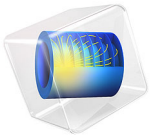

Accelerated Life Testing

Introduction

Fatigue testing of nonlinear materials with a creep mechanism is a time consuming process. In accelerated life testing, the experiment time is greatly reduced by subjecting the material to testing conditions more severe than the operating conditions. In this model an aggressive thermal load cycle is simulated, and its effect on the fatigue life of a solder joint is examined.

This example demonstrates how to evaluate fatigue driven by a specific strain or energy quantity, and to this end, a simple schematic representation of an electronic component is used. Moreover, a single cycle is simulated, as opposed to several cycles that would normally be required to obtain a steady state cycle when working with nonlinear materials. This simplification is motivated by the fact that the purpose of this model is to show how to evaluate fatigue based on a user defined variable. The required strain and energy variables are defined explicitly via ordinary differential equations and calculated during the simulation of the thermal load cycle.

Model Definition

Microelectronic components consist of several materials. When subjected to temperature changes, differences in coefficients of thermal expansion introduce a stress concentration caused by the discontinuous deformation. With repeated heating and cooling, the stress state changes back and forth until the component finally fails because of fatigue.

This example uses a schematic geometry which captures the structural function of the main components rather than an accurate geometry of the microelectronic component. A symmetric model is used and half of its geometry is shown in Figure 1. The total size of the resistor and of the printed circuit board (PCB) is 4×0.5 mm^2. The size of the solder is 0.5×0.25 mm^2. It is assumed that the structure experiences plane strain conditions.

The resistor is made of alumina, the printed circuit board is a fiberglass based laminate and the solder joint is an SnAgCu alloy that responds both linearly and nonlinearly to an applied load. The elastic and the thermal properties of the materials are given in Table 1.

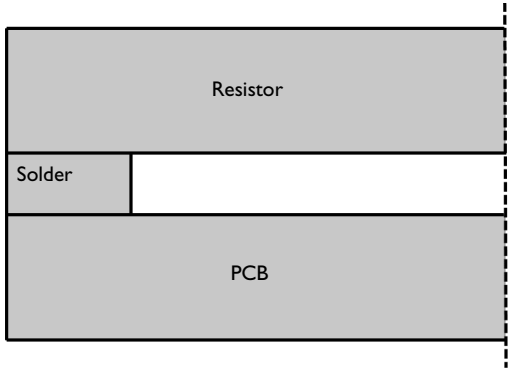

Figure 1: Geometry of the electronic component. The dashed line indicates symmetry.

TABLE 1: ELASTIC AND THERMAL PROPERTIES OF MATERIALS.

Material	Young's modulus (GPa)	Poisson's ratio	Coefficient of thermal expansion (ppm/°C)
Fiberglass	22	0.4	21
SnAgCu	50	0.4	21
Alumina	300	0.22	8

The solder material nonlinearity is creep, which is a material behavior where deformation changes over the time, under constant stress. Generally, materials creep at different rates in three distinct phases: primary creep, secondary creep and tertiary creep; see Figure 2.

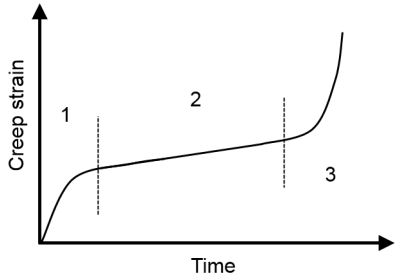

Figure 2: Creep strain development at constant stress. Primary creep denoted with 1, secondary with 2 and tertiary with 3.

The secondary creep is also called the steady state creep since the strain rate at constant stress is constant. In the representation of the solder material, the primary and tertiary

creep regimes are disregarded, and the steady state creep is represented by a double Norton law according to

$$\dot{\varepsilon}_c = A_{\mathrm{I}}\left(\frac{\sigma_e}{\sigma_n}\right)^{n_{\mathrm{I}}} + A_{\mathrm{II}}\left(\frac{\sigma_e}{\sigma_n}\right)^{n_{\mathrm{II}}}$$

where $\dot{\varepsilon}_c$ is the creep rate, σ_e is the equivalent stress and A_{I}, n_{I}, σ_n, A_{II}, and n_{II} are the material constants given by Table 2. The first term represents a creep mechanism observed at low stresses while the second term describes the dominating creep behavior at high stresses, see Figure 3.

