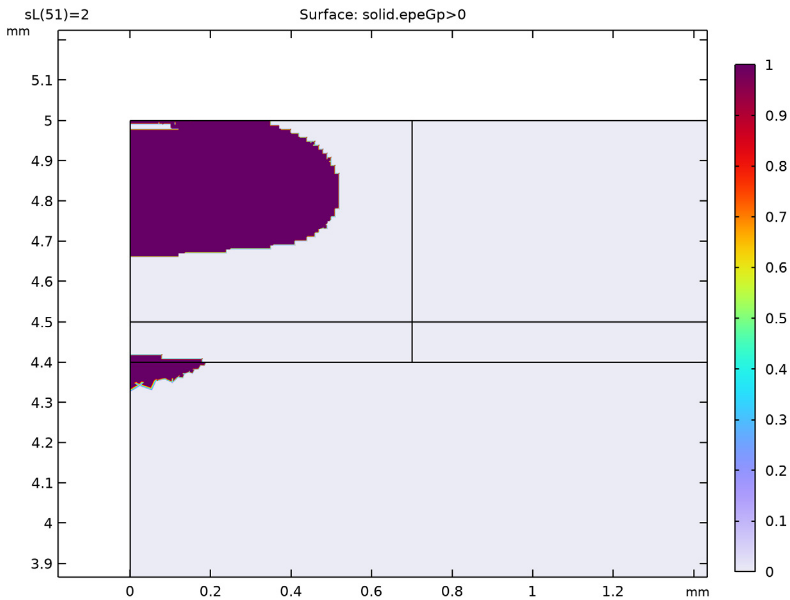


Figure 4: Plastically deformed volume after two load cycles.

.

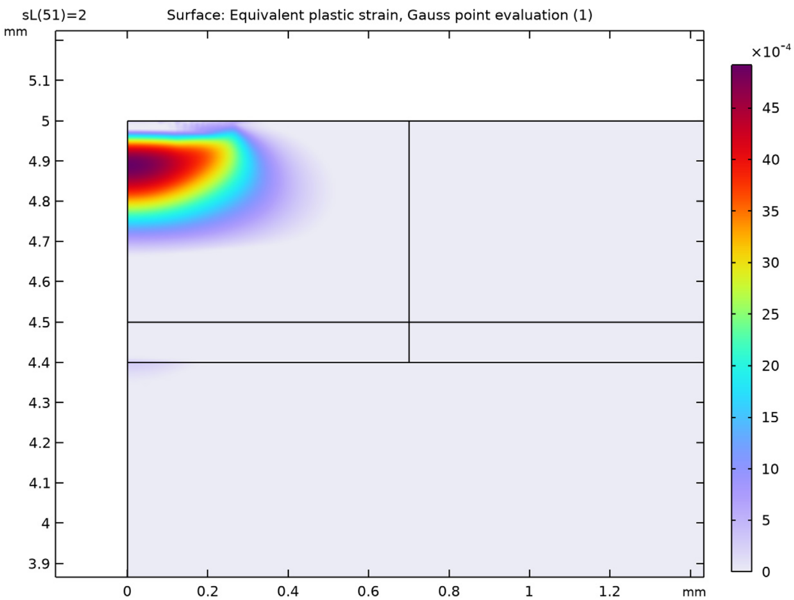


Figure 5: Equivalent plastic strain after two load cycles.

