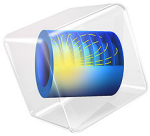


Thermal Fatigue of a Surface Mount Resistor

Introduction

A surface mount resistor is subjected to thermal cycling. The difference in the thermal expansion between the materials introduces thermal stresses in the structure. The solder, connecting the resistor to the printed circuit board, is seen as the weakest link in the assembly. Because the operating temperature is high when compared to the melting point of the solder, creep deformation occurs. In order to assure the structural integrity of the component a fatigue analysis is performed where the life prediction from two different fatigue models is compared.

Model Definition

A resistor is fastened on a printed circuit board (PCB) with SnAgCu solder. The solder is connected to the printed circuit board through two copper pads and to the resistor through a NiCr conductor. In reality there are additional thin films around the resistor but they are disregarded in current analysis. A sketch of the surface mount assembly is shown in Figure 1.

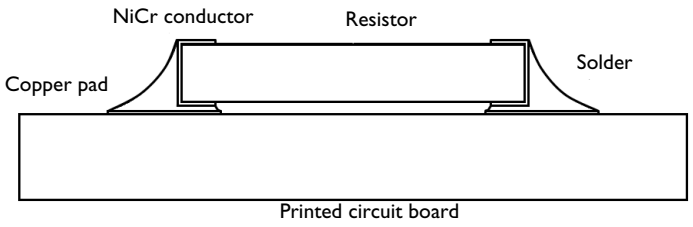


Figure 1: Schematic description of the surface mount resistor.

The resistor is made out of alumina and has dimensions 3.2 mm x 0.55 mm. It is covered on both edges with a 0.025 mm layer of NiCr conductor. The thin layer continues 0.325 mm along the lower and the upper side of the resistor. The printed circuit board is large in comparison with the resistor and is here modeled 0.8 mm thick. It has two copper pads on the top side that are 0.025 mm thick and 1.05 mm wide. The thickness of the solder fillet between the copper pads and the NiCr conductor is 0.05 mm. The remaining shape of the solder joint varies greatly between each examined solder joint and is here modeled with two representative roundings.

Because the out-of-plane dimensions is 1.55 mm, which is significant in comparison with the size of the resistor, the model is simulated in 2D with plane strain conditions.

The elastic properties of the materials are summarized in Table 1.

TABLE 1: ELASTIC AND THERMAL MATERIAL PROPERTIES.

MATERIAL	YOUNG'S MODULUS (GPA)	POISSON'S RATIO	COEFFICIENT OF THERMAL EXPANSION (PPM/°C)
PCB laminate	22	0.4	21
Copper	141	0.35	17
SnAgCu	50	0.4	21
NiCr	170	0.31	13
Alumina	300	0.22	8

The SnAgCu solder material exhibits creep behavior, which can be modeled by a Garofalo model where creep rate is described with

$$\frac{d\varepsilon_{ij}^{c}}{dT} = 2.62 \cdot 10^{5}\left[\sinh\left(\frac{\sigma_e}{39.1 \cdot 10^{6}}\right)\right]^{6.19} e^{-\left(\frac{5.32 \cdot 10^{4}}{RT}\right)} \cdot \frac{3}{2}\frac{s_{ij}}{\sigma_e} \tag{1}$$

where ε_{ij}^{c} is the creep strain tensor, T is the temperature, σ_e is the equivalent stress, R is the universal gas constant, and s_{ij} is the deviatoric stress tensor.

The thermal load during an operating cycle is prescribed as a temperature which varies between 20 °C and 70 °C. Each temperature change takes 2 minutes and is followed by a 3 minutes dwell. This means that one fatigue cycle requires 10 minutes, see Figure 2.

