Created in COMSOL Multiphysics 6.0

Energy-Based Thermal Fatigue Prediction in a Ball Grid Array

Introduction

In a cooling system, a microelectronic component has been identified as the critical link. Since the power is repeatedly switched on and off, the component is subjected to thermal cycling. As a result, a crack grows through a solder joint and disconnects the chip from the printed circuit board, causing a malfunction. In the simulation, the solder lifetime in two ball grid assemblies is predicted. The prediction is based on the Darveaux energy-based model. The fatigue model evaluates damage based on an averaged energy dissipation density in a thin layer, where a crack is expected to grow.

This example is based on a model from the Nonlinear Structural Materials Module — Viscoplastic Solder Joints. Since the model contains several solder joints that are modeled with a viscoplastic material, many degrees of freedom are required in order to simulate the correct creep behavior in all elements. From the fatigue point of view, only the critical part of the model is of interest. In order to capture it, the concept of submodeling is used. This technique consists of two steps. In the first one, the full model is analyzed with a coarse mesh in order to capture the general trends and to identify the critical part of the model. In the second step a fine submodel containing the critical part is made and the study is resolved. The global effects from the full model are transferred to the submodel via appropriate boundary conditions.

Model Definition

The microelectronic component consists of a flat printed circuit board that is covered with a thin copper layer. Two microprocessors are connected to the copper layer with a ball grid array of 60Sn-40Pb solder joints; see Figure 1.

Both microprocessors generate power when they are switched on and when they are in stand-by. This generates heat throughout its operational lifetime. The power cycle is at its maximum, $5 \cdot 10^7\ \mathrm{W/m^3}$, during 4 h and at its minimum $1 \cdot 10^7\ \mathrm{W/m^3}$, during 2 h. The switch between high and low power is not instantaneous, but takes few minutes.

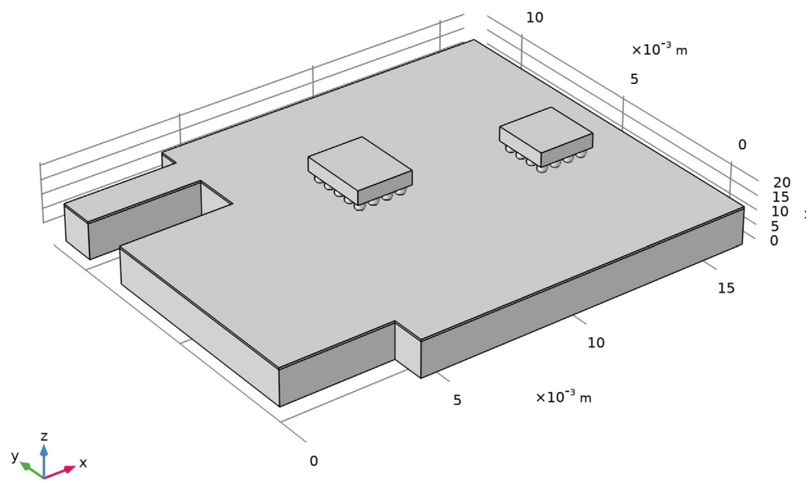

Figure 1: Geometry of the microelectronic component.

The elastic and thermal properties of the material can be taken from the built-in material library in COMSOL Multiphysics. The heat transfer coefficient for all free surfaces can be approximated with $10\ \mathrm{W/(m^2{\cdot}K)}$. The nonlinear behavior of the solder material follows the Anand material model with parameters summarized in Table 1.

TABLE 1: CONSTANTS OF THE ANAND MODEL.

PROPERTY	VALUE	DESCRIPTION
A	$1.49 \cdot 10^7$ 1/s	Pre-exponential factor
Q	90046 J/mol	Activation energy/Boltzmann constant
ξ	11	Stress multiplier
m	0.303	Strain rate sensitivity of stress
s_{sat}	80.42 MPa	Coefficient for deformation resistance saturation
s_{init}	56.33 MPa	Initial value of deformation resistance
h_0	2640.75 MPa	Hardening coefficient
a	1.34	Strain rate sensitivity of hardening
n	0.0231	Sensitivity for deformation resistance

The Darveaux fatigue model is representative for life prediction of the solder material. The model combines life contributions from crack initiation and crack propagation using the expression

$$N = K_1\left(\frac{\Delta W_{\mathrm{ave}}}{W_{\mathrm{ref}}}\right)^{k_2} + \frac{a}{K_3\left(\frac{\Delta W_{\mathrm{ave}}}{W_{\mathrm{ref}}}\right)^{k_4}}$$

where N is the fatigue life given in number of cycles, ΔW_{ave} is the averaged dissipated energy density in a fatigue cycle, a is the distance the crack needs to propagate for the failure to occur, and K_1, k_2, K_3, k_4, and W_{ref} are material constants. The numerical values of the material constants are given in Table 2.