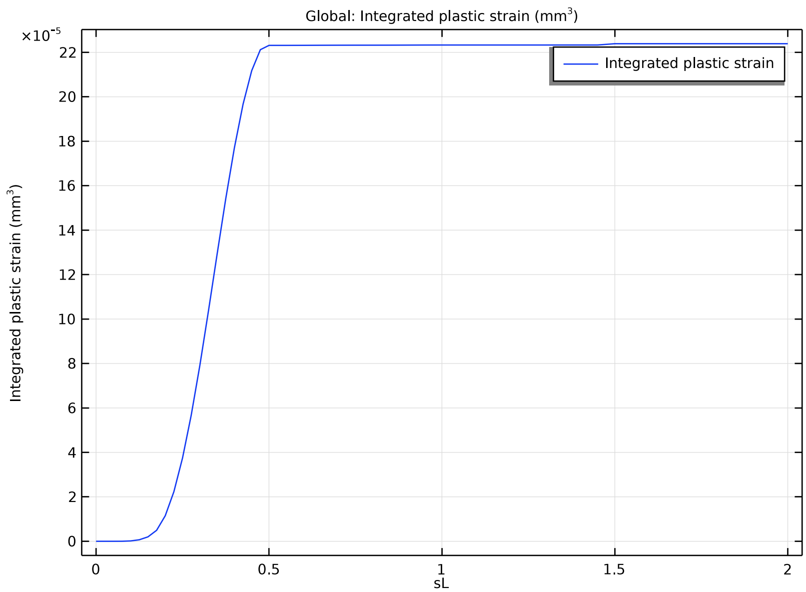


Figure 6: Development of plastic strains, integrated over the volume

In Figure 7 and Figure 8, stress contours are shown at the peak load in the second load cycle. Both the highest equivalent stress and the maximum shear stress are found below the surface. This is commonly seen in contact problems.

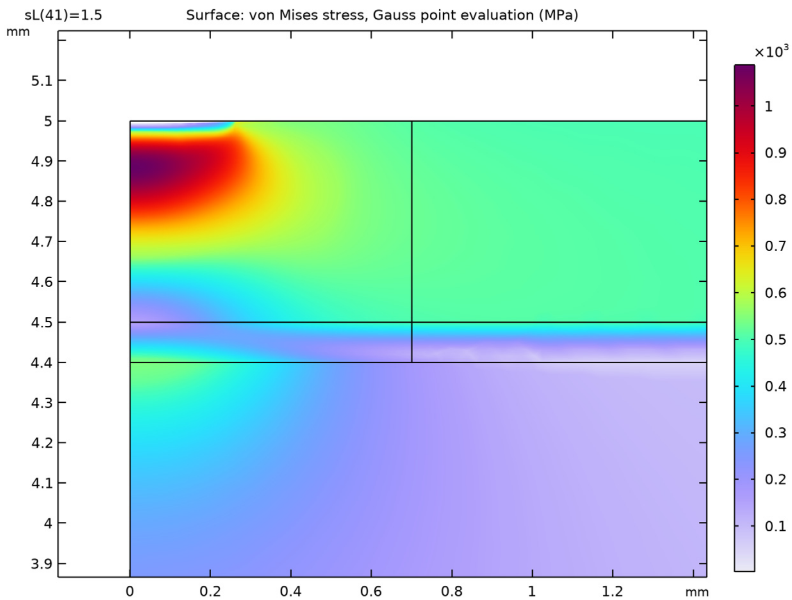


Figure 7: Equivalent stress at peak load.

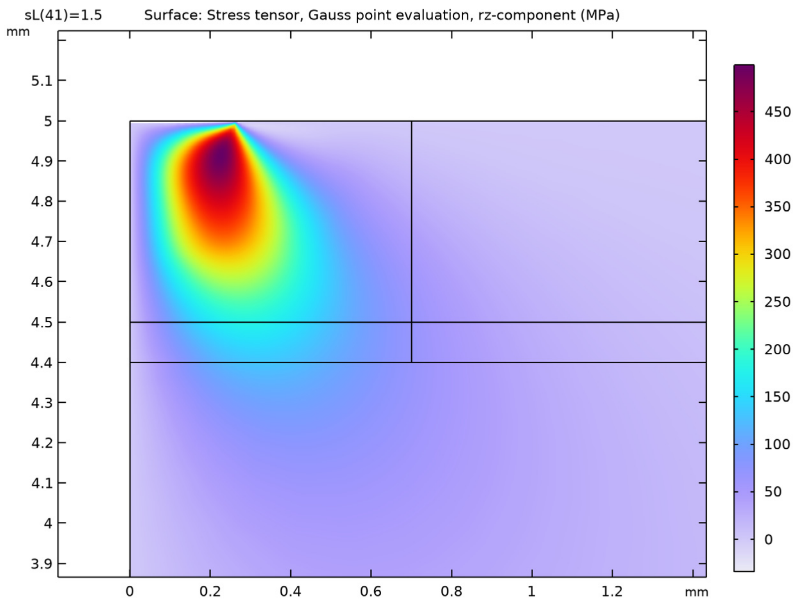


Figure 8: The stress contours of the shear stress at peak load.

The fatigue analysis shows that the highest risk of fatigue is in the case about 0.1 mm below the surface. At the interface between the core and the transition layer there is also an increased risk of fatigue, see Figure 9. By inspecting the material far away from the contact, one can observe that the fatigue usage factor in the case layer has a negative value while in the core it has a positive value. This is caused by the combination of the residual stress from surface hardening and almost zero stress amplitude. A compressive hydrostatic state is beneficial for fatigue prevention while positive hydrostatic stress has a negative influence on fatigue.