TABLE 2: CREEP MATERIAL CONSTANTS.

Material constant	Value
A_{I}	$8.03 \cdot 10^{-12}$ 1/s
n_{I}	3
σ_n	1 MPa
A_{II}	$1.96 \cdot 10^{-23}$ 1/s
n_{II}	12

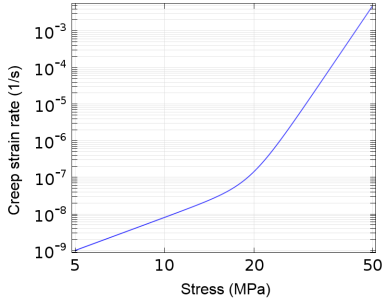

Figure 3: Creep model for the solder SnAgCu solder material.

Two fatigue models are evaluated. In the first, the lifetime is based on a Coffin–Manson type model. The development of the shear strain controls the fatigue according to

$$\Delta\gamma_{\mathrm{II}} = 0.587 \cdot (2N)^{-0.61}$$

where $\Delta\gamma_{\mathrm{II}}$ is the range of the creep shear strain of the second creep mechanism experienced in a load cycle and N is the number of cycles to fatigue.

The second fatigue evaluation depends on the energy dissipation and follows a Morrow type criterion according expressed as

$$\Delta W_{\mathrm{II}} = 74 \cdot 10^{6} \cdot (2N)^{-0.79}$$

where ΔW_{II} is the dissipated creep energy of the second creep mechanism during a load cycle.

Results and Discussion

First the user defined creep strains and energies are verified. The creep strain calculated in the structural analysis is a summation of creep contributions from different mechanisms and thus

$$\varepsilon_c = \varepsilon_c^{\mathrm{I}} + \varepsilon_c^{\mathrm{II}}$$

where $\varepsilon_c^{\mathrm{I}}$ is the creep strain of the first mechanism and $\varepsilon_c^{\mathrm{II}}$ is the creep strain contribution from the second mechanism. The individual contributions of each creep mechanism are calculated in two separate ODE interfaces. A composition of creep results is shown in Figure 4.

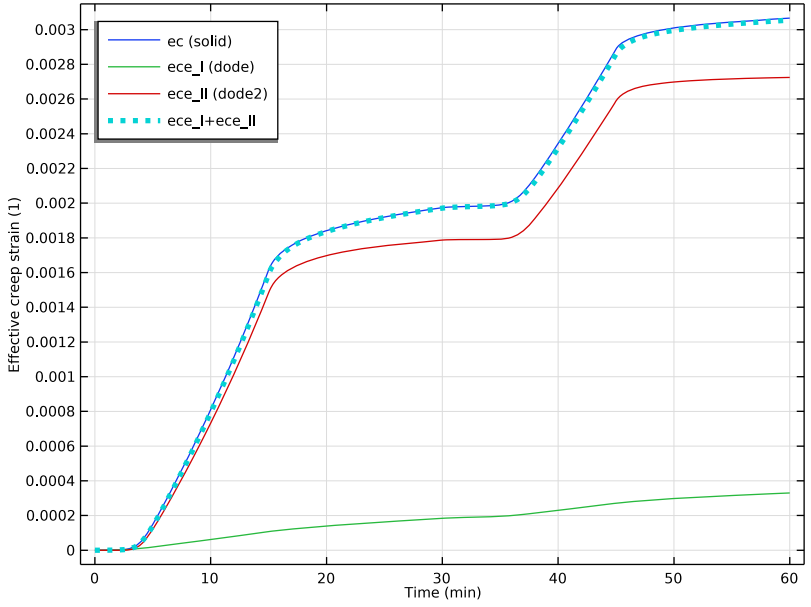

Figure 4: Creep strain history. The equivalent creep strain is shown.

From the figure it is clear that the creep strain calculated using the Nonlinear Structural Materials Module equals the sum of the creep strains calculated with the user defined method.