TABLE 2: FATIGUE MATERIAL PARAMETERS.

PROPERTY	VALUE	DESCRIPTION
K_1	13173	Crack initiation energy coefficient
k_2	-1.45	Crack initiation energy exponent
K_3	$3.92 \cdot 10^{-7}$ in	Crack propagation energy coefficient
k_4	1.12	Crack propagation energy exponent
W_{ref}	1 psi^3	Reference energy density

The distance a is in this analysis be taken to be $2.6457 \cdot 10^{-4}$ in. This is based on the assumption that the problem is not symmetric and a crack is expected to start on one side of the joint only and not all around the joint at the same time.

The concept of submodeling is utilized in this example. This technique requires that first an analysis of the full model is performed in order to capture general trends, followed by an analysis of a submodel that is studied in detail. The following steps are done:

1 A coupled structural and thermal analysis is performed during four load cycles on the full model. Since the model contains several solder joints, a coarse mesh is used. All joints are meshed in the same way in order to minimize numerical discrepancies that are mesh dependent.

2 A fatigue prediction is made on the fourth cycle. The energy dissipation volume average and corresponding life is evaluated for each individual solder joint. The critical joint is identified.

3 A submodel of the critical joint with a fine mesh is created. The displacments from step 1 are prescribed as **Prescribed Displacement** on the boundaries where the submodel is cut out of the full model. Similarly, the temperature from step 1 is prescribed as a

Temperature in the heat transfer analysis of the submodel. This is done for all time steps of the four simulated cycles.

4 A fatigue analysis is performed on the critical solder joint in the submodel. A life prediction is made based on the energy dissipation in a 50 μm thick layer. Two layers are evaluated. One that is in connection with the copper side and one that is in connection with the microchip side.

The submodel is shown in Figure 2. The purple color denotes the boundaries where the results from the structural analysis of the global model are prescribed. The solder joint is divided in three domains: a central one and two domains close to the interface to the other materials.

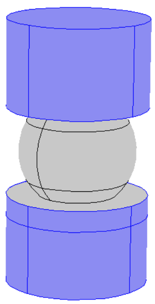

Figure 2: Submodel containing the critical joint.

Results and Discussion

The fatigue life prediction for all joints is shown in Figure 3. For both microchips, the critical joints are located in the corners of each ball grid array. This is expected, since those joints experience highest strains due to the differences in thermal properties. The solder joints of the microchip with the larger ball grid array shows a shorter life than the joints of the other microchip. The life prediction for all four corner joints is about the same, $10^{4.8}$ cycles.

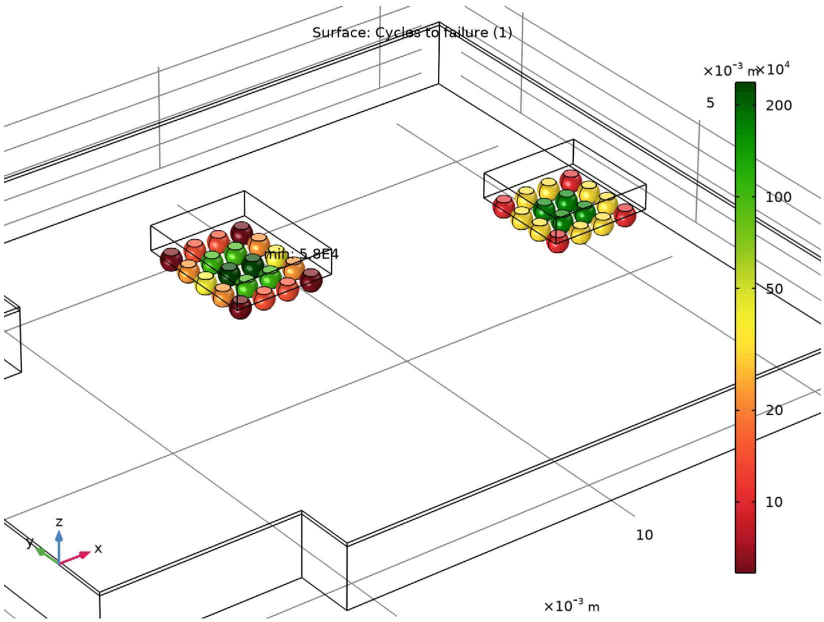

Figure 3: Fatigue life prediction for all joints based on the global model.

One of the four critical corner joints is reanalyzed in the submodel. In order to verify that the temperature is correctly prescribed in the submodel a comparison of the temperature history in a point on the upper sider of the joint is shown in Figure 4. The results of both models are in prefect match. Note that much fewer data points are stored for the submodel.