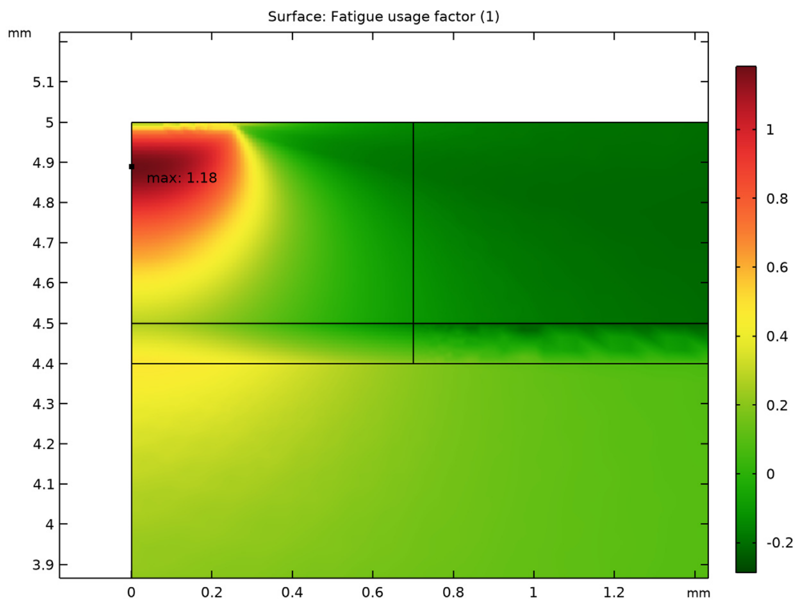


Figure 9: Fatigue usage factor.

Application Library path: `Fatigue_Module/Stress_Based/standing_contact_fatigue`

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 **2D Axisymmetric**.

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**.

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
`H`	`5 [mm]`	0.005 m	`Model height`
`dH`	`0.5 [mm]`	5E-4 m	`Case depth`
`dT`	`0.1 [mm]`	1E-4 m	`Transition depth`
`W`	`5 [mm]`	0.005 m	`Model width`
`dW`	`0.7 [mm]`	7E-4 m	`Fine zone width`
`P`	`384 [N]`	384 N	`Max load`
`E`	`200 [GPa]`	2E11 Pa	`Young's modulus`
`nu`	`0.30`	0.3	`Poisson's ratio`
`rho`	`7800 [kg/m^3]`	7800 kg/m³	`Density`
`Es`	`E/(2*(1-nu^2))`	1.0989E11 Pa	`Hertzian contact stiffness`
`ri`	`7 [mm]`	0.007 m	`Indenter radius`
`sL`	`0`	0	`Load magnifier`

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**.

Rectangle 1 (r1)

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

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

3 In the **Height** text field, type `H-dH-dT`.

4 In the **Width** text field, type `W`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `dW`.
4 In the **Height** text field, type `dT`.
5 Locate the **Position** section. In the **z** text field, type `H-dH-dT`.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `W-dW`.
4 In the **Height** text field, type `dT`.
5 Locate the **Position** section. In the **r** text field, type `dW`.
6 In the **z** text field, type `H-dH-dT`.

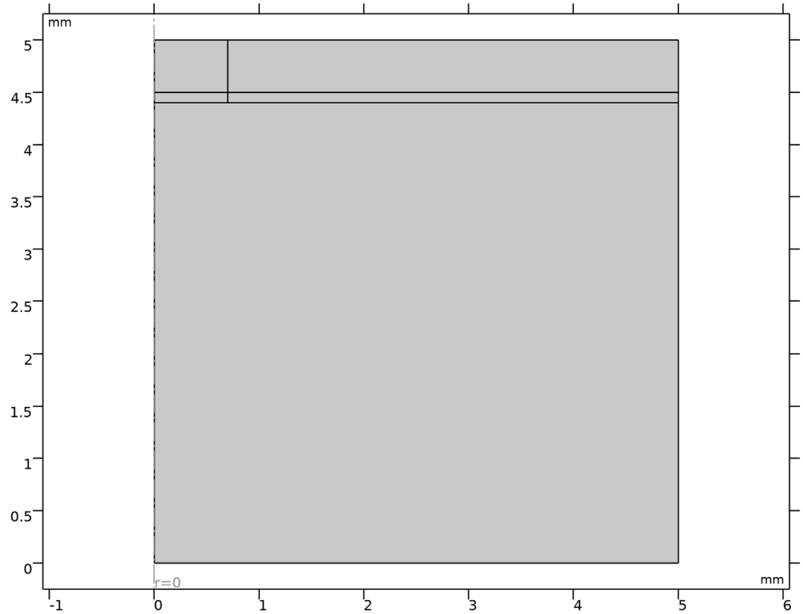
Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `dW`.
4 In the **Height** text field, type `dH`.
5 Locate the **Position** section. In the **z** text field, type `H-dH`.

