COMSOL MULTIPHYSICS®

Viscoplastic Creep in Solder Joints

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Viscoplastic Creep in Solder Joints

Introduction

This example studies viscoplastic creep in solder joints under thermal loading using the Anand viscoplasticity model.

The Anand model is suitable for large, isotropic, viscoplastic deformations in combination with small elastic deformations. The following flow equation takes the strain rate dependence on the stress into account:

$$\dot{\varepsilon}_p = A\exp\left(-\frac{Q}{RT}\right)\left[\sinh\left(\xi\frac{\sigma}{s}\right)\right]^{\frac{1}{m}} \tag{1}$$

where $\dot{\varepsilon}_p$ is the inelastic strain rate, A is the pre-exponential factor, Q is the activation energy, m is the strain rate sensitivity, ξ is the multiplier of stress, R is the Boltzmann constant, and T is the absolute temperature.

The internal variable s is called deformation resistance, and it obeys the evolution equation

$$\dot{s} = \left\{h_0\left|1-\frac{s}{s^*}\right|^a \operatorname{sign}\left(1-\frac{s}{s^*}\right)\right\}\dot{\varepsilon}_p \tag{2}$$

where

$$s^* = \hat{s}\left[\frac{\dot{\varepsilon}_p}{A}\exp\left(\frac{Q}{RT}\right)\right]^n \tag{3}$$

is the saturation value of s, h_0 is the hardening constant, a is the strain rate sensitivity of hardening, $\hat{s}$ is the coefficient for deformation resistance saturation, and n is the strain rate sensitivity of deformation resistance.

Model Definition

The model geometry is shown in Figure 1. It includes two electronic components (chips) mounted on a circuit board by means of several solder ball joints.

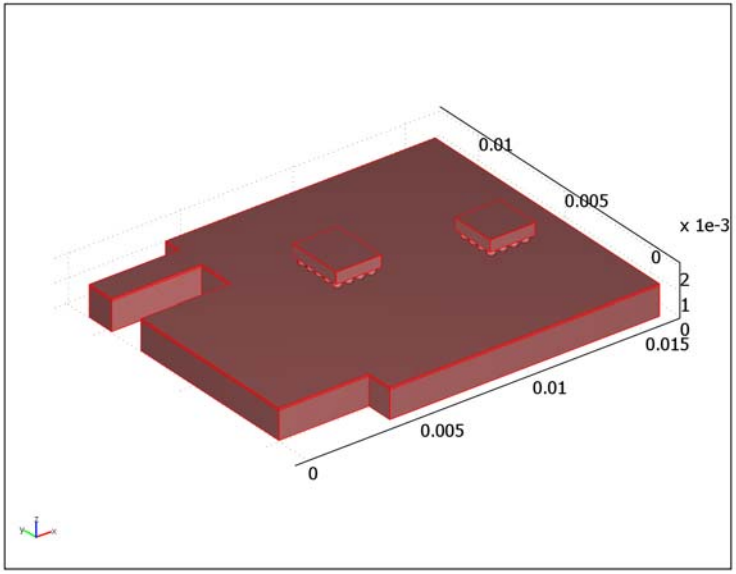


Figure 1: Model geometry.

The solder material is 60Sn40Pb. The circuit bard consists of two layers: a thin layer of copper and a thicker layer of FR4 material. The chips are made of silicon. You can find the material and thermal properties for these three materials and for 60Sn40Pb in the material library available in COMSOL Multiphysics.