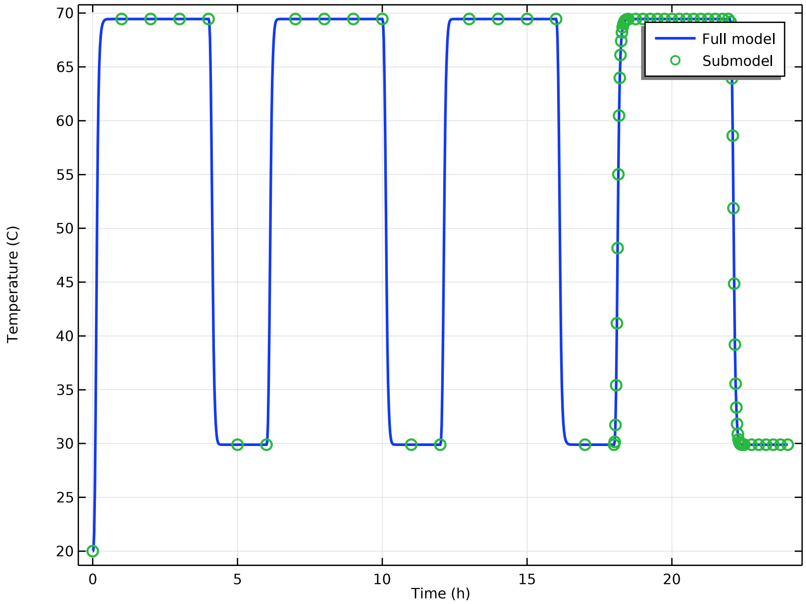

Figure 4: Temperature history for both the full model and the submodel.

A comparison of the dissipated energy in a point on the upper side of the joint is shown in Figure 5. The results differs between the two models. This difference is caused by the difference in the mesh. The finer mesh in the submodel is better than the coarser mesh of the full model in capturing the strain gradient close to the upper side of the solder joint. Note that much fewer data points are stored in the submodel for the first three cycles, hence, the presented results are not a smooth. This does not affect the accuracy of the solution.

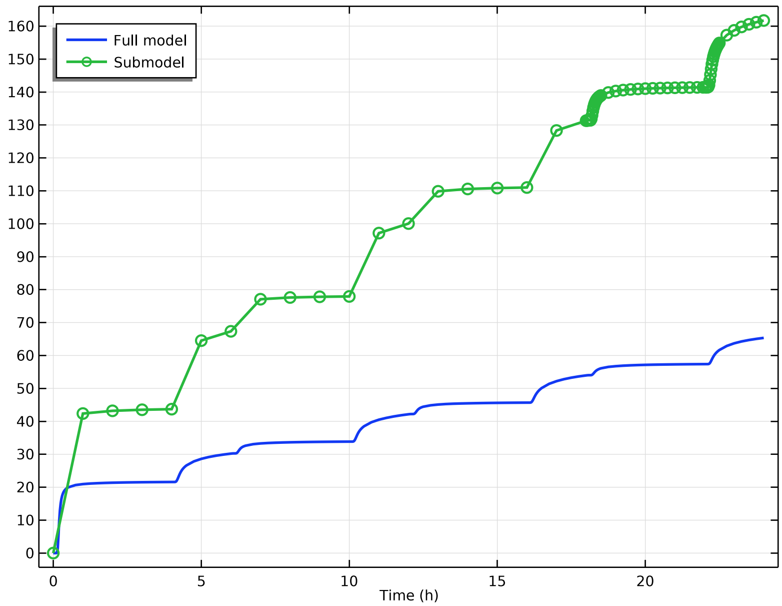

Figure 5: Comparison of the dissipated creep energy in both models.

The results of the fatigue analysis of the critical joint in the submodel are shown in Figure 6 and Figure 7. In the first figure, the fatigue life is based on the energy dissipation volume average evaluated over the whole joint, while in the second figure the volume average is performed over separate domains. The fatigue life predicted by different models is summarized in Table 3.

TABLE 3: FATIGUE LIFE BASED ON DIFFERENT MODELING TECHNIQUES.

EVALUATION METHOD	**FATIGUE LIFE (CYCLES)**	**SOURCE**
Entire joint in the full model	$5.8 \cdot 10^4$	Figure 3
Entire joint in the submodel	$8.1 \cdot 10^4$	Figure 6
Thin layer in the submodel	$9.1 \cdot 10^3$	Figure 7

The difference between the fatigue life prediction of the full model and of the submodel is small. It is however observed in real applications that a crack grows through a solder joint close to the interface and therefore a thin layer is required. If a thin layer were to be created in all joints of the full model, the simulation would require large computational resources. In the current example, the full model consists of about 170,000 DOFs while the submodel consists of 40,000 DOFs.

