Uniaxial Loading of a Shape Memory Alloy

Introduction

Shape memory alloys (SMAs) are used in a wide range of applications due to their ability to recover their initial shape when heated, and to the hysteresis they show during a loading-unloading cycle before recovering their initial state. These two properties, called the shape memory effect and pseudoelasticity, make these alloys interesting for many industrial applications, such as the aerospace, transportation, and medical industries.

This example model shows the SMA behavior under uniaxial loading. Three studies are performed: a boundary load sweep shows the pseudoelasticity effect at different fixed temperatures, a prescribed displacement sweep shows the pseudoelasticity effect with a partial unloading–partial loading loop, and the shape memory effect is portrayed after increasing the temperature.

PHASE TRANSFORMATION

Shape memory alloys exist in two crystalline structures. Austenite is present are higher temperature, and Martensite is present at lower temperatures. The transformation from one to the other phase is the cause of the specific behavior of SMAs. In the general case, the material contains both austenite and martensite phases, due to an incomplete transformation. The transformation rate is denoted by the martensite volume fraction ξ.

According to Ref. 1, the evolution of the transformation is calculated by Kuhn-Tucker conditions on the general thermodynamics force conjugated to ξ

$$\begin{array}{lll} \text{if } \dot{\xi} > 0; & \pi - Y \le 0; & (\pi - Y)\dot{\xi} \le 0 \\ \text{if } \dot{\xi} < 0; & -\pi - Y \le 0; & (-\pi - Y)\dot{\xi} \le 0 \end{array}$$

where

$$\pi(\sigma, T, \xi) = \sigma{:}\Lambda + \frac{1}{2}\sigma{:}\Delta S{:}\sigma + \rho C_p\left((T - T_0) - T\ln\left(\frac{T}{T_0}\right)\right) + \rho\Delta s_0 T - \rho\Delta u_0 - \frac{\partial f}{\partial \xi}$$

Model Definition

A cylinder made of nickel–titanium (NiTi) alloy is submitted to axial tension. This is modeled by a rectangle in a 2D axisymmetric geometry. The **Shape Memory Alloy** node is set to use the Lagoudas SMA model. Table 1 lists the material properties for each phase.

TABLE 1: MATERIAL PROPERTIES OF EACH PHASE.

Material property	**Value**
Young's modulus, austenite	55 GPa
Young's modulus, martensite	46 GPa
Poisson's ratio	0.33
Heat capacity at constant pressure	400 J/(kg.K)
Density	6500 kg/m^3

Table 2 lists the phase transformation parameters.

TABLE 2: PHASE TRANSFORMATION PARAMETERS.

Parameter	**Value**
Martensite start temperature, M_s	245 K
Martensite finish temperature, M_f	230 K
Austenite start temperature, A_s	270 K
Austenite finish temperature, A_f	280 K
Slope of martensite limit curve, C_M	7.4 MPa/K
Slope of austenite limit curve, C_A	7.4 MPa/K
Maximum transformation strain	0.056

The first study runs a parametric sweep for four different prescribed temperatures. Two of the temperatures (328 K, 308 K) are above the austenite finish temperature, one temperature (276 K) lies between the austenite start and finish temperature, and one temperature (260 K) is below the austenite start temperature. The boundary load is a loading–unloading cycle up to 850 MPa for first study and up to 300 MPa for the third study, see Figure 1.

The second study runs a parametric sweep on the axial displacement with a constant temperature of 298 K. The prescribed displacement is applied on one face in order to reach an axial strain value of 0.07. After reaching this maximum value, the axial displacement is decreased down to 40% of its value, then increased up to 80% the maximum, to finally decrease to 0, see Figure 2.

The third study runs at a constant temperature of 260 K, which is below the austenite start temperature. After one mechanical cycle according to Figure 1, the temperature is uniformly increased to 300 K to achieve the reverse transformation to austenite following Figure 3.