The creep energy dissipation can also be calculated using an additive decomposition since

$$\delta W_c = \delta\varepsilon_c\sigma_e = \delta\varepsilon_c^{I}\sigma_e + \delta\varepsilon_c^{II}\sigma_e$$

where δ denotes an increment, and W_c denotes the energy dissipation density. The results of the energy comparison in Figure 5 shows a perfect agreement between values calculated by the built-in materials and the user defined material.

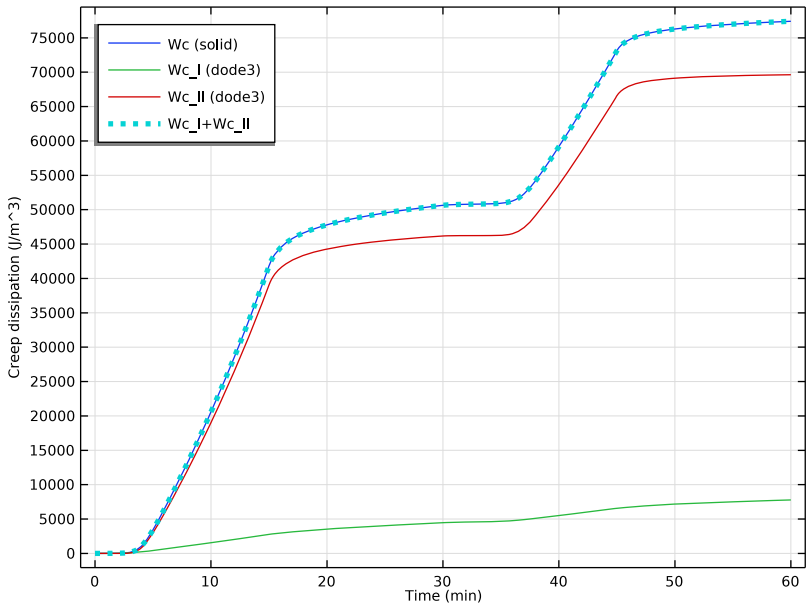

Figure 5: The energy dissipation history.

The fatigue lives predicted by the Coffin–Manson type model and the Morrow type model are shown in Figure 6 and Figure 7, respectively. The predicted life differs by approximately a factor of two, which is not uncommon when using fatigue models of different types, with some uncertainty in the model parameters. Both models indicate that the upper left corner is the critical point in the assembly.

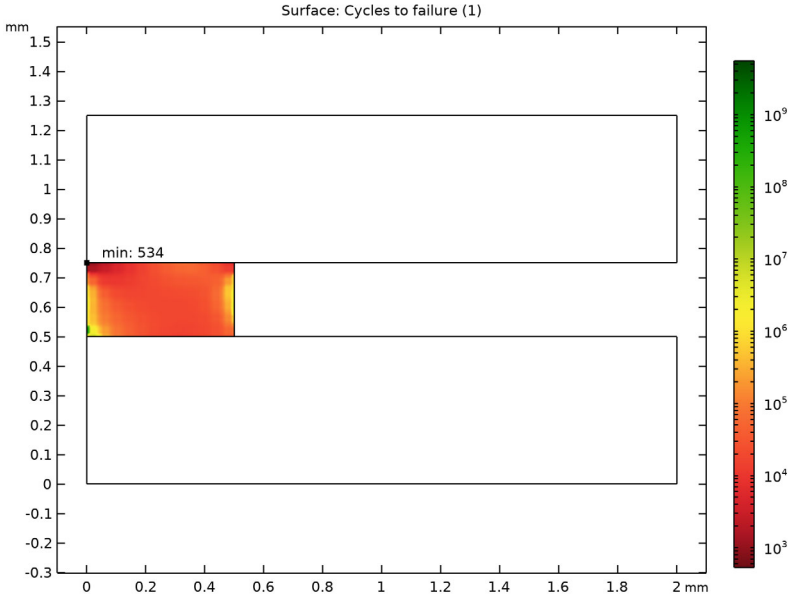

Figure 6: Fatigue life based on the shear strain.