The nine material parameters needed to apply the Anand model for this solder are available in the literature (Ref. 1). They are summarized in the following table:

TABLE 1: MODEL DATA FOR THE VISCOPLASTIC SOLDER JOINTS MODEL

PROPERTY	VALUE	DESCRIPTION
A	$1.49 \cdot 10^7$ 1/s	Pre-exponential factor
Q/R	10,830 K	Activation energy/Boltzmann constant
m	0.303	Strain rate sensitivity of stress
n	0.0231	Sensitivity for deformation resistance
a	1.34	Strain rate sensitivity of hardening
$\hat{s}$	80.42 MPa	Coefficient for deformation resistance saturation
s_0	56.33 MPa	Initial value of deformation resistance

TABLE 1: MODEL DATA FOR THE VISCOPLASTIC SOLDER JOINTS MODEL

PROPERTY	VALUE	DESCRIPTION
ξ	11	Multiplier of stress
h_0	2640.75 MPa	Hardening constant
T_0	313 K	Initial temperature

The structure has initially constant temperature $T_0 = 313$ K. The heat generation within the chips causes the thermal loading of the structure. The simulation runs for 90 s of such loading. You model it by the constant heat source $Q = 1{\cdot}10^8$ W/m^3.

Results and Discussion

The temperature distribution after 80 s of operation is shown in Figure 2. The maximum temperature increase is about 55 degrees.

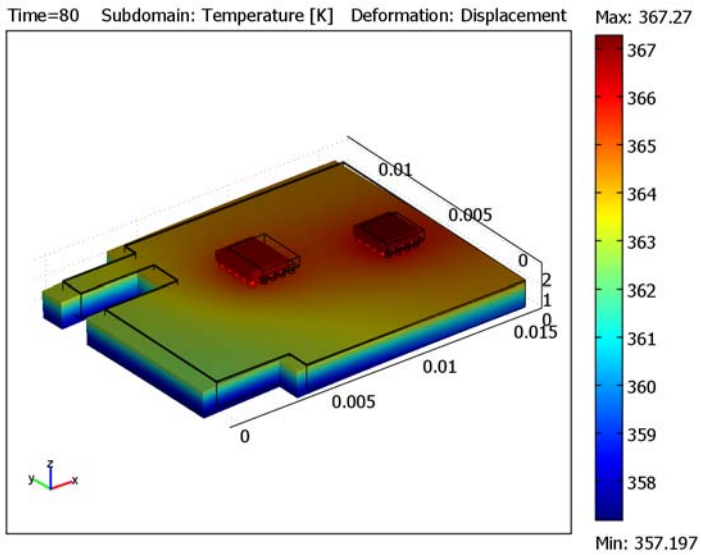


Figure 2: Final temperature distribution.

Figure 3 shows about 6% increase of the deformation resistance at the end of the process.

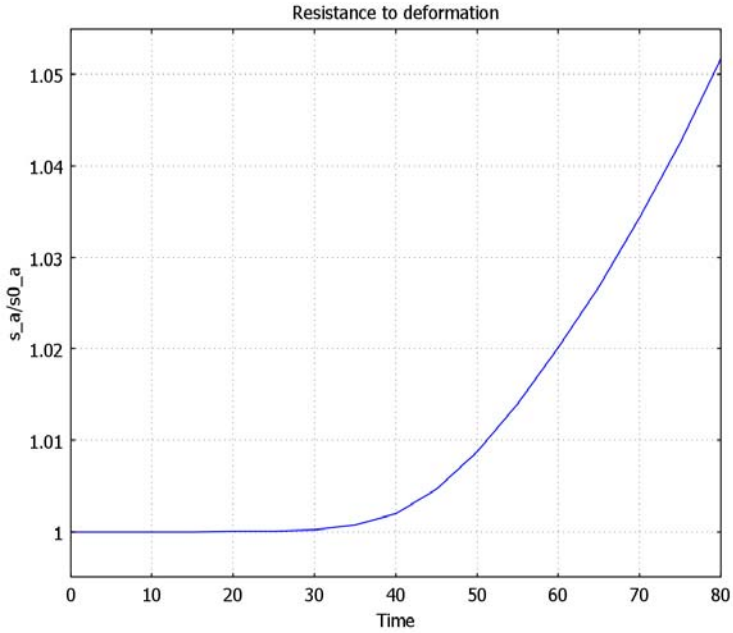


Figure 3: Hardening.