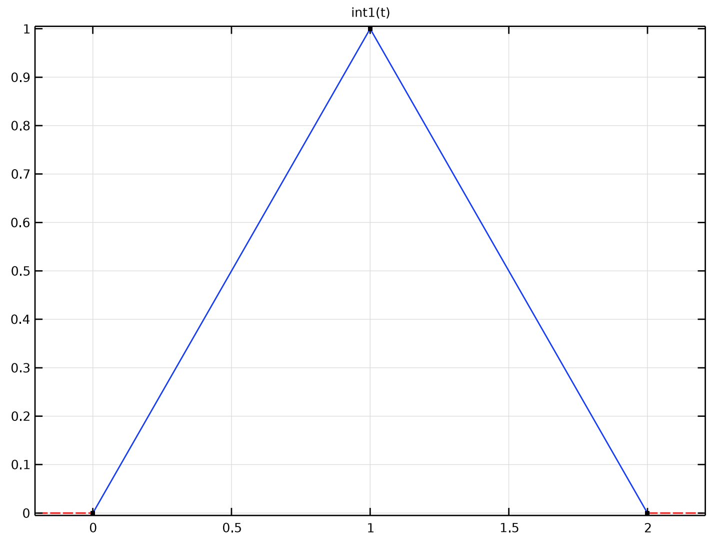

Figure 1: Boundary load for the first and third studies.

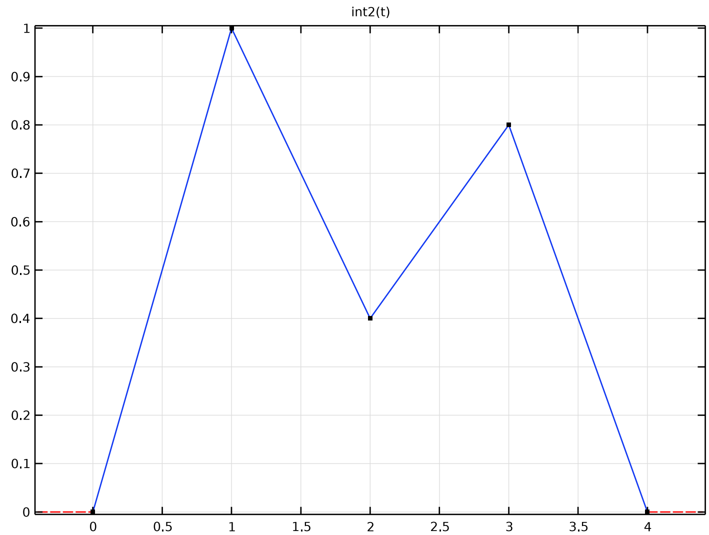

Figure 2: Prescribed displacement for the second study.

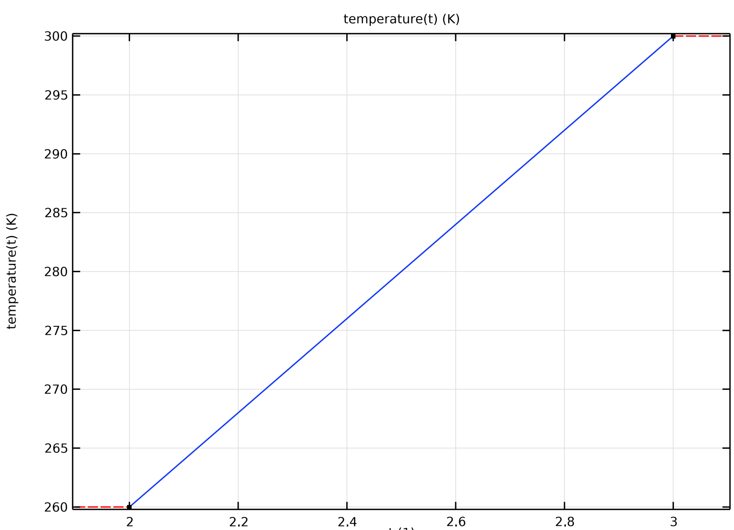

Figure 3: Temperature increase in the third study.

