Created in COMSOL Multiphysics 6.2

Thermal Stresses in a Layered Plate

Introduction

In this example, thermal stresses in a layered plate are analyzed. The plate consists of three layers: a coating, a substrate, and a carrier. The coating is deposited onto the substrate at a temperature of 800°C. At this temperature both the coating and the substrate are stress-free. During the first stage of the analysis, the temperature of the plate is lowered to 150°C, which induces thermal stresses in the coating/substrate assembly. At this temperature the coating/substrate assembly is epoxied to a stress-free carrier layer. During the second stage of the analysis, the temperature in the entire assembly is lowered to 20°C, and the thermal stresses are examined.

Model Definition

The plate is considered to be thick and therefore in a state of plane strain. It is modeled using the 2D Solid Mechanics interface. The geometry of the plate is shown in Figure 1. The bottom layer of the geometry is the carrier, the middle layer is the substrate, and the top layer is the coating.

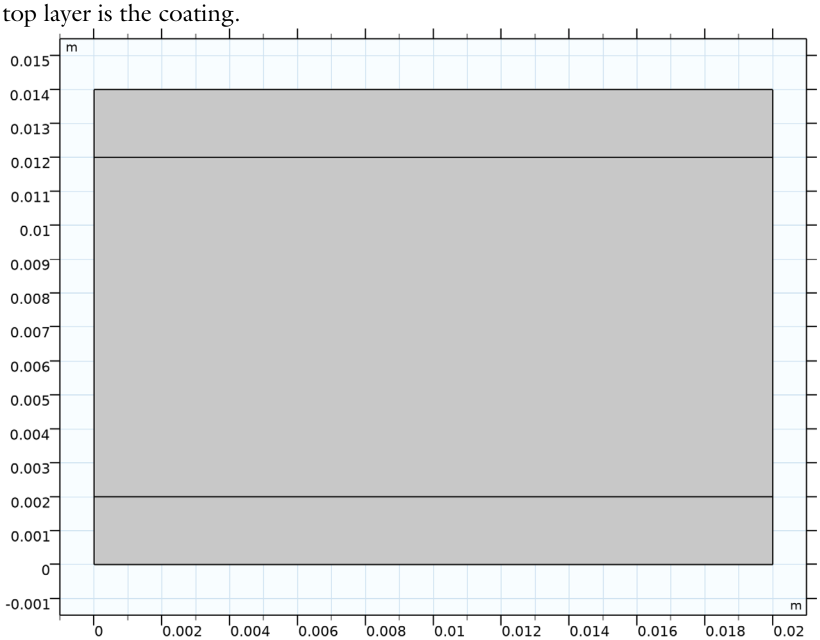

Figure 1: The plate geometry.

Material Properties

The three layers are modeled as isotropic and linear elastic. Their coefficients of thermal expansion are constant. The material properties of the layers are shown in Table 1, Table 2, and Table 3.

TABLE 1: MATERIAL PROPERTIES OF THE CARRIER.

MATERIAL PROPERTY	VALUE
E	215 GPa
ν	0.3
ρ	1000 kg/m^3
α	$6.6 \cdot 10^{-6}$ K^{-1}

TABLE 2: MATERIAL PROPERTIES OF THE SUBSTRATE.

MATERIAL PROPERTY	VALUE
E	130 GPa
ν	0.28
ρ	1000 kg/m^3
α	$3 \cdot 10^{-6}$ K^{-1}

TABLE 3: MATERIAL PROPERTIES OF THE COATING.

MATERIAL PROPERTY	VALUE
E	70 GPa
ν	0.17
ρ	1000 kg/m^3
α	$5 \cdot 10^{-7}$ K^{-1}

Activation of the Carrier

The carrier is only present at the second stage of the analysis. The activation of this layer is readily performed using the **Activation** subnode under **Linear Elastic Material**. Note that the carrier will be activated in a stress-free state, even though its strain reference temperature (800°C) is different from the temperature at activation.

Loading and Boundary Conditions

Loading on the plate consists of an applied homogeneous temperature field. First, the temperature of the coating and substrate is reduced from the initial temperature 800°C to 150°C. During this temperature change, the carrier is not yet present. At 150°C, the carrier is activated using the **Activation** subfeature, and the temperature of the whole assembly is reduced to 20°C.