The elastic and inelastic strains at the same position within the solder are shown in Figure 4. A clear start of intensive plastic flow appears after about 40 s of the loading. This is in good agreement with the material behavior and shows the ability of Anand model to predict results comparable with the plasticity models based on the yield stress criteria, which requires computations of the yield surfaces.

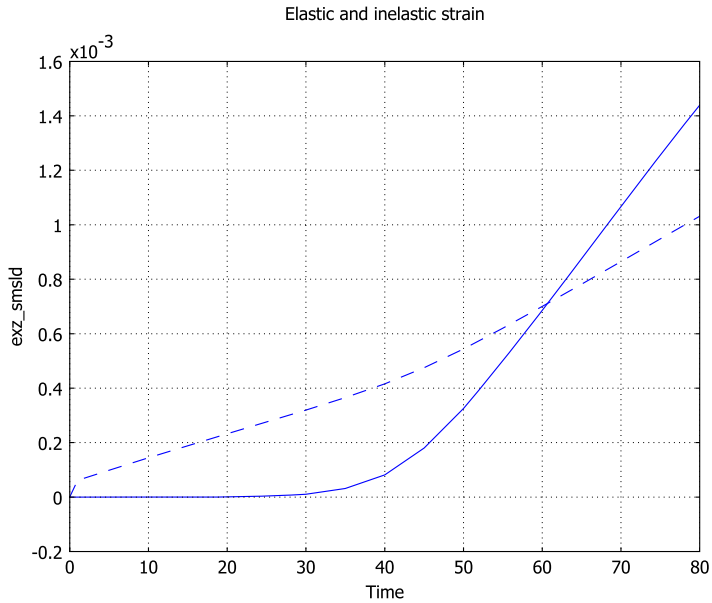


Figure 4: Viscoplasticity.

Modeling in COMSOL Multiphysics

You use two PDE general form modes to compute the inelastic strains and resistance to deformation, respectively. You couple these modes to the Solid, Stress-Strain application mode. Because the same displacement field represents a combination of both plastic and elastic deformation, you need to modify the application mode to exclude the inelastic strains from the expressions for the elastic stresses (Hooke's law). To achieve this, use the possibility provided in COMSOL Multiphysics to edit any predefined expressions on the equation-system level.

Reference

1. Z.N. Cheng, G.Z. Wang, L. Chen, J. Wilde, and K. Becker, "Viscoplastic Anand model for solder alloys and its application," *Soldering & Surface Mount Technology*, vol. 12, no. 2, pp. 31–36, 2000.

Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/viscoplastic_solder_joints`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator** select **3D** from the **Space dimension** list on the **New** page.

2 Click the **Multiphysics** button.

3 Select **Structural Mechanics Module>Solid, Stress-Strain>Transient analysis**.

4 Click **Add**.

5 Select **COMSOL Multiphysics>Heat Transfer>Conduction>Transient analysis**.

6 Click **Add**.

7 Select **COMSOL Multiphysics>PDE Modes>PDE, General Form>Time-dependent analysis**.

8 In the **Application mode name** edit field, type `creep`.

9 In the **Dependent variables** edit field, type `ex_c ey_c ez_c exy_c eyz_c exz_c`.

10 Click **Add**.

11 In the **Application mode name** edit field, type `resistance`.

12 In the **Dependent variables** edit field, type `s_a`.

13 Click **Add**, then click **OK**.

GEOMETRY MODELING

1 From the **File** menu, select **Import>CAD Data From File**.

2 Browse to the model's Model Library folder (see the Model Library path on page 6).

3 Select the file `viscoplastic_solder_joints_geom.mphbin`, then click **Import**.

4 Click the **Zoom Extents** button on the Main toolbar to see the geometry similar to that shown in Figure 1.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu, select **Constants**.

2 Click the **Import Variables From File** icon.

3 In the **Import Variables** dialog box, browse to the Model Library folder of the model MPH-file and select the file: `viscoplastic_solder_joints_const.txt`, then click **Open**.