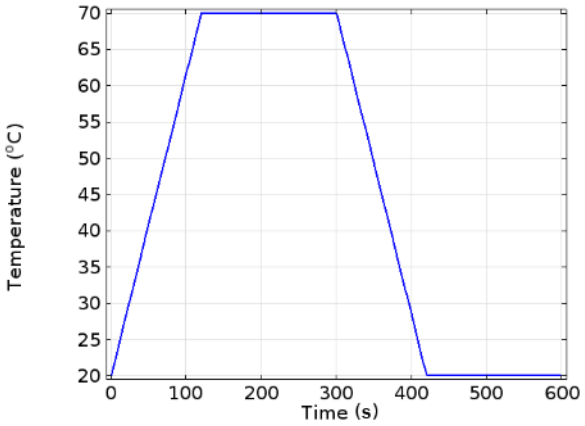


Figure 2: Temperature load.

Since the solder material is nonlinear, see Equation 1, several cycles may need to simulated before a stable cycle is obtained.

Two fatigue models are evaluated. The first is a strain-based Coffin-Manson type model with the equivalent creep strain as the damage controlling mechanism, and the second is an energy-based Morrow type model with the dissipated creep energy as the damage controlling mechanism. The material constants for the Coffin-Manson model are ε_f'=0.281 and c=-0.51. The material constants for the Morrow type model are W_f'=55.0 J/m^3 and m=-0.69.

Results and Discussion

The difference in the elastic and thermal properties introduces thermal stresses in the device. Although they are not very high the solder experiences significant inelastic strains. In Figure 3 accumulated equivalent creep strain after six cycles is shown.

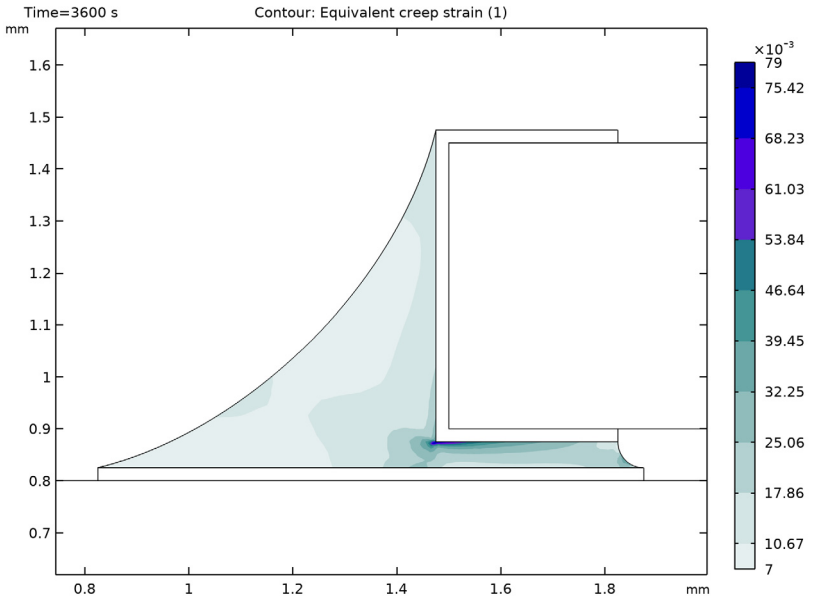


Figure 3: Creep strains in the solder joint.

The highest strains occur in the thin solder layer just below the resistor. It is mainly the shear strain component which contributes to the equivalent creep strain in that layer. The slightly higher values around the edge are also affected by modeling of a sharp corner. With a fillet instead, the strains will be somewhat lower. Nevertheless, the location of highest strain agrees well with the crack path in real applications.

In order to evaluate fatigue, it is important to obtain a stable load cycle. In applications involving solder joints, frequently either inelastic strain or dissipated energy is used to predict fatigue. The change of creep strain during the first six cycles is therefore evaluated in a point just below the resistor slightly shifted to the right form the sharp corner. The position of this point can be debated. It is however located in the area where the largest strains occurs and is therefore seen as the critical point. In Figure 4 the equivalent creep strain and the shear creep strain component are shown.