Rectangle 5 (r5)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `W-dW`.
4 In the **Height** text field, type `dH`.
5 Locate the **Position** section. In the **r** text field, type `dW`.
6 In the **z** text field, type `H-dH`.

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



DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Locate the **Advanced** section. From the **Frame** list, choose **Material (R, PHI, Z)**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

3 From the E list, choose **User defined**. In the associated text field, type `E`.

4 From the ν list, choose **User defined**. In the associated text field, type `nu`.

5 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

Plasticity 1

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

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

3 Click **Clear Selection**.

4 Select Domains 3 and 5 only.

5 Locate the **Plasticity Model** section. From the σ_{ys0} list, choose **User defined**. In the associated text field, type `570 [MPa]`.

6 Find the **Isotropic hardening model** subsection. From the E_{Tiso} list, choose **User defined**. In the associated text field, type `69 [GPa]`.

Linear Elastic Material 1

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

Plasticity 2

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

Plasticity Case

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1** click **Plasticity 1**.

2 In the **Settings** window for **Plasticity**, type `Plasticity Case` in the **Label** text field.

Plasticity Core

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1** click **Plasticity 2**.

2 In the **Settings** window for **Plasticity**, type `Plasticity Core` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domain 1 only.

5 Locate the **Plasticity Model** section. From the σ_{ys0} list, choose **User defined**. In the associated text field, type `515 [MPa]`.

6 Find the **Isotropic hardening model** subsection. From the list, choose **Ludwik**.

7 From the k list, choose **User defined**. In the associated text field, type `2.47 [GPa]`.

8 From the n list, choose **User defined**. In the associated text field, type `0.55`.

GLOBAL DEFINITIONS

Transition function

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Transition function` in the **Label** text field.

3 Locate the **Definition** section. In the table, enter the following settings:

t	f(t)
H-dH-dT	1
H-dH	0

4 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	m

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
s	int1(z)		Transition layer position
nT	1-0.45*s		Hardening exponent transition layer
kT	10^(-1.62*s+11) [Pa]	Pa	Strength coefficient transition layer
s0T	(570e6*(1-s)+515e6*s) [Pa]	Pa	Initial yield stress transition layer

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Plasticity Transition

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

2 In the **Settings** window for **Plasticity**, type `Plasticity Transition` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domains 2 and 4 only.

5 Locate the **Plasticity Model** section. From the σ_{ys0} list, choose **User defined**. In the associated text field, type `s0T`.

6 Find the **Isotropic hardening model** subsection. From the list, choose **Ludwik**.

7 From the k list, choose **User defined**. In the associated text field, type `kT`.

8 From the n list, choose **User defined**. In the associated text field, type `nT`.

GLOBAL DEFINITIONS

Residual stress

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Residual stress` in the **Label** text field.

3 Locate the **Definition** section. In the table, enter the following settings:

t	f(t)
0	-500
1	100

4 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
int2	MPa

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the S_0 table, enter the following settings:

`int2(s)`	0	0
0	`int2(s)`	0
0	0	0

Roller 1

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

2 Select Boundaries 2 and 13–15 only.

Prescribe load from the indenter. Both the contact pressure and the contact radius vary with load magnitude. Define the load as a function. Define the contact pressure and the contact radius as variables.

GLOBAL DEFINITIONS

Periodic load

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Periodic load` in the **Label** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `0.5*(1-cos(x*2*pi))`.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Variables 1**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Unit**	**Description**
`ai`	`(3/4*P*an1(sL)*ri/Es)^(1/3)`	m	`Indentation radius`
`p0`	`3*P*an1(sL)/(2*pi*ai*ai)`	N/m²	`Max pressure`

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 Select Boundary 7 only.

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

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `if(r<ai,p0*sqrt(1-(r/ai)^2),0)`.

Create a fine mesh in the vicinity of the contact area.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

4 Select Domains 2 and 3 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 4, 6, and 7 only.

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

4 In the **Number of elements** text field, type `70`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 5 and 10 only.