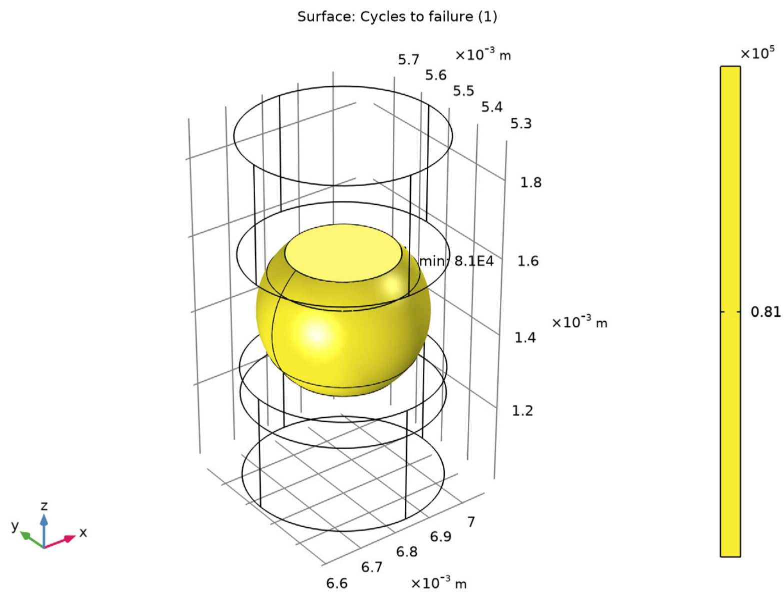

Figure 6: Fatigue prediction based on the volume average of the entire joint.

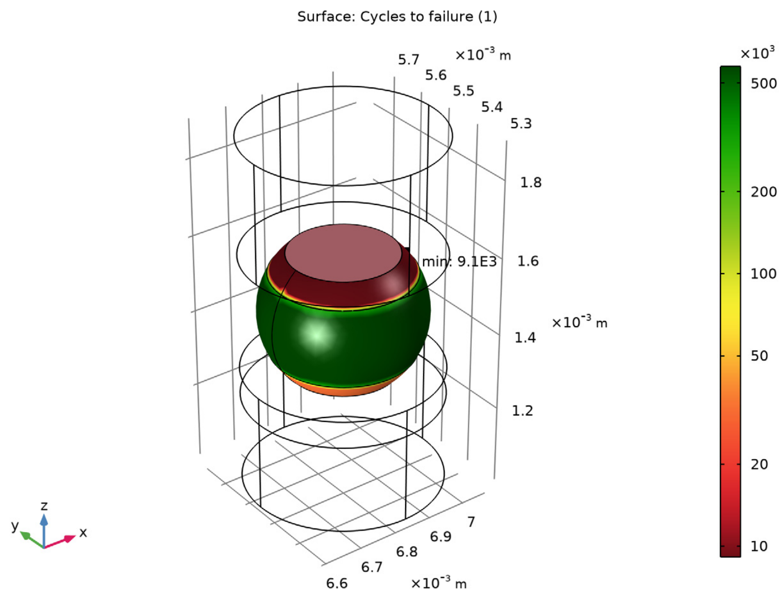

Figure 7: Fatigue prediction based on the volume average of different domains.

Notes About the COMSOL Implementation

In submodeling, the results are mapped from one component — a full model — to another one — a submodel. This is possible through the **General Extrusion** operator. In this example, the mapping of results is simple since it is 1 to 1. The **General Extrusion** operator allows also for mapping into other shapes.

Often, a load history in a fatigue study is repeatable. In order to write the cyclic function in a compact way, a modulo function can be used. This function calculates the reminder of a number, dividend, divided by an other number, divisor. In COMSOL Multiphysics it is defined as `mod(f,p)`, where the first argument is the dividend and the second one is the divisor.

From the numerical point of view, an abrupt change in a load parameter can be challenging. In the current example, a sudden increase or decrease of the power provides such a challenge. In such a case it is favorable to use a smooth function that changes from one value to an other. This is possible in via `flc2hs(t,p)` function that is a Heaviside function with a smooth second derivative. The first argument defines a position when the step takes place and the second argument defines an interval on each side of the step position where the smooth transition takes place.

Application Library path: `Fatigue_Module/Energy_Based/viscoplastic_solder_joints_fatigue`

Modeling Instructions

In this example you will start from an existing model which is an example in the Nonlinear Structural Materials Module.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Nonlinear Structural Materials Module>Viscoplasticity>viscoplastic_solder_joints** in the tree.

3 Click **Open**.

FULL MODEL

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

2 In the **Settings** window for **Component**, type `Full Model` in the **Label** text field.

Define the crack size that will be used in crack models.