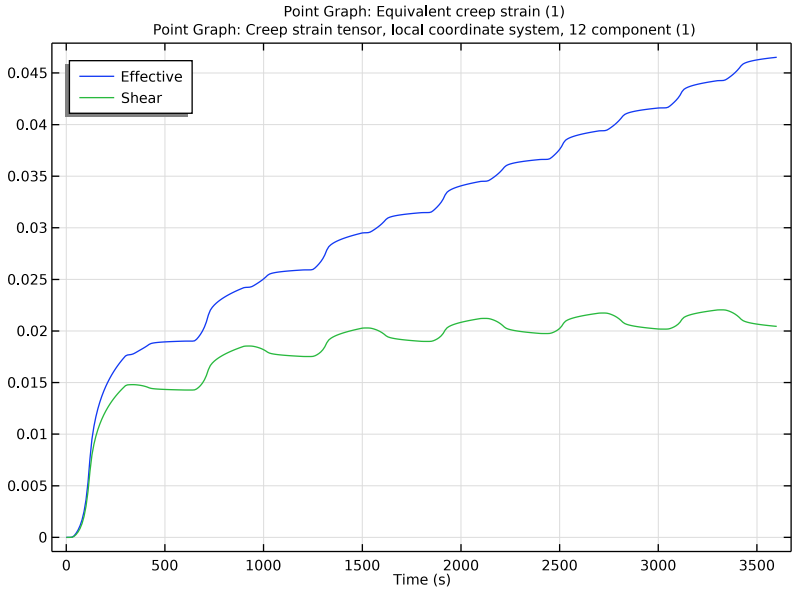


Figure 4: Creep strain development in a critical point below the resistor.

The dissipated energy represents a combined contribution of changes in stresses and strains during a cycle and is in Figure 5 shown with a shear hysteresis. The shear component has been chosen since it gives the dominating contribution to the equivalent creep strain in the critical point.

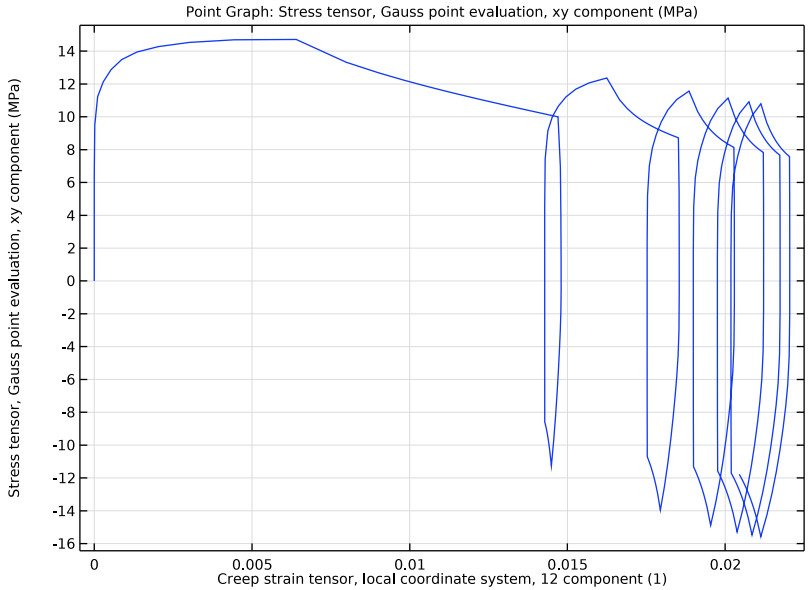


Figure 5: Shear hysteresis evaluated in a critical point just below the resistor.

It is clear from the two last figures that the first cycle is not representative for fatigue analysis, since its response differs significantly from the one experienced in the following cycles. Even after six cycles, the stress-strain loop has not stabilized. The temperature cycling can by extended with additional cycles to evaluate whether the state stabilizes further or not. Under some conditions, it may happen that the hysteresis loop is moving in stress-strain space. In this example additional cycles are not simulated since the difference in the creep strain and the dissipated energy between cycle five and six is small. Assuming that the consecutive cycles follow the trend and deform less as, well as dissipate less energy, the fatigue analysis based on the results of the sixth cycle gives a conservative fatigue prediction.