4 Click **OK** to close the **Constants** dialog box.

Scalar Expressions

1 From the **Options** menu, select **Expressions>Scalar Expressions**.

2 Click the **Import Variables From File** icon.

3 Select the file `viscoplastic_solder_joints_expr.txt`, then click **Open**.

4 Click **OK** to close the **Scalar Expressions** dialog box.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Multiphysics** menu, select **1 Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select all subdomains.

4 Click the **Load** tab and select the **Include thermal expansion** check box.

5 Type `T` in the **Temp** edit field and `T0` in the **Tempref** edit field.

6 Select Subdomain 1.

7 On the **Material** page, click the **Load** button.

8 Select **Basic Material Properties>FR4 (circuit board)**, then click **OK**.

9 Select Subdomain 2.

10 On the **Material** page, click the **Load** button.

11 Select **Basic Material Properties>Copper**, then click **OK**.

12 Select Subdomains 3 and 24.

13 On the **Material** page, click the **Load** button.

14 Select **Basic Material Properties>Silicon**, then click **OK**.

15 Select Subdomains 4–23 and 25–40.

16 On the **Material** page, click the **Load** button.

17 Select **Basic Material Properties>Solder, 60Sn-40Pb**, then click **OK**.

18 Go to the **Groups** page and type `solder` in the **Name** edit field.

19 Click **OK**.

Editing Equations

1 From the **Physics** menu, select **Equation System>Subdomain Settings**.

2 Select Subdomains 4–23 and 25–40.

3 Click the **Variables** tab.

4 Change the following seven variables as indicated in the table:

NAME	EXPRESSION
sx_smsld	4*G_smsld*(ex_smsld-ex_c)/3-2*G_smsld*(ey_smsld-ey_c)/3-2*G_smsld*(ez_smsld-ez_c)/3-p
sy_smsld	4*G_smsld*(ey_smsld-ey_c)/3-2*G_smsld*(ex_smsld-ex_c)/3-2*G_smsld*(ez_smsld-ez_c)/3-p
sz_smsld	4*G_smsld*(ez_smsld-ez_c)/3-2*G_smsld*(ex_smsld-ex_c)/3-2*G_smsld*(ey_smsld-ey_c)/3-p
sxy_smsld	2*G_smsld*(exy_smsld-exy_c)
syz_smsld	2*G_smsld*(eyz_smsld-eyz_c)
sxz_smsld	2*G_smsld*(exz_smsld-exz_c)
p	-K_smsld*(evol_smsld-ex_c-ey_c-ez_c-3*mat4_alpha*(Temp_smsld-Tempref_smsld))

5 Click **Apply**, then click the **Differentiate** button.

6 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, select **Boundary Settings**.

2 Select Boundaries 17 and 19.

3 On the **Constraint** page, select **x-z symmetry plane** from the **Constraint condition** list.

4 Select Boundaries 182 and 183.

5 On the **Constraint** page, select **y-z symmetry plane** from the **Constraint condition** list.

6 Click **OK**.

Point Settings

1 From the **Physics** menu, select **Point Settings**.

2 Select Point 193.

3 On the **Constraint** page, select the **Constraint z-dir.** check box. Leave the edit field at its default value of 0.

4 Click **OK**.

Subdomain Settings

1 From the **Multiphysics** menu, select **2 Heat Transfer by Conduction (ht)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 On the **Subdomains** page, select all subdomains.

4 On the **Init** page, type `T0` in the **Temperature** edit field.

5 Select Subdomain 1.

6 On the **Material** page, select **FR4 (circuit board)** from the **Library material** list.

7 Select Subdomain 2.

8 Select **Copper** from the **Library material** list.

9 Select Subdomains 3 and 24.

10 Select **Silicon** from the **Library material** list.

11 Type `1e8` in the **Heat source** edit field.

12 Select Subdomains 4–23 and 25–40.

13 On the **Material** page, select **Solder, 60Sn-40Pb** from the **Library material** list.

14 Go to the **Groups** page, and type `solder` in the **Name** edit field.

15 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, select **Boundary Settings**.