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**
`lcrack`	`2.6457e-4[in]`	6.7201E-6 m	`Crack size`

Define the power load cycle.

Analytic 1 (power)

1 In the **Model Builder** window, click **Analytic 1 (power)**.

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

3 In the **Expression** text field, type `(flc2hs(x-0.1,0.1)*50)*(x<6)-flc2hs(mod(x,6)-4.1,0.1)*40+(flc2hs(mod(x,6)-0.1,0.1)*40+10)*(x>=6)`.

FULL MODEL (COMP1)

Create a **General Extrusion** operator that will be used when the submodel is set up.

DEFINITIONS

General Extrusion 1 (genext1)

1 In the **Model Builder** window, expand the **Full Model (comp1)** node.

2 Right-click **Full Model (comp1)>Definitions** and choose **Nonlocal Couplings> General Extrusion**.

3 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.

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

FULL MODEL: LOAD HISTORY

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

2 In the **Settings** window for **Study**, type `Full Model: Load History` in the **Label** text field.

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Full Model: Load History** node, then 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 `0 0.005 range(0.025,0.025,0.5) range(0.75,0.25,3.75) 3.975 4+{range(0,0.025,0.5) range(0.75,0.25,2)} 6+{range(0.025,0.025,0.5) range(0.75,0.25,3.75) 3.975 4+{range(0,0.025,0.5) range(0.75,0.25,2)}} 12+{range(0.025,0.025,0.5) range(0.75,0.25,3.75) 3.975 4+{range(0,0.025,0.5) range(0.75,0.25,2)}} 18+{range(0.025,0.025,0.5) range(0.75,0.25,3.75) 3.975 4+{range(0,0.025,0.5) range(0.75,0.25,2)}}`.

Solver Configurations

In the **Model Builder** window, expand the **Full Model: Load History>Solver Configurations** node.

Step 2: Time Dependent 2

In the **Times** text field, enter the same steps as **Step 1**.

1 In the **Model Builder** window, expand the **Full Model: Load History>Solver Configurations>Solution 1 (sol1)** node.

2 Right-click **Full Model: Load History** and choose **Compute**.

RESULTS

Stress (solid)

Perform a fatigue study on the full model in order to find out which solder joint is the critical one.

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 **Full Model: Load History**.

5 Click **Add to Full Model** in the window toolbar.

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

FATIGUE (FTG)

Energy-Based 1

1 Right-click **Full Model (comp1)>Fatigue (ftg)** 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 **Criterion** list, choose **Darveaux**.

6 From the **Energy type** list, choose **Viscoplastic dissipation density**.

7 Locate the **Fatigue Model Parameters** section. In the a text field, type `lcrack`.

ROOT

In the **Model Builder** window, expand the **Full Model (comp1)>Materials** node.

GLOBAL DEFINITIONS

In the **Model Builder** window, expand the **Global Definitions>Materials** node.

Solder, 60Sn-40Pb (mat4)

1 In the **Model Builder** window, click **Solder, 60Sn-40Pb (mat4)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Crack initiation energy coefficient	K1_Darveaux	13173	1	Darveaux
Crack initiation energy exponent	k2_Darveaux	-1.45	1	Darveaux
Crack propagation energy coefficient	K3_Darveaux	3.92e-7[in]	m	Darveaux
Crack propagation energy exponent	k4_Darveaux	1.12	1	Darveaux
Reference energy density	Wref_Darveaux	1[psi]	J/m³	Darveaux

ROOT

From the **Home** menu, choose **Add Study**.

ADD STUDY

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

2 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Solid Mechanics (solid)** and **Heat Transfer in Solids (ht)**.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Fatigue**.

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

5 From the **Home** menu, choose **Add Study**.

FULL MODEL: FATIGUE EVALUATION

1 In the **Model Builder** window, expand the **Solder, 60Sn-40Pb (mat4)** node, then click **Study 2**.

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

Step 1: Fatigue

1 In the **Model Builder** window, under **Full Model: Fatigue Evaluation** 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 **Full Model: Load History, Time Dependent 2**.

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

7 From the list, select time steps from 18h to 24h.

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

RESULTS

Cycles to Failure (ftg)

The solder joint with the shortest life is the critical one. Create a submodel of that joint.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

SUBMODEL

In the **Settings** window for **Component**, type `Submodel` in the **Label** text field.

GEOMETRY 2

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `viscoplastic_solder_joints.mphbin`.