The fatigue life based on the Coffin-Manson model is shown in Figure 6, and the fatigue life based on the Morrow model is shown in Figure 7.

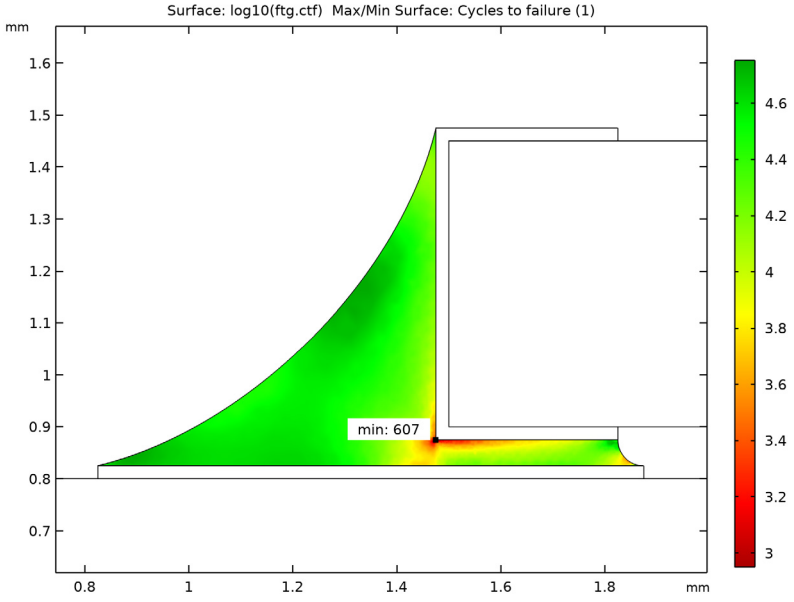


Figure 6: Fatigue life based on the creep strain.

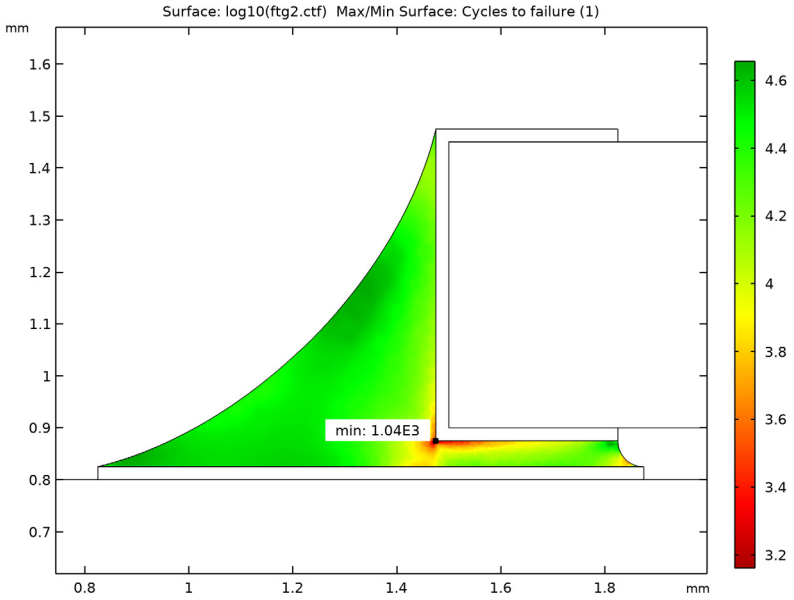


Figure 7: Fatigue life based on the dissipated energy.

Fatigue based on strain gives lifetime of about 600 cycles, while the energy prediction gives 1000 cycles.

Application Library path: `Fatigue_Module/Energy_Based/surface_mount_resistor`

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**.