2 Select Boundary 3.

3 On the **Coefficients** page, select **Heat flux** from the **Boundary condition** list.

4 Type `10` in the **Heat transfer coefficient** edit field and `T0` in the **External temperature** edit field.

5 Click **OK**.

Subdomain Settings

1 From the **Multiphysics** menu, select **3 PDE General form (creep)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomains 1–3 and 24.

4 Clear the **Active in this domain** check box.

5 Click **Apply**.

6 Select Subdomains 4–23 and 25–40.

7 On the Γ page, set all the edit fields of the **Flux vector** to `0`.

8 On the **F** page, set the edit fields according to this table:

`1.5*ep_a*sdx/(mises_smsld+eps)`
`1.5*ep_a*sdy/(mises_smsld+eps)`
`1.5*ep_a*sdz/(mises_smsld+eps)`
`1.5*ep_a*sxy_smsld/(mises_smsld+eps)`
`1.5*ep_a*syz_smsld/(mises_smsld+eps)`
`1.5*ep_a*sxz_smsld/(mises_smsld+eps)`

9 On the **Init** page, type `eps` in the first (upper) six edit fields. Leave the remaining six edit fields at the default value of `0`.

10 Go to the **Groups** page and type `solder` in the **Name** edit field.

11 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, select **Boundary Settings**.

2 Select all boundaries.

3 On the **Type** page, click the **Neumann boundary condition** option button.

4 Click **OK** to close the dialog box.

Subdomain Settings

1 From the **Multiphysics** menu, select **4 PDE General form (resistance)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomains 1–3 and 24.

4 Clear the **Active in this domain** check box.

5 Select Subdomains 4–23 and 25–40.

6 Type `0` in all three edit fields for the **Flux vector**.

7 Type `h_a*ep_a` in the **Source term** edit field.

8 Click the **Init** tab.

9 Type `s0_a` in the first edit field; leave the other one at the default value of `0`.

10 Go to the **Groups** page and type `solder` in the **Name** edit field.

11 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, select **Boundary Settings**.

2 Select all boundaries.

3 On the **Type** page, click the **Neumann boundary condition** option button.

4 Click **OK** to close the dialog box.

MESH GENERATION

1 From the **Mesh** menu, select **Free Mesh Parameters**.

2 On the **Global** page, select **Coarse** from the **Predefined mesh sizes** list.

3 On the **Subdomain** page, select Subdomains 4–23 and 25–40.

4 Click the **Mesh Selected** button.

5 Go back to the **Global** page and select **Coarser** from the **Predefined mesh sizes** list.

6 On the **Boundary** page, select Boundaries 7, 28, and 114.

7 Select **Triangle** from the **Method** list.

8 Click the **Mesh Selected** button.

9 Click **OK** to close the dialog box.

10 From the **Mesh** menu, select **Swept Mesh Parameters**.

11 Check that **Coarser** is selected in the **Predefined mesh sizes** list.

12 Select Subdomains 1–3 and 24.

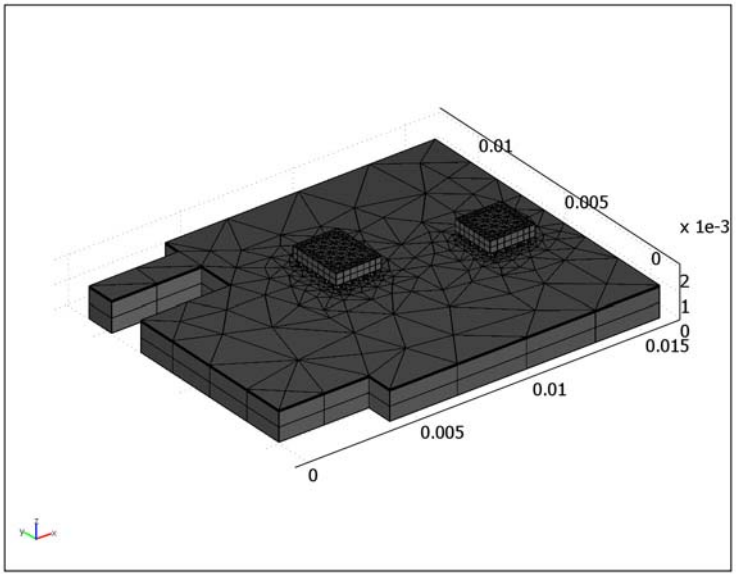
13 Select the **Manual specification of element layers** check box.