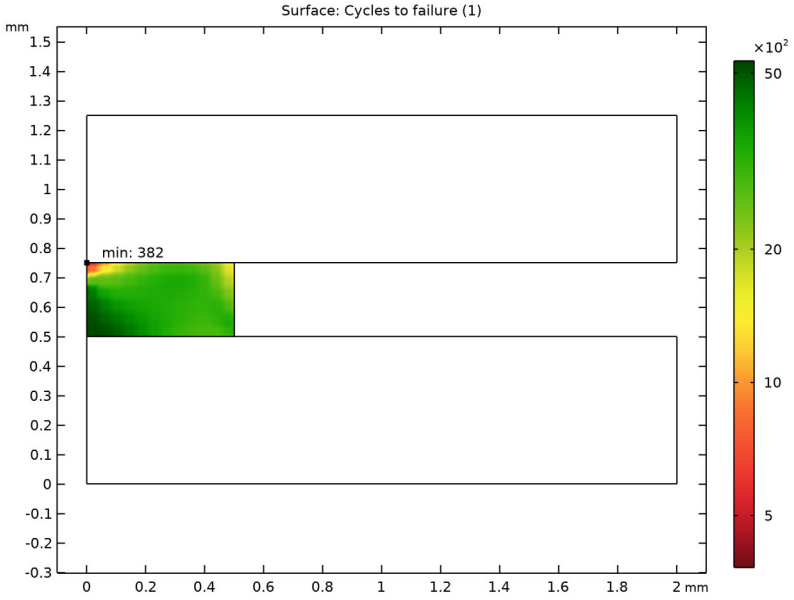

Figure 7: Fatigue life based on the dissipated energy.

Notes About the COMSOL Implementation

Fatigue can be evaluated using many different models. Several models share the same mathematical form. This is the case for the Coffin–Manson based and Morrow based models where the only difference is the controlling strain or energy measure. In rocks and rubbers it is commonly the elastic strain energy or the total strain energy that controls fatigue while in materials subjected to cyclic creep, the dissipated energy controls the fatigue. The Coffin–Manson model that initially related plastic strain to fatigue life has been modified by several researchers that propose a variety of different strain measures as the fatigue controlling mechanism.

In COMSOL Multiphysics, the Morrow and the Coffin–Manson relations

$$\Delta W = W_f' \cdot N^m$$

$$\frac{\Delta \varepsilon}{2} = \varepsilon_f' \cdot N^c$$

can be modified so that the fatigue evaluation is based on a user defined energy density, W, or strain, ε. If the variable controlling fatigue is already defined in a physics interface, it can

be directly specified in the fatigue model node. The variable must be specified together with the corresponding physics interface tag (for example, `solid.Ws` when using the elastic strain energy density). When the variable is not defined in a physics interface, it can be calculated using an ODE Interface. This procedure is demonstrated in this model, where the fatigue is dependent on a specific creep strain and a specific dissipated energy density.

The basic form for an ODE is

$$e_a \frac{\partial^2 \mathbf{u}}{\partial t^2} + d_a \frac{\partial \mathbf{u}}{\partial t} = f \tag{1}$$

where $\mathbf{u}$ is the field variable, t is the time, e_a denotes the mass coefficient matrix, d_a denotes the damping coefficient matrix and f is the source term. The development of creep following the Norton's law is defined with

$$\begin{aligned} \frac{d\varepsilon_c^{ij}}{dt} &= A\left(\frac{\sigma_e}{\sigma_{ref}}\right)^n \cdot \frac{3}{2} \cdot \frac{s^{ij}}{\sigma_e} \\ \frac{d\varepsilon_c}{dt} &= \sqrt{\frac{2}{3} \cdot \frac{d\varepsilon_c^{ij}}{dt} \frac{d\varepsilon_c^{ij}}{dt}} \end{aligned} \tag{2}$$

where ij denotes a specific component, σ_e is the effective von Mises stress, s^{ij} is the deviatoric stress, and A, σ_{ref} and n are material constants. Since the problem is a 2D plane stress analysis, only strains in x, y, z, and xy directions can develop. This gives four dependent variables: `ecx`, `ecy`, `ecz`, and `ecxy`. By comparing the first relation of Equation 2 with Equation 1 it can be identified that there is no contribution from the mass matrix, the damping matrix is an identity matrix and the source term equals the right-hand side of the first relation in Equation 2. This relation contains both the deviatoric stress tensor and the equivalent stress. Since both are already defined in the Solid Mechanics interface, it seems that they can be used in the definition of the source term. COMSOL Multiphysics requires, however, that the derivatives of all user-defined variables can be evaluated at all calculation steps of the analysis. At zero stress the numerical derivative of the equivalent stress