PSEUDOELASTICITY — SINGLE LOADING CYCLE

In Figure 4, the curves for high temperatures (328 K and 308 K) show the pseudoelasticity effect: the stress-strain relation is linear up to a temperature-dependent stress limit. Above this limit, the martensite transformation begins, resulting in a material with lower stiffness. The transformation is complete when reaching the maximum strain, the alloy microstructure is then 100% martensite, and the tangent stiffness is taken from the Young’s modulus of the martensite. During unloading, the reverse transformation occurs at a lower stress level than the stress limit for the forward transformation.

The axial stress–strain curve for a prescribed temperature of 276 K (between the austenite start and finish temperatures) shows that the forward and reverse transformations occur at lower stress levels. Also, the reverse transformation is not complete when the stress is completely released. This can be seen in Figure 5 where the forward transformation occurs sooner, that is, at lower stress level. The backward transformation starts later, and it is not complete at the end of the sweep. At a lower temperature, the forward transformation occurs at a lower stress level, and the revers transformation does not even start, resulting in residual strains.

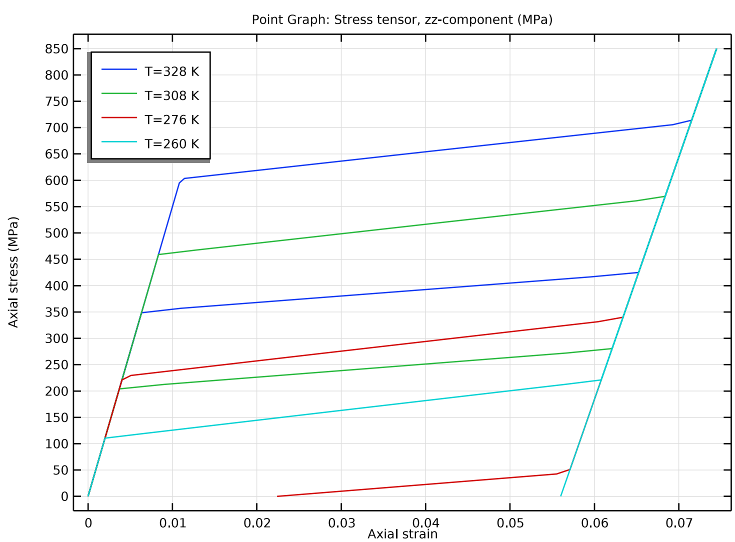

Figure 4: Stress versus strain curve at several temperatures.

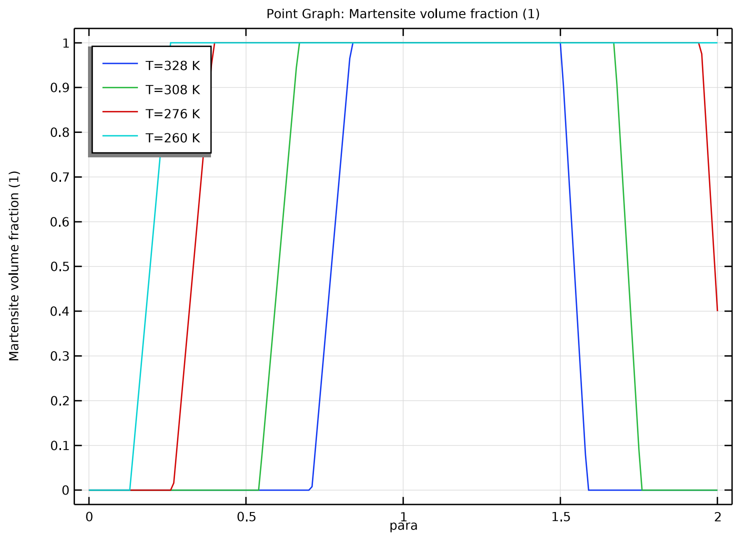

Figure 5: Evolution of martensite volume fraction at several temperatures.

PSEUDOELASTICITY — MULTIPLE LOADING CYCLES

The first loading step and the first unloading step are similar to the ones in the previous study. At a strain value of 0.028, the material is loaded again, which leads to a stress increase with high stiffness as seen in Figure 6. This part of the curve is represented by a plateau in Figure 7. When the yield stress limit is reached, the forward transformation continues and follows the same path as the first load. When the material is unloaded, the stress decreased, and the material undergoes the reverse transformation to pure austenite. Finally the stress is decreased to zero.