14 Set the **Number of element layers** to 2.

15 Click the **Mesh Selected** button.

16 Click **OK**.

This produces a mesh similar to that shown in the following figure:



COMPUTING THE SOLUTION

1 From the **Solve** menu, select **Solver Manager**.

2 In the **Initial value** area, click the **Initial value expression** button.

3 Click **OK** to close the **Solver Manager** dialog box.

4 From the **Solve** menu, select **Solver Parameters**.

5 In the **Analysis type** area, select **Quasi-static transient** from the drop-down list under **Solid Stress-Strain (smsld)** only.

6 On the **General** page, select **GMRES** from the **Linear system solver** list and **Geometric multigrid** from the **Preconditioner** list.

7 Enter the following values in the indicated edit fields:

SETTING	VALUE
Times	`0 1 range(5,5,80)`
Relative tolerance	`0.01`
Absolute tolerance	`0.0010`

8 Select the **Plot while solving** check box.

9 Click the **Settings** button in the **Linear system solver** area.

10 In the **Linear system solver** list to the left, click **Coarse solver**. Select **PARDISO** from the **Coarse solver** list, then click **OK** to return to the **Solver Parameters** dialog box.

11 Click the **Time Stepping** tab.

12 From the **Method** list, select **Generalized alpha**.

13 From the **Time steps taken by solver** list, select **Manual**.

14 In the **Time step** edit field type `1`.

15 In the **Advanced** area, set the **Amp. factor for high frequencies** to `0.5`.

16 Click **OK** to close the **Solver Parameters** dialog box.

17 From the **Solve** menu, select **Get Initial Value**.

18 Click the **Plot Parameters** button on the Main toolbar.

19 In the **Plot type** area, clear the **Slice** check box and select the **Subdomain** and **Deformed shape** check boxes.

20 On the **Subdomain** page, type `T` in the **Expression** edit field.

21 Click **OK** to close the dialog box.

22 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

When the solver is finished, you get a plot of the final temperature distribution, which closely resembles that shown in Figure 2.

To plot the hardening, elastic, and inelastic strains, follow these steps:

1 From the **Postprocessing menu**, select **Domain Plot Parameters**.

2 On the **General** page, select **New figure** from the **Plot in** menu.

3 Click **Title/Axis** button.

4 Click the **Title** option button, then type `Resistance to deformation` in the associated edit field.

5 Click **OK** to close the dialog box.

6 Click the **Point** tab.

7 Select Point 36 and enter `s_a/s0_a` in the **Expression** edit field.

8 Click **Apply** to produce a plot of the hardening shown in Figure 3.

9 Type `exz_c` in the **Expression** edit field.

10 Go to the **General** page and select **New figure** from the **Plot in** menu.

11 Click **Title/Axis** button.

12 Click the **Title** option button, then type `Elastic and inelastic strain` in the associated edit field.

13 Click **OK** to close the dialog box.

14 Click **Apply**.

15 Select the **Keep current plot** check box.

16 Click the **Point** tab.

17 Type `exz_smsld` in the **Expression** edit field.

18 Click the **Line Settings** button.

19 Change the **Line style** to **Dashed line**, then click **OK**.

20 Click **OK** to display the graphs.

This produces the strain plot in Figure 4.