$$\sigma_e = \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_z^2 + \sigma_x\sigma_y + \sigma_x\sigma_z + \sigma_y\sigma_z + 3\tau_{xy}^2}$$

tries to evaluate a negative power of zero. A remedy to this challenge is to provide your own definition of the equivalent stress with a small addition of stress.

$$\sigma_e^{\text{user}} = \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_z^2 + \sigma_x\sigma_y + \sigma_x\sigma_z + \sigma_y\sigma_z + 3\tau_{xy}^2 + 1}$$

The nonzero stress ensures a finite derivative at stress free conditions. In the example a stress addition of 1Pa was chosen. This gives a negligible contribution to the equivalent stress since the resulting stresses have the order of magnitude of MPa.

In addition to the dependent variables used for the creep strain components, one additional dependent variable, `ece`, for the equivalent creep strain as defined by the second relation in Equation 2 is needed. The source term for this relation can be defined using the built-in derivative operator in COMSOL Multiphysics. For example a partial derivative of `u` with respect to `x` is simply defined as `d(u,x)`. By putting it all together, the time derivatives of the five dependent variables `ecx`, `ecy`, `ecz`, `ecxy`, and `ece` become:

- `alpha*solid.sdevx`
- `alpha*solid.sdevy`
- `alpha*solid.sdevz`
- `alpha*solid.sdevxy`
- `(2/3*(d(ecx,TIME)^2+d(ecy,TIME)^2+d(ecz,TIME)^2+2*(d(ecxy,TIME)^2))+1e-20)^0.5`

where `alpha=3/2*A/s_mises*(s_mises/s_ref)^n` and `s_mises=sqrt(solid.sx^2+solid.sy^2+solid.sz^2-solid.sx*solid.sy-solid.sy*solid.sz-solid.sz*solid.sx+3*solid.sxy^2+(1e-6[MPa])^2)`.

The time derivative of the dissipated creep energy density is

$$\frac{dW_c}{dt} = \frac{d\varepsilon_c}{dt} \cdot \sigma_e$$

By using the derivative operator, the source term for the dependent variable, `Wc`, is simply `d(ece,TIME)*s_mises`.

The calculation of the strain variables and the energy density must be made in separate ODE Interfaces since the units of both variables differ.

Application Library path: `Fatigue_Module/Strain_Life/accelerated_life_testing`

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

1 In the **Settings** window for **Geometry**, locate the **Units** section.

2 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 **Width** text field, type `2`.

4 In the **Height** text field, type `0.5`.

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 `0.5`.

4 In the **Height** text field, type `0.25`.

5 Locate the **Position** section. In the **y** text field, type `0.5`.

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 `2`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **y** text field, type `0.75`.

Point 1 (pt1)

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

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

3 In the **x** text field, type `0.25`.

4 In the **y** text field, type `0.625`.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

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>Solid Mechanics (solid)**.

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

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

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
A_I	8.03e-12[1/s]	8.03E-12 1/s	Low stress creep rate coefficient
n_I	3	3	Low stress creep rate exponent
A_II	1.96e-23[1/s]	1.96E-23 1/s	High stress creep rate coefficient
n_II	12	12	High stress creep rate exponent
s_ref	1[MPa]	1E6 Pa	Reference stress

Interpolation 1 (int1)

1 In the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `thermLC`.

4 In the table, enter the following settings:

t	f(t)
0	25
15	100

t	f(t)
30	100
45	25
60	25

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

Argument	Unit
t	min

6 In the **Function** table, enter the following settings:

Function	Unit
thermLC	degC

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

3 Click **OK**.

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

5 In the **Settings** window for **Linear Elastic Material**, click to expand the **Energy Dissipation** section.

6 Select the **Calculate dissipated energy** check box.

Thermal Expansion 1

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

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `thermLC(t)`.