The plate is constrained using the **Rigid Motion Suppression** feature.

Results and Discussion

Figure 2 shows the normal stress in the x direction after the first stage of the analysis. The substrate material has a higher coefficient of thermal expansion than the coating material. This means that the substrate shrinks more than the coating, causing tensile stresses in the substrate area next to the coating and compressive stresses in the coating.

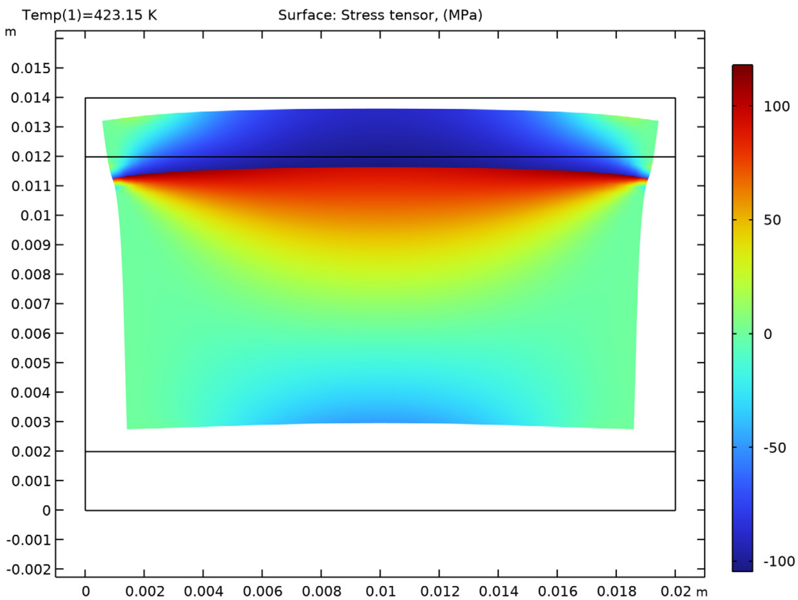

Figure 2: Normal stress in the x direction for the first stage of the analysis.

Note that during the first stage of the analysis, the carrier is inactive.

Figure 3 shows the normal stress in the x direction after the second stage of the analysis, where the temperature is lowered to 20 °C. The stress levels in the substrate have increased slightly near to the coating, as have the compressive stresses in the coating compared to after the first stage.

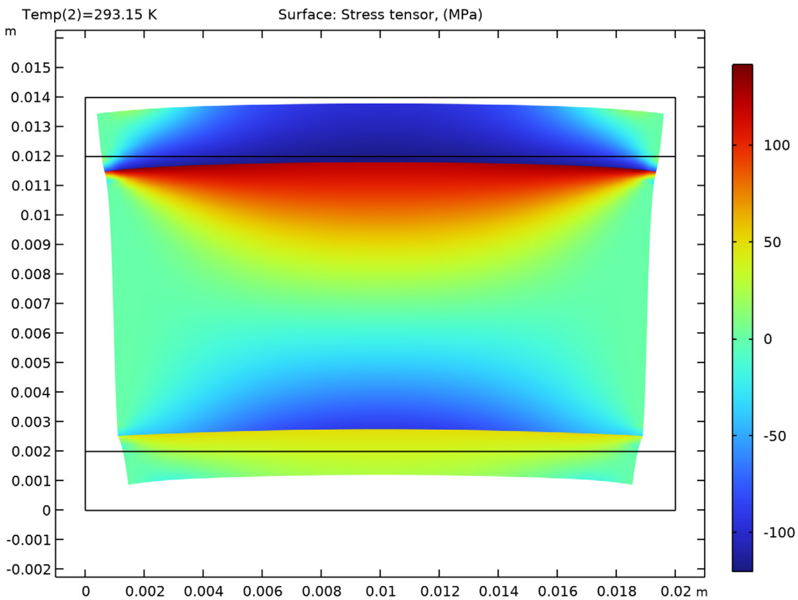

Figure 3: Residual thermal stress at room temperature.

The coefficient of thermal expansion is higher in the carrier than in the substrate. As the temperature is decreased, the carrier experiences tensile stresses, while the substrate near the carrier experiences compressive stresses.

Figure 4 below shows how the bottom surface deformation from a planar surface.