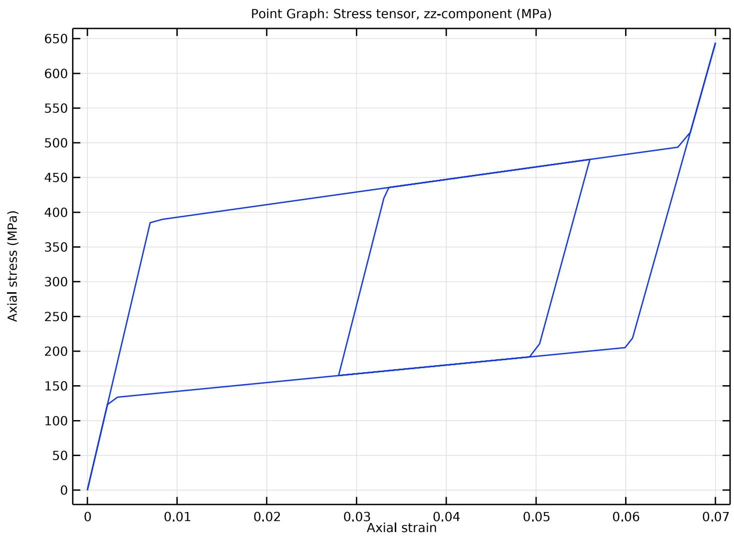

Figure 6: Stress versus strain curve for loading–unloading cycle with an internal loop.

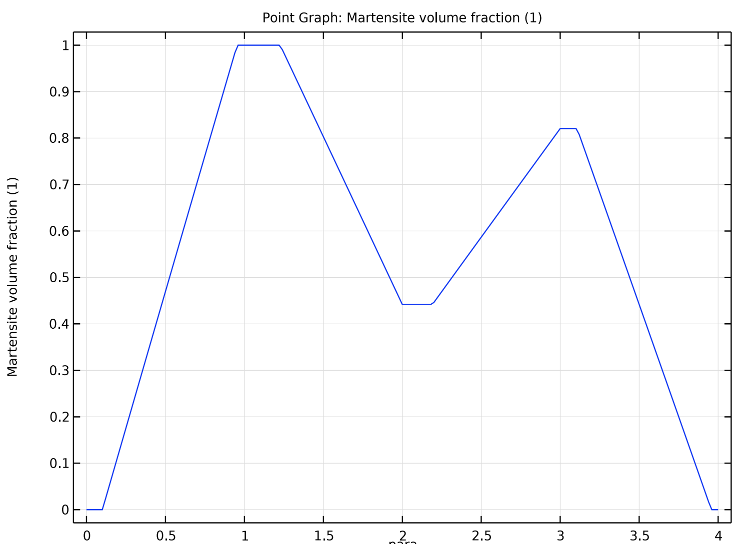

Figure 7: Evolution of martensite volume fraction during the loading–unloading cycle with an internal loop.

SHAPE MEMORY EFFECT

The mechanical loading–unloading cycle is the same as in the first study (260 K). At this point the residual strain is 0.056. Increasing the temperature from 260 K to 300 K decreases the residual strain to 0 as shown in Figure 8.