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

4 In the **Number of elements** text field, type `50`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 3 and 8 only.

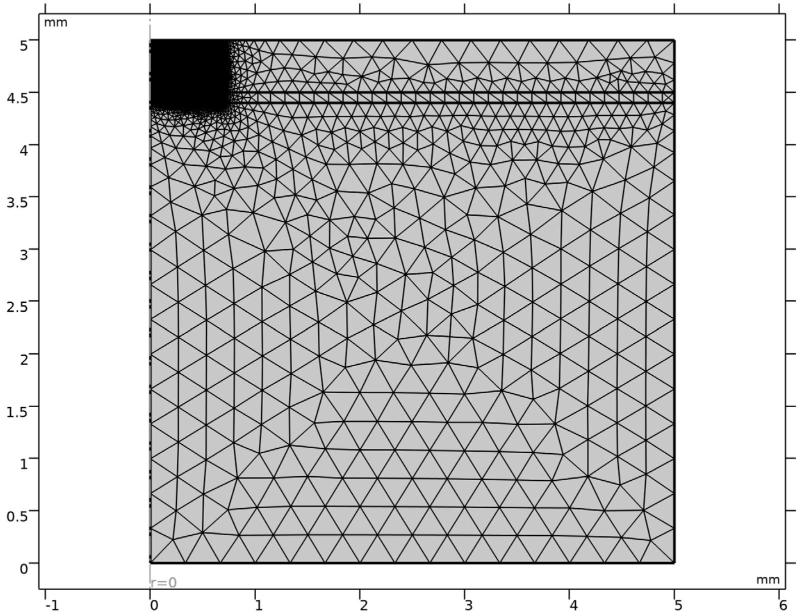
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `10`.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

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

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sL (Load magnifier)	`range(0.0,0.025,0.5)` `range(0.55,0.05,2)`	

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

RESULTS

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. In the associated text field, type `1`.

Surface 1

1 In the **Model Builder** window, expand the **Stress, 3D (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. In the associated text field, type `10`.

Stress (solid)

Evaluate development of plasticity.

Plastically deformed volume

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Plastically deformed volume` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Plastically deformed volume** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Equivalent plastic strain, Gauss point evaluation**.

3 Locate the **Expression** section. In the **Expression** text field, type `solid.epeGp>0`.

Deformation

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

2 Right-click **Results>Plastically deformed volume>Surface 1>Deformation** and choose **Delete**.

Plastic strain

1 In the **Model Builder** window, right-click **Plastically deformed volume** and choose **Duplicate**.

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

Surface 1

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

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

3 In the **Expression** text field, type `solid.epeGp`.

Plastic Strain History

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 History` in the **Label** text field.

Global 1

1 In the **Plastic Strain History** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`intop1(solid.epeGp)`	`mm^3`	`Integrated plastic strain`

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

Evaluate stresses at peak load.

Equivalent stress at peak load

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Equivalent stress at peak load` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (sL)** list, choose **1.5**.

Shear stress at peak load

1 In the **Model Builder** window, expand the **Equivalent stress at peak load** node.

2 Right-click **Equivalent stress at peak load** and choose **Duplicate**.

3 In the **Settings** window for **2D Plot Group**, type `Shear stress at peak load` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Shear stress at peak load** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress (Gauss points)>Stress tensor, Gauss point evaluation (spatial frame) - N/m²>solid.sGprz - Stress tensor, Gauss point evaluation, rz component**.

3 In the **Shear stress at peak load** 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.

FATIGUE (FTG)

Stress-Based 1

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

2 In the **Settings** window for **Stress-Based**, locate the **Domain Selection** section.

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

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

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

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

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 **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Select Domains 3 and 5 only.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Hydrostatic stress sensitivity coefficient	a_DangVan	`0.19`	1	Dang Van
Limit factor	b_DangVan	`282[MPa]`	Pa	Dang Van

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domains 1, 2, and 4 only.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Hydrostatic stress sensitivity coefficient	a_DangVan	`0.23`	1	Dang Van
Limit factor	b_DangVan	`248[MPa]`	Pa	Dang Van

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 From the **Parameter value (sL)** list, choose **From list**.

6 In the **Parameter value (sL)** list, choose **0.6**, **0.65**, **0.7**, **0.75**, **0.8**, **0.85**, **0.9**, **0.95**, **1**, **1.05**, **1.1**, **1.15**, **1.2**, **1.25**, **1.3**, **1.35**, **1.4**, **1.45**, **1.5**, **1.55**, and **1.6**.

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

RESULTS

Fatigue Usage Factor (ftg)

Two new plot groups are added to plot the fatigue usage factor both in 2D (Figure 9) and in 3D revolution.