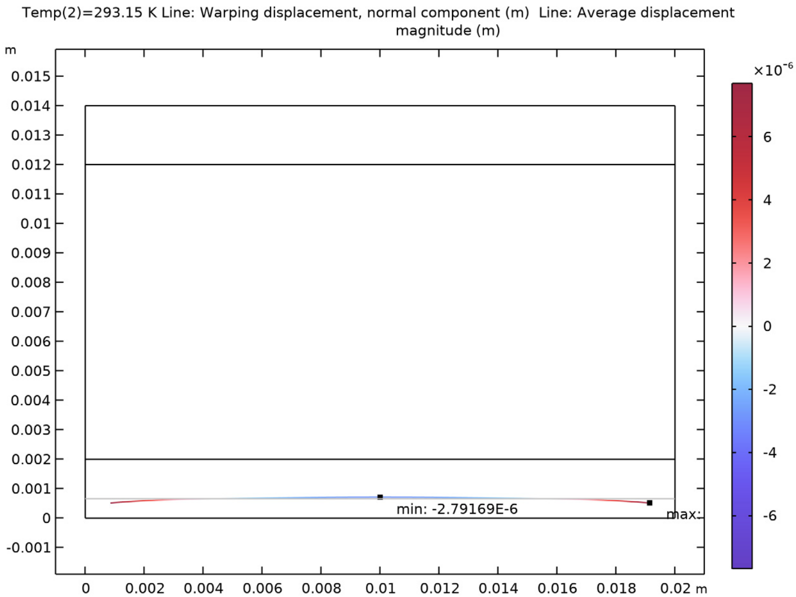

Figure 4: Warping displacement at the bottom surface.

Application Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/layered_plate`

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>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`Tdeposition`	`800[degC]`	1073.2 K	`Coating deposition temperature`
`Tepoxying`	`150[degC]`	423.15 K	`Temperature when the coating/ substrate is epoxied to the carrier`
`Troom`	`20[degC]`	293.15 K	`Room temperature`
`Temp`	`1[K]`	1 K	`Temperature parameter`

GEOMETRY 1

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 `0.02`.

4 In the **Height** text field, type `0.014`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.002`
`Layer 2`	`0.01`

6 Click **Build All Objects**.

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

SOLID MECHANICS (SOLID)

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

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

Linear Elastic Material 1

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

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_{ref} list, choose **User defined**. In the associated text field, type `Tdeposition`.
4. From the T list, choose **User defined**. In the associated text field, type `Temp`.

The carrier is only active during the second stage of the analysis. Use an **Activation** node for conditional activation of the domain.

Linear Elastic Material 1

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

Activation 1

1. In the **Physics** toolbar, click **Attributes** and choose **Activation**.
2. In the **Settings** window for **Activation**, locate the **Domain Selection** section.
3. Click **Clear Selection**.
4. Select Domain 1 only.
5. Locate the **Activation** section. In the **Activation expression** text field, type `Temp<Tepoxying`.

Rigid Motion Suppression 1

1. In the **Physics** toolbar, click **Domains** and choose **Rigid Motion Suppression**.
2. Select Domain 2 only.

Warpage 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Warpage**.
2. Select Boundary 2 only.

MATERIALS

Carrier

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 `Carrier` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Domain 1 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`2.15e11`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`1000`	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`6e-6`	1/K	Basic

Substrate

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

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

3 Select Domain 2 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`1.3e11`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.28`	1	Young's modulus and Poisson's ratio

Property	Variable	Value	Unit	Property group
Density	rho	1000	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	3e-6	1/K	Basic

Coating

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

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

3 Select Domain 3 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	7e10	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.17	1	Young's modulus and Poisson's ratio
Density	rho	1000	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	5e-7	1/K	Basic

MESH 1

Mapped 1

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

Size

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Temp (Temperature parameter)	`Tepoxying Troom`	K

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.

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

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

4 From the **Unit** list, choose **MPa**.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

7 Click **OK**.

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

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

3 From the **Parameter value (Temp (K))** list, choose **423.15**.

4 In the **Stress (solid)** toolbar, click **Plot**.

5 From the **Parameter value (Temp (K))** list, choose **293.15**.

6 In the **Stress (solid)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

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

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Solid Mechanics>Warpage (wrp1)**.

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

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