Linear Elastic Material 1

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

Creep 1

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

2 Select Domain 2 only.

3 In the **Settings** window for **Creep**, locate the **Creep Model** section.

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

5 From the σ_{ref} list, choose **User defined**. In the associated text field, type `s_ref`.

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

Additional Creep 1

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

2 Select Domain 2 only.

3 In the **Settings** window for **Additional Creep**, locate the **Creep Model** section.

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

5 From the σ_{ref} list, choose **User defined**. In the associated text field, type `s_ref`.

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

Symmetry 1

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

2 Select Boundaries 11 and 12 only.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 8 only.

MATERIALS

PCB

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

2 In the **Settings** window for **Material**, type `PCB` in the **Label** text field.

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	22[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.4	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	21e-6	1/K	Basic

Solder

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

2 In the **Settings** window for **Material**, type `Solder` in the **Label** text field.

3 Select Domain 2 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	50[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.4	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	21e-6	1/K	Basic

Alumina

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

2 In the **Settings** window for **Material**, type `Alumina` in the **Label** text field.

3 Select Domain 3 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	`300[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.22`	1	Young's modulus and Poisson's ratio
Density	rho	`1`	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`8e-6`	1/K	Basic

DEFINITIONS

Variables 1

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

Define help variables for the user defined strains.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`s_mises`	`sqrt(solid.sx^2+solid.sy^2+solid.sz^2-solid.sx*solid.sy-solid.sy*solid.sz-solid.sz*solid.sx+3*solid.sxy^2+3*solid.syz^2+3*solid.sxz^2+(1e-6[MPa])^2)`	N/m²	

Name	Expression	Unit	Description
alpha_I	3/2*A_I/s_mises*(s_mises/s_ref)^n_I	m·s/kg	
alpha_II	3/2*A_II/s_mises*(s_mises/s_ref)^n_II	m·s/kg	

ADD PHYSICS

1 In the **Physics** toolbar, click  **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Mathematics>ODE and DAE Interfaces>Domain ODEs and DAEs (dode)**.

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

5 In the tree, select **Recently Used>Domain ODEs and DAEs (dode)**.

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

7 In the tree, select **Recently Used>Domain ODEs and DAEs (dode)**.

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

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

DOMAIN ODES AND DAES (DODE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Domain ODEs and DAEs (dode)**.

2 Select Domain 2 only.

3 In the **Settings** window for **Domain ODEs and DAEs**, locate the **Units** section.

4 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1/s

5 Click to expand the **Dependent Variables** section. In the **Field name** text field, type ec_I.

6 In the **Number of dependent variables** text field, type 5.

7 In the **Dependent variables** table, enter the following settings:

ecx_I
ecy_I
ecz_I
ecxy_I
ece_I

Set the same Gauss-point integration order as in the built-in Norton material.

8 Click to expand the **Discretization** section. From the **Shape function type** list, choose **Gauss point data**.

9 From the **Element order** list, choose **4**.

Distributed ODE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Domain ODEs and DAEs (dode)** click **Distributed ODE 1**.

2 In the **Settings** window for **Distributed ODE**, locate the **Source Term** section.

3 In the *f* text-field array, type `alpha_I*solid.sdevx` on the first row.

4 In the *f* text-field array, type `alpha_I*solid.sdevy` on the second row.

5 In the *f* text-field array, type `alpha_I*solid.sdevz` on the third row.

6 In the *f* text-field array, type `alpha_I*solid.sdevxy` on the fourth row.

7 In the *f* text-field array, type

```
2/3*(d(ecx_I,TIME)^2+d(ecy_I,TIME)^2+d(ecz_I,TIME)^2+
2*(d(ecxy_I,TIME)^2))+(1e-20))^0.5
```

on the fifth row.

DOMAIN ODES AND DAES 2 (DODE2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Domain ODEs and DAEs 2 (dode2)**.

2 Select Domain 2 only.

3 In the **Settings** window for **Domain ODEs and DAEs**, locate the **Units** section.

4 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1/s

5 Locate the **Dependent Variables** section. In the **Field name** text field, type `ec_II`.

6 In the **Number of dependent variables** text field, type 5.

7 In the **Dependent variables** table, enter the following settings:

ecx_II
ecy_II
ecz_II
ecxy_II
ece_II

8 Locate the **Discretization** section. From the **Shape function type** list, choose **Gauss point data**.

9 From the **Element order** list, choose **4**.

Distributed ODE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Domain ODEs and DAEs 2 (dode2)** click **Distributed ODE 1**.