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>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

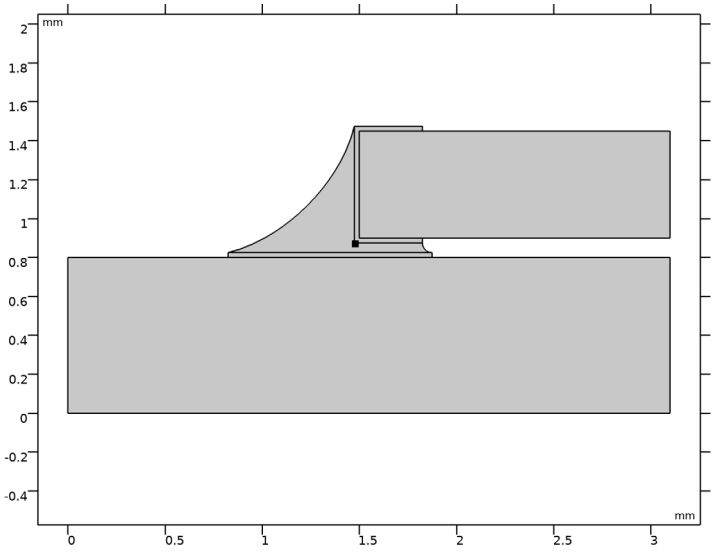
The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Appendix — Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `surface_mount_resistor_geom_sequence.mph`.

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

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



GLOBAL DEFINITIONS

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 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `surface_mount_resistor_thermal_load_cycle.txt`.

6 Click **Import**.

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

8 Locate the **Units** section. In the **Arguments** text field, type `min`.

9 In the **Function** text field, type `degC`.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type `1.55 [mm]`.

4 Locate the **Structural Transient Behavior** section. From the list, choose **Quasistatic**.

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

3 From the **Selection** list, choose **Solder**.

4 Locate the **Model Input** section. From the T list, choose **User defined**. In the associated text field, type `thermLC(t)`.

5 Locate the **Creep Data** section. From the **Material model** list, choose **Garofalo (hyperbolic sine)**.

6 Select the **Include temperature dependency** check box.

7 In the Q text field, type `53200`.

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

Symmetry 1

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

2 Select Boundaries 19 and 20 only.

Roller 1

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

2 Select Boundary 2 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 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Rectangle 1**.

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	Basic
Poisson's ratio	nu	`0.4`	1	Basic
Density	rho	0	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`21e-6`	1/K	Basic

Copper

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Rectangle 3**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	`141[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.35`	1	Basic
Density	rho	0	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`16.6e-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 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Solder**.

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	Basic
Poisson's ratio	nu	`0.4`	1	Basic
Density	rho	0	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`21e-6`	1/K	Basic
Creep rate coefficient	A_gar	`262000`	1/s	Garofalo (hyperbolic sine)
Reference stress	sigRef_gar	`39.1[MPa]`	N/m²	Garofalo (hyperbolic sine)
Garofalo n parameter	n_gar	`6.19`	1	Garofalo (hyperbolic sine)

NiCr

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **NiCr**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	170[GPa]	Pa	Basic
Poisson's ratio	nu	0.31	1	Basic
Density	rho	0	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	13e-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 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Rectangle 2**.

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	Basic
Poisson's ratio	nu	0.22	1	Basic
Density	rho	0	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	8e-6	1/K	Basic

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

Free Triangular 1

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

Distribution 1

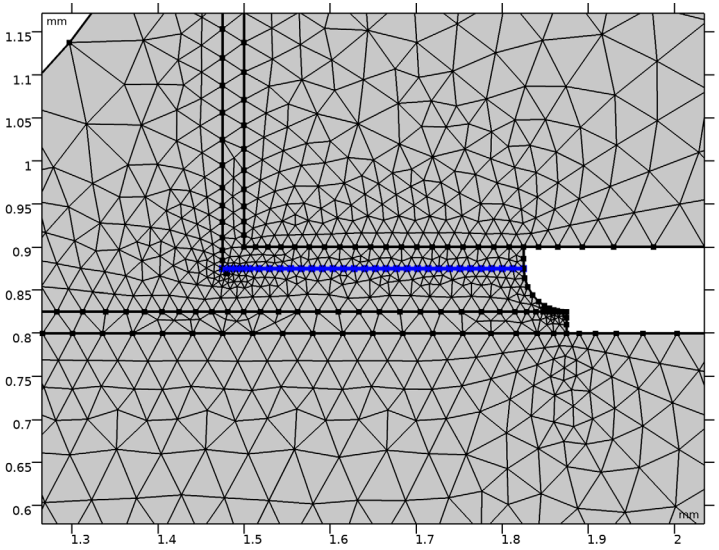
1 Right-click **Free Triangular 1** and choose **Distribution**.