5 Click **Import**.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `2.4e-4`.

4 In the **Height** text field, type `9e-4`.

5 Locate the **Position** section. In the **x** text field, type `68.28e-4`.

6 In the **y** text field, type `55.38e-4`.

7 In the **z** text field, type `10e-4`.

Union 1 (uni1)

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

2 Click the **Select All** button in the **Graphics** toolbar and remove the cylinder from the selection.

Intersection 1 (int1)

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

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

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

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

3 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

4 In the **z-coordinate** text field, type `13.5e-4`.

Work Plane 2 (wp2)

1 Right-click **Work Plane 1 (wp1)** and choose **Duplicate**.

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

3 In the **z-coordinate** text field, type `15.5e-4`.

Partition Objects 1 (par1)

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

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

3 From the **Partition with** list, choose **Work plane**.

4 From the **Work plane** list, choose **Work Plane 1 (wp1)**.

5 From the **Repair tolerance** list, choose **Relative**.

6 In the **Relative repair tolerance** text field, type `3E-4`.

7 Select the object **int1** only.

Partition Objects 2 (par2)

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

2 Select the object **par1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build All Objects**.

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

GEOMETRY 2

In the **Model Builder** window, collapse the **Submodel (comp2)>Geometry 2** node.

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>Thermal-Structure Interaction>Thermal Stress, Solid**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Full Model: Load History** and **Full Model: Fatigue Evaluation**.

5 Click **Add to Submodel** in the window toolbar.

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

SOLID MECHANICS 2 (SOLID2)

1 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.

2 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 **Submodel (comp2)>Solid Mechanics 2 (solid2)** 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.

Viscoplasticity 1

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

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

3 Click **Clear Selection**.

4 Select Domains 4–6 only.

Prescribe results of the structural analysis via displacements on the shared boundaries.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2 Select Boundaries 1–5, 8, 9, 11, and 22–27 only.
3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4 Select the **Prescribed in x direction** check box.
5 In the u_{0x} text field, type `comp1.genext1(comp1.u)`.
6 Select the **Prescribed in y direction** check box.
7 In the u_{0y} text field, type `comp1.genext1(comp1.v)`.
8 Select the **Prescribed in z direction** check box.
9 In the u_{0z} text field, type `comp1.genext1(comp1.w)`.
10 Click to expand the **Constraint Settings** section. From the **Apply reaction terms on** list, choose **Current physics (internally symmetric)**.

HEAT TRANSFER IN SOLIDS 2 (HT2)

In the **Model Builder** window, under **Submodel (comp2)** click **Heat Transfer in Solids 2 (ht2)**.

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 Select Domain 3 only.
3 In the **Settings** window for **Heat Source**, locate the **Heat Source** section.
4 In the Q_0 text field, type `power(t)`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 Select Boundaries 7, 10, 12–15, 17, and 18 only.
3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4 From the **Flux type** list, choose **Convective heat flux**.
5 In the h text field, type `10`.
6 In the T_{ext} text field, type `T0`.

Prescribe results of the heat transfer analysis via temperature on the shared boundaries.

Temperature 1

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

2 Select Boundaries 1–5, 8, 9, 11, and 22–27 only.

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

4 In the T_0 text field, type `comp1.genext1(comp1.T)`.

MATERIALS

Material Link 5 (matlnk5)

1 In the **Model Builder** window, under **Submodel (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

2 Select Domain 1 only.

Material Link 6 (matlnk6)

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 Select Domain 2 only.

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

4 From the **Material** list, choose **Copper (mat2)**.

Material Link 7 (matlnk7)

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 Select Domain 3 only.

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

4 From the **Material** list, choose **Silicon (mat3)**.

Material Link 8 (matlnk8)

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 Select Domains 4–6 only.

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

4 From the **Material** list, choose **Solder, 60Sn-40Pb (mat4)**.

Refine the mesh in the solder ball.

MESH 2

1 In the **Model Builder** window, under **Submodel (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size 1

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

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

4 Select Domains 4–6 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

6 Click **Build All**.

Simulate the initial cycles using the submodel.

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)**, **Heat Transfer in Solids (ht)**, and **Fatigue (ftg)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Time Dependent**.

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

6 In the **Model Builder** window, click the root node.

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

SUBMODEL: LOAD HISTORY

In the **Settings** window for **Study**, type `Submodel: Load History` in the **Label** text field.

Step 1: Time Dependent

Store data mainly for the thermal cycle where we want to perform the fatigue evaluation.

1 In the **Model Builder** window, under **Submodel: Load History** click **Step 1: Time Dependent**.

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `0 range(1,1,17) 18+{range(0,0.025,0.5) range(0.75,0.25,3.75) 3.95 4+{range(0.025,0.025,0.5) range(0.75,0.25, 2)}}`.

5 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

7 From the **Study** list, choose **Full Model: Load History, Time Dependent 2**.

8 From the **Time (h)** list, choose **Automatic (all solutions)**.

Set up the solver in a similar way as in **Study 1**.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.