2 In the **Settings** window for **Distributed ODE**, locate the **Source Term** section.

3 In the *f* text-field array, type `alpha_II*solid.sdevx` on the first row.

4 In the *f* text-field array, type `alpha_II*solid.sdevy` on the second row.

5 In the *f* text-field array, type `alpha_II*solid.sdevz` on the third row.

6 In the *f* text-field array, type `alpha_II*solid.sdevxy` on the fourth row.

7 In the *f* text-field array, type

```
2/3*(d(ecx_II,TIME)^2+d(ecy_II,TIME)^2+d(ecz_II,TIME)^2+
2*(d(ecxy_II,TIME)^2))+(1e-20))^0.5
```

on the fifth row.

DOMAIN ODES AND DAES 3 (DODE3)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Domain ODEs and DAEs 3 (dode3)**.

2 Select Domain 2 only.

3 In the **Settings** window for **Domain ODEs and DAEs**, locate the **Units** section.

4 Click  **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `energydensity` in the text field.

6 Click  **Filter**.

7 In the tree, select **Electromagnetics>Energy density (J/m^3)**.

8 Click **OK**.

9 In the **Settings** window for **Domain ODEs and DAEs**, locate the **Units** section.

10 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	J/(s*m^3)

11 Locate the **Dependent Variables** section. In the **Field name** text field, type Wc.

12 In the **Number of dependent variables** text field, type 2.

13 In the **Dependent variables** table, enter the following settings:

Wc_I
Wc_II

14 Locate the **Discretization** section. From the **Shape function type** list, choose **Gauss point data**.

15 From the **Element order** list, choose **4**.

Distributed ODE 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Domain ODEs and DAEs 3 (dode3)** click **Distributed ODE 1**.

2 In the **Settings** window for **Distributed ODE**, locate the **Source Term** section.

3 In the *f* text-field array, type `d(ece_I,TIME)*s_mises` on the first row.

4 In the *f* text-field array, type `d(ece_II,TIME)*s_mises` on the second row.

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 Domain 2 only.

Distribution 1

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

2 Select Boundary 3 only.

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

4 In the **Number of elements** text field, type 6.

Distribution 2

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

2 Select Boundary 4 only.

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

4 In the **Number of elements** text field, type 12.

Free Triangular 1

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

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

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

4 Click  **Build All**.

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>** **Time Dependent**.

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

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

STUDY 1

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 From the **Time unit** list, choose **min**.

3 In the **Output times** text field, type `range(0,0.5,14.5) range(14.6,0.1,15.4) range(15.5,0.5,29.5) range(29.6,0.1,30.4) range(30.5,0.5,44.5) range(44.6,0.1,45.4) range(45.5,0.5,60)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click  **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Intermediate**.

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

RESULTS

Stress (solid)

Click the **Zoom Extents** button in the **Graphics** toolbar.

Creep strain I (dode)

1 In the **Model Builder** window, under **Results** click **2D Plot Group 2**.

2 In the **Settings** window for **2D Plot Group**, type `Creep strain I (dode)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Creep strain I (dode)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `ece_I`.

Creep strain II (dode2)

1 In the **Model Builder** window, expand the **Results>2D Plot Group 3** node, then click **2D Plot Group 3**.

2 In the **Settings** window for **2D Plot Group**, type `Creep strain II (dode2)` in the **Label** text field.

Surface 1

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

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

3 In the **Expression** text field, type `ece_II`.

Dissipated energy (dode3)

1 In the **Model Builder** window, under **Results** click **2D Plot Group 4**.

2 In the **Settings** window for **2D Plot Group**, type `Dissipated energy (dode3)` in the **Label** text field.

Create a plot that shows how strain develops during one cycle.

Creep 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 `Creep strain history` in the **Label** text field.