2 Select Boundary 8 only.

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

4 In the **Number of elements** text field, type 30.

5 Click **Build All**.



STUDY 1

Step 1: Time Dependent

Simulate a time history of 6 cycles.

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

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

3 In the **Output times** text field, type `range(0,10,60*60)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-4`.

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

7 In the **Settings** window for **Study**, type `Time History` in the **Label** text field.

Solution 1 (sol1)

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

Force strict time stepping in order to improve the creep results.

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 **Strict**.

5 In the **Model Builder** window, expand the **Time History>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node.

6 Right-click **Time History** and choose **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.

2 Right-click **Deformation** and choose **Enable**.

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

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

5 In the associated text field, type `20`.

Contour 1

Display creep strain history. The shear component gives the largest contribution to the equivalent creep strain.

Creep Strain

1 In the **Model Builder** window, expand the **Equivalent Creep Strain (solid)** node.

2 Right-click **Results** and choose **1D Plot Group**.

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

Point Graph 1

1 Right-click **Creep Strain** and choose **Point Graph**.

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

3 From the **Selection** list, choose **Point 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)>solid.eceGp - Equivalent creep strain**.

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
Effective

Point Graph 2

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

2 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 (Gauss points)>Creep strain tensor, local coordinate system> solid.ecGp12 - Creep strain tensor, local coordinate system, 12 component**.

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

Legends
Shear

Creep Strain

1 In the **Model Builder** window, click **Creep Strain**.

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

3 From the **Position** list, choose **Upper left**.

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

Display stress-strain hysteresis of the shear behavior.

Shear Hysteresis

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

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

Point Graph 1

1 Right-click **Shear Hysteresis** and choose **Point Graph**.

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

3 From the **Selection** list, choose **Point 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress (Gauss points)> Stress tensor, Gauss point evaluation (spatial frame) - N/m²>solid.sGpxy - Stress tensor, Gauss point evaluation, xy component**.

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

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

7 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)> Creep strain tensor, local coordinate system>solid.ecGp12 - Creep strain tensor, local coordinate system, 12 component**.

8 In the **Shear Hysteresis** 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 **Time History**.

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

FATIGUE, COFFIN-MANSON

In the **Settings** window for **Fatigue**, type `Fatigue, Coffin-Manson` in the **Label** text field.

Strain-Life 1

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

2 In the **Settings** window for **Strain-Life**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Solder**.

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

5 Locate the **Fatigue Model Selection** section. From the **Criterion** list, choose **Coffin-Manson**.

6 From the **Strain type** list, choose **Equivalent creep strain**.

ADD PHYSICS

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

2 In the tree, select **Structural Mechanics>Fatigue (ftg)**.

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

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, MORROW

In the **Settings** window for **Fatigue**, type `Fatigue, Morrow` in the **Label** text field.

Energy-Based 1

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

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

3 From the **Selection** list, choose **Solder**.

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

5 Locate the **Fatigue Model Selection** section. From the **Energy type** list, choose **Creep dissipation density**.

MATERIALS

Solder (mat3)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Solder (mat3)** node, then click **Solder (mat3)**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Solid Mechanics>Fatigue Behavior>Energy-Based>Morrow**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Solid Mechanics>Fatigue Behavior>Strain-Based>Coffin-Manson**.

6 Click **Add to Material**.

7 In the **Model Builder** window, under **Component 1 (comp1)>Materials>Solder (mat3)** click **Morrow (fatigueEnergyMorrow)**.