3 In the **Model Builder** window, expand the **Submodel: Load History>Solver Configurations>Solution 4 (sol4)>Dependent Variables 1** node, then click **Viscoplastic dissipation density (comp2.solid2.Wvp)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `1e5`.

7 In the **Model Builder** window, under **Submodel: Load History>Solver Configurations>Solution 4 (sol4)** click **Time-Dependent Solver 1**.

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

9 From the **Steps taken by solver** list, choose **Strict**.

10 Click to expand the **Output** section. Clear the **Store time derivatives** check box.

11 In the **Model Builder** window, expand the **Submodel: Load History>Solver Configurations>Solution 4 (sol4)>Time-Dependent Solver 1** node, then click **Segregated 1**.

12 In the **Settings** window for **Segregated**, locate the **General** section.

13 From the **Termination technique** list, choose **Iterations**.

14 In the **Model Builder** window, expand the **Submodel: Load History>Solver Configurations>Solution 4 (sol4)>Time-Dependent Solver 1>Segregated 1** node, then click **Temperature**.

15 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

16 From the **Termination technique** list, choose **Tolerance**.

17 In the **Tolerance factor** text field, type `1`.

18 In the **Model Builder** window, under **Submodel: Load History>Solver Configurations>Solution 4 (sol4)>Time-Dependent Solver 1>Segregated 1** click **Solid Mechanics 2**.

19 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

20 From the **Nonlinear method** list, choose **Constant (Newton)**.

21 From the **Termination technique** list, choose **Tolerance**.

22 In the **Maximum number of iterations** text field, type 20.

23 Locate the **General** section. In the **Variables** list, select **Viscoplastic dissipation density (comp2.solid2.Wvp)**.

24 Under **Variables**, click **Delete**.

25 In the **Model Builder** window, under **Submodel: Load History>Solver Configurations>Solution 4 (sol4)>Time-Dependent Solver 1** right-click **Segregated 1** and choose **Segregated Step**.

26 Drag and drop **Submodel: Load History**
Solver Configurations
Solution 4 (sol4)
Time-Dependent Solver 1
Segregated 1
Segregated Step 3 below **Solid Mechanics 2**.

27 In the **Settings** window for **Segregated Step**, locate the **General** section.

28 Under **Variables**, click **Add**.

29 In the **Add** dialog box, select **Viscoplastic dissipation density (comp2.solid2.Wvp)** in the **Variables** list.

30 Click **OK**.

31 In the **Settings** window for **Segregated Step**, type `Energy Dissipation` in the **Label** text field.

32 Locate the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.

33 In the **Tolerance factor** text field, type 1.

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

RESULTS

Stress (solid2)

Simulate the fatigue load cycle on the submodel and evaluate fatigue response of the critical solder joint.

SUBMODEL (COMP2)

In the **Model Builder** window, click **Submodel (comp2)**.

ADD PHYSICS

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

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

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

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Full Model: Load History**, **Full Model: Fatigue Evaluation**, and **Submodel: Load History**.

5 Click **Add to Submodel** in the window toolbar.

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

FATIGUE 2 (FTG2)

Energy-Based 1

1 Right-click **Submodel (comp2)>Fatigue 2 (ftg2)** and choose the domain evaluation **Energy-Based**.

2 Select Domains 4–6 only.

3 In the **Settings** window for **Energy-Based**, locate the **Solution Field** section.

4 From the **Physics interface** list, choose **Solid Mechanics 2 (solid2)**.

5 Locate the **Fatigue Model Selection** section. From the **Criterion** list, choose **Darveaux**.

6 From the **Energy type** list, choose **Viscoplastic dissipation density**.

7 Locate the **Evaluation Settings** section. From the **Volume average method** list, choose **Entire selection**.

8 Locate the **Fatigue Model Parameters** section. In the a text field, type `lcrack`.

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 **Recently Used>Fatigue (ftg)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Full Model: Load History**, **Full Model: Fatigue Evaluation**, and **Submodel: Load History**.

5 Click **Add to Submodel** in the window toolbar.

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

FATIGUE 3 (FTG3)

Right-click **Submodel (comp2)>Fatigue 3 (ftg3)** and choose the domain evaluation **Energy-Based**.

Energy-Based 1

1 Select Domains 4–6 only.

2 In the **Settings** window for **Energy-Based**, locate the **Solution Field** section.

3 From the **Physics interface** list, choose **Solid Mechanics 2 (solid2)**.

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

5 From the **Energy type** list, choose **Viscoplastic dissipation density**.

6 Locate the **Fatigue Model Parameters** section. In the a text field, type `lcrack`.

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)**, **Heat Transfer in Solids (ht)**, **Fatigue (ftg)**, **Solid Mechanics 2 (solid2)**, and **Heat Transfer in Solids 2 (ht2)**.