Point Graph 1

1 Right-click **Creep strain history** and choose **Point Graph**.

2 Select Point 5 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.eclGp11`.

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
ec_x

Point Graph 2

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

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

3 In the **Expression** text field, type `solid.eclGp22`.

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

Legends
ec_y

Point Graph 3

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

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

3 In the **Expression** text field, type `solid.eclGp33`.

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

Legends
ec_z

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

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

3 In the **Expression** text field, type `solid.eclGp12`.

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

Legends
ec_xy

Creep strain history

1 In the **Model Builder** window, click **Creep strain history**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Creep strain (1)`.

Verify that strains and energies are correctly calculated in the analysis.

1D Plot Group 6

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

Point Graph 1

1 Right-click **1D Plot Group 6** and choose **Point Graph**.

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

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

4 Select Point 5 only.

5 Locate 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
ec (solid)

Point Graph 2

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

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

3 In the **Expression** text field, type `ece_I`.

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

Legends
ece_I (dode)

Point Graph 3

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

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

3 In the **Expression** text field, type `ece_II`.

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

Legends
`ece_II (dode2)`

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

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

3 In the **Expression** text field, type `ece_I+ece_II`.

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

Legends
`ece_I+ece_II`

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

6 From the **Width** list, choose **4**.

Effective creep history

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Effective creep history` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Effective creep strain (1)`.

Creep dissipation history

1 Right-click **Effective creep history** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Creep dissipation history` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Creep dissipation history** node, then click **Point Graph 1**.

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

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

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

Legends
Wc (solid)

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 In the **Expression** text field, type `Wc_I`.

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

Legends
Wc_I (dode3)

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 In the **Expression** text field, type `Wc_II`.

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

Legends
Wc_II (dode3)

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

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

3 In the **Expression** text field, type `Wc_I+Wc_II`.

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

Legends
Wc_I+Wc_II

Creep dissipation history

1 In the **Model Builder** window, click **Creep dissipation history**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Creep dissipation (J/m^3)`.

4 In the **Creep dissipation history** 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)

Strain-Life 1

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

2 Select Domain 2 only.

3 In the **Settings** window for **Strain-Life**, locate the **Fatigue Model Selection** section.

4 From the **Criterion** list, choose **Coffin-Manson**.

5 From the **Strain type** list, choose **User defined**.

6 In the ε_i text field, type `2*ecxy_II`.

7 Locate the **Fatigue Model Parameters** section. From the ε_f' list, choose **User defined**. In the associated text field, type `0.587`.

8 From the c list, choose **User defined**. In the associated text field, type `-0.61`.

ADD PHYSICS

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

2 In the tree, select **Recently Used>Fatigue (ftg)**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

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

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

FATIGUE 2 (FTG2)

Select Domain 2 only.

Energy-Based 1

1 Right-click **Component 1 (comp1)>Fatigue 2 (ftg2)** and choose the domain evaluation **Energy-Based**.

2 Select Domain 2 only.

3 In the **Settings** window for **Energy-Based**, locate the **Fatigue Model Selection** section.

4 From the **Energy type** list, choose **User defined**.

5 In the W_d text field, type `Wc_II`.

6 Locate the **Fatigue Model Parameters** section. From the W_f' list, choose **User defined**. In the associated text field, type `74e6`.

7 From the m list, choose **User defined**. In the associated text field, type `-0.79`.

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 boxes for **Solid Mechanics (solid)**, **Domain ODEs and DAEs (dode)**, **Domain ODEs and DAEs 2 (dode2)**, and **Domain ODEs and DAEs 3 (dode3)**.

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, Time Dependent**.

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

The plots shown in Figure 6 and Figure 7 are generated by default.

RESULTS

Cycles to Failure (ftg)

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 **Coloring and Style** section.

3 From the **Anchor point** list, choose **Lower left**.

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

Cycles to Failure (ftg2)

In the **Model Builder** window, expand the **Results>Cycles to Failure (ftg2)** node.

Marker 1

1 In the **Model Builder** window, expand the **Results>Cycles to Failure (ftg2)>Surface 1** node, then click **Marker 1**.

2 In the **Settings** window for **Marker**, locate the **Coloring and Style** section.

3 From the **Anchor point** list, choose **Lower left**.

4 In the **Cycles to Failure (ftg2)** toolbar, click **Plot**.