8 In the **Settings** window for **Property Group**, locate the **Output Properties** section.

9 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Fatigue energy coefficient	Wf_Morrow	55e6 [J/m^3]	J/m³	1x1
Fatigue energy exponent	m_Morrow	-0.69	1	1x1

10 In the **Model Builder** window, under **Component 1 (comp1)>Materials>Solder (mat3)** click **Coffin-Manson (fatigueStrainCoffinManson)**.

11 In the **Settings** window for **Property Group**, locate the **Output Properties** section.

12 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Fatigue ductility coefficient	epsilonf_CM	0.218	1	1x1
Fatigue ductility exponent	c_CM	-0.51	1	1x1

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.

FATIGUE

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

2 In the **Settings** window for **Study**, type `Fatigue` in the **Label** text field.

Step 1: Fatigue

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

2 In the **Settings** window for **Fatigue**, locate the **Values of Dependent Variables** section.

3 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

5 From the **Study** list, choose **Time History, Time Dependent**.

6 From the **Time (s)** list, choose **From list**.

7 From the **Time (s)** selection list, select time steps from 3000 s to 3600 s.

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

Two new plot groups are created. They show the cycles to failure for each fatigue model, see Figure 6 and Figure 7.

RESULTS

Cycles to Failure, Coffin-Manson

In the **Settings** window for **2D Plot Group**, type `Cycles to Failure, Coffin-Manson` in the **Label** text field.

Max/Min Surface 1

1 In the **Model Builder** window, expand the **Cycles to Failure, Coffin-Manson** node, then click **Max/Min Surface 1**.

2 In the **Settings** window for **Max/Min Surface**, locate the **Coloring and Style** section.

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

Cycles to Failure, Morrow

1 In the **Model Builder** window, under **Results** click **Cycles to Failure (ftg2)**.

2 In the **Settings** window for **2D Plot Group**, type `Cycles to Failure, Morrow` in the **Label** text field.

Max/Min Surface 1

1 In the **Model Builder** window, expand the **Cycles to Failure, Morrow** node, then click **Max/Min Surface 1**.

2 In the **Settings** window for **Max/Min Surface**, locate the **Coloring and Style** section.

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

Appendix — Geometry Modeling Instructions

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 **Width** text field, type `3.1`.

4 In the **Height** text field, type `0.8`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

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 `1.6`.

4 In the **Height** text field, type `0.55`.

5 Locate the **Position** section. In the **x** text field, type `1.5`.

6 In the **y** text field, type `0.9`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

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 `1.05`.

4 In the **Height** text field, type `0.025`.

5 Locate the **Position** section. In the **x** text field, type `0.825`.

6 In the **y** text field, type `0.8`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

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 `0.35`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. In the **x** text field, type `1.475`.

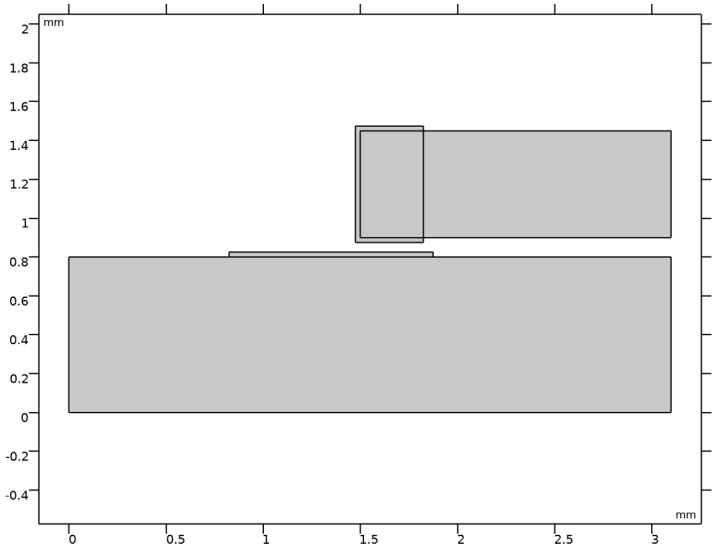
6 In the **y** text field, type `0.875`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog box, type `NiCr` in the **Name** text field.