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.

SUBMODEL: FATIGUE EVALUATION

1 In the **Model Builder** window, click **Study 4**.

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

Step 1: Fatigue

1 In the **Model Builder** window, under **Submodel: Fatigue Evaluation** 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 **Submodel: Load History, Time Dependent**.

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

7 From the list, select time steps from 18h to 24h.

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

RESULTS

Temperature (ht2)

Compare the viscoplastic dissipation in the full model and in the submodel.

Dissipation History

1. In the **Model Builder** window, click **Dissipation History**.
2. In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1. In the **Model Builder** window, expand the **Dissipation History** node, then click **Point Graph 1**.
2. In the **Settings** window for **Point Graph**, click to expand the **Legends** section.
3. Select the **Show legends** check box.
4. From the **Legends** list, choose **Manual**.
5. In the table, enter the following settings:

Legends
`Full model`

Point Graph 2

1. Right-click **Results>Dissipation History>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 `solid2.WvpGp`.
4. In the **Unit** field, type `kJ/m^3`.
5. Locate the **Data** section. From the **Dataset** list, choose **Submodel: Load History/ Solution 4 (5) (sol4)**.
6. Locate the **Selection** section. Click to select the **Activate Selection** toggle button.
7. Select Point 9 only.
8. Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
9. From the **Positioning** list, choose **In data points**.

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

Legends
`Submodel`

Shear Viscoplasticity History

1 In the **Model Builder** window, right-click **Dissipation History** and choose **Duplicate**.

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

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `Shear viscoplasticity`.

Point Graph 1

1 In the **Model Builder** window, expand the **Shear Viscoplasticity History** node, then click **Point Graph 1**.

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 **Full Model (comp1)>Solid Mechanics>Strain (Gauss points)>Viscoplastic strain tensor, local coordinate system>solid.evplGp13 - Viscoplastic strain tensor, local coordinate system, 13 component**.

Point Graph 2

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

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 **Submodel (comp2)>Solid Mechanics 2>Strain (Gauss points)>Viscoplastic strain tensor, local coordinate system>solid2.evplGp13 - Viscoplastic strain tensor, local coordinate system, 13 component**.

3 In the **Shear Viscoplasticity History** toolbar, click **Plot**.

Verify that the temperature is the same in both studies.

Temperature History

1 In the **Model Builder** window, under **Results** click **Temperature History**.

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

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **y-axis label** check box.

5 In the associated text field, type `Temperature (C)`.

Point Graph 1

1 In the **Model Builder** window, expand the **Temperature History** node, then click **Point Graph 1**.

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

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Full model`

Point Graph 2

1 Right-click **Results>Temperature History>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 `T2`.

4 In the **Unit** field, type `degC`.

5 Locate the **Data** section. From the **Dataset** list, choose **Submodel: Load History/ Solution 4 (5) (sol4)**.

6 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

7 Select Point 9 only.

8 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

10 From the **Positioning** list, choose **In data points**.

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

Legends
`Submodel`

Display the minimum cycles to failure for each solution.

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 **Text Format** section.

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

Cycles to Failure (ftg2)

In the **Model Builder** window, expand the **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 **Text Format** section.

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

Cycles to Failure (ftg3)

In the **Model Builder** window, expand the **Cycles to Failure (ftg3)** node.

Marker 1

1 In the **Model Builder** window, expand the **Results>Cycles to Failure (ftg3)>Surface 1** node, then click **Marker 1**.

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

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

Evaluation Group 1

In the **Results** toolbar, click **Evaluation Group**.

Volume Minimum 1

1 Right-click **Evaluation Group 1** and choose **Volume Minimum**.

2 In the **Settings** window for **Volume Minimum**, locate the **Data** section.

3 From the **Dataset** list, choose **Full Model: Fatigue Evaluation/Solution 3 (sol3)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Solder**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Full Model (comp1)>Fatigue>ftg.ctf - Cycles to failure**.

Evaluation Group 1

In the **Model Builder** window, click **Evaluation Group 1**.

Volume Minimum 2

1 In the **Evaluation Group 1** toolbar, click MIN **Minimum** and choose **Volume Minimum**.

2 In the **Settings** window for **Volume Minimum**, locate the **Data** section.

3 From the **Dataset** list, choose **Submodel: Fatigue Evaluation/Solution 5 (7) (sol5)**.

4 Select Domains 4–6 only.

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

Expression	Unit	Description
ftg2.ctf	1	Cycles to failure, submodel
ftg3.ctf	1	Cycles to failure, thin layer in submodel

6 In the **Evaluation Group 1** toolbar, click **= Evaluate**.