9 Click **OK**.

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



Quadratic Bézier 1 (qb1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **x** to `0.825`.

4 In row **1**, set **y** to `0.825`.

5 In row **2**, set **x** to `1.35`.

6 In row **2**, set **y** to `0.95`.

7 In row **3**, set **x** to `1.475`.

8 In row **3**, set **y** to `1.475`.

9 Click **Build Selected**.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

x (mm)	y (mm)
1.475	1.475
1.475	0.875
1.825	0.875

5 Click **Build Selected**.

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **x** text field, type 1.875.

4 In the **y** text field, type 0.875.

5 Locate the **Radius** section. In the **Radius** text field, type 0.05.

6 Locate the **Angles** section. In the **Start angle** text field, type 180.

7 In the **End angle** text field, type 270.

8 Click **Build Selected**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **ca1**, select Point 2 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Find the **End vertex** subsection. Select the **Activate Selection** toggle button.

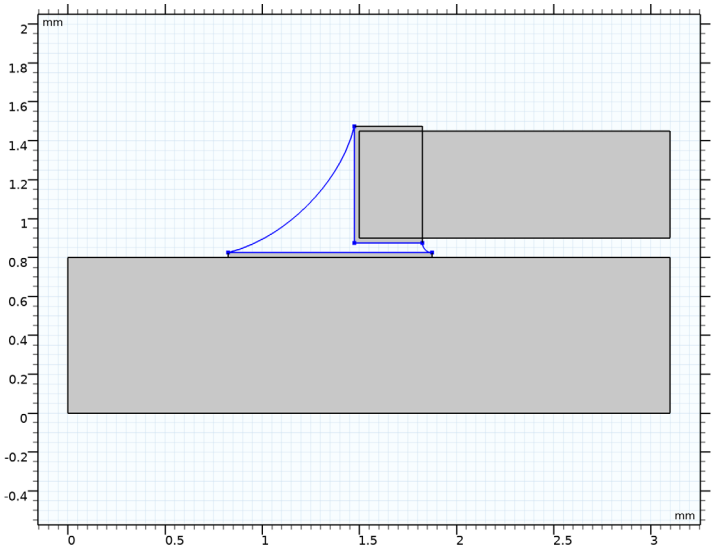
5 On the object **qb1**, select Point 1 only.

6 Click **Build Selected**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the objects **ca1**, **ls1**, **pol1**, and **qb1** only.



3 In the **Settings** window for **Convert to Solid**, locate the **Selections of Resulting Entities** section.

4 Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Solder` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

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 `1.48`.

4 In the **y** text field, type `0.87`.

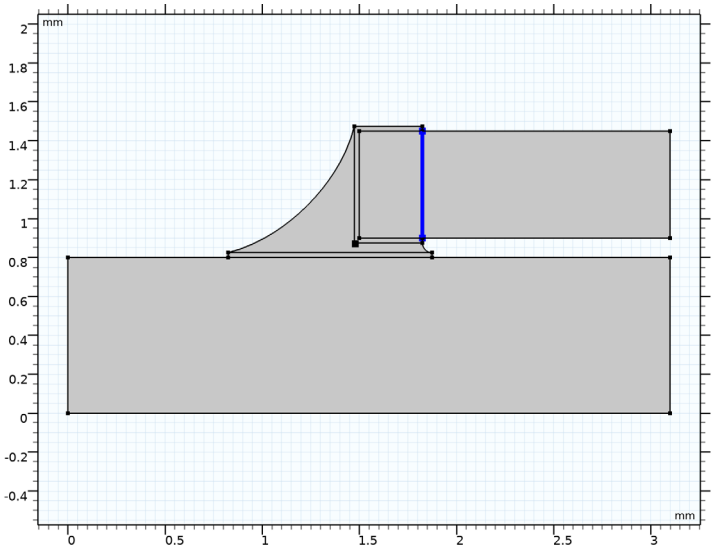
5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Boundary 14 only.



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

Form Union (fin)

Click **Build All**.