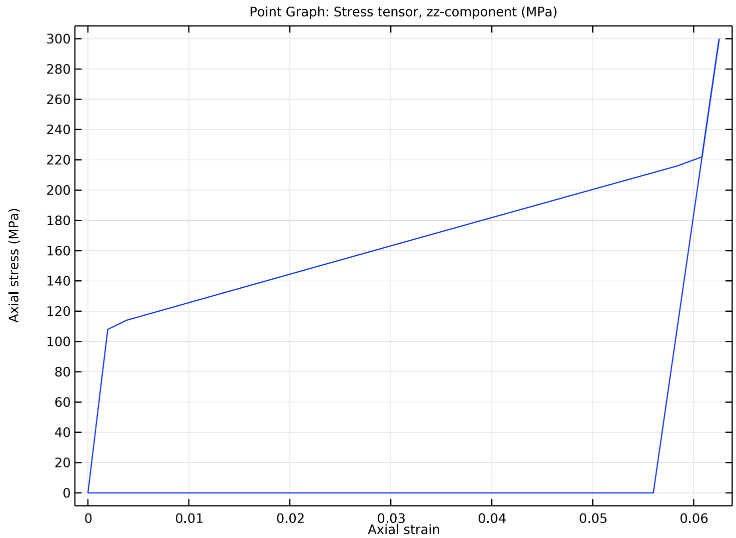

Figure 8: Stress versus strain curve showing the shape memory effect.

Reference

1. D.C. Lagoudas (ed.), *Shape Memory Alloys*, Springer 2008

Application Library path: `Nonlinear_Structural_Materials_Module/Shape_Memory_Alloys/uniaxial_loading_of_shape_memory_alloy`

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 Axisymmetric**.
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
T	298[K]	298 K	Applied temperature
para	0	0	Continuation parameter

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 **cm**.

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 6.
4 In the **Height** text field, type 20.

SOLID MECHANICS (SOLID)

Shape Memory Alloy 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>Shape Memory Alloy**.
2 In the **Settings** window for **Shape Memory Alloy**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All domains**.

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

Roller 1

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

2 Select Boundary 2 only.

Create an interpolation function to apply the load cycle.

DEFINITIONS

Interpolation 1 (int1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Functions>Interpolation**.

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

4 In the table, enter the following settings:

t	f(t)
0	0
1	1
2	0

5 Click **Plot**.

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	r
850[MPa]*int1(para)	z

MATERIALS

Material 1 (mat1)

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

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**
Reference temperature	T0	318	K	Lagoudas model
Poisson's ratio	nu	0.33	1	Basic
Young's modulus	E_A	55[GPa]	Pa	Austenite phase
Heat capacity at constant pressure	Cp_A	400	J/(kg·K)	Austenite phase
Young's modulus	E_M	46[GPa]	Pa	Martensite phase
Heat capacity at constant pressure	Cp_M	400	J/(kg·K)	Martensite phase
Martensite start temperature	TMs	245	K	Lagoudas model
Martensite finish temperature	TMf	230	K	Lagoudas model
Slope of martensite limit curve	CM	7.4e6	Pa/K	Lagoudas model
Austenite start temperature	TAs	270	K	Lagoudas model
Austenite finish temperature	TAf	280	K	Lagoudas model
Slope of austenite limit curve	CA	7.4e6	Pa/K	Lagoudas model
Maximum transformation strain	etrmaxLagoudas	0.056	1	Lagoudas model
Calibration stress level	sigmaStar	0	N/m²	Lagoudas model
Density	rho	6500	kg/m³	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 **Coarser**.

4 Click **Build All**.

STUDY: PSEUDOELASTICITY, SINGLE LOADING CYCLE

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

2 In the **Settings** window for **Study**, type `Study: Pseudoelasticity, Single Loading Cycle` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
T (Applied temperature)	328 308 276 260	K

Step 1: Stationary

1 In the **Model Builder** window, 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
para (Continuation parameter)	range(0,0.01,2)	

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study: Pseudoelasticity, Single Loading Cycle>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 Select the **Tuning of step size** check box.

6 In the **Model Builder** window, click **Study: Pseudoelasticity, Single Loading Cycle**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

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

RESULTS

Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)

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

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Pseudoelasticity, Single Loading Cycle/Parametric Solutions 1 (sol2)**.

Point Graph 1

1 Right-click **Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)** and choose **Point Graph**.

2 Select Point 4 only.

3 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>Stress>Stress tensor (spatial frame) - N/m²>solid.sGpzz - Stress tensor, zz-component**.

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

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>Strain tensor (material and geometry frames)>solid.eZZ - Strain tensor, ZZ-component**.

6 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Point** check box.

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

Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)

1 In the **Model Builder** window, click **Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)**.

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

3 Select the **x-axis label** check box.

4 Select the **y-axis label** check box.

5 In the **x-axis label** text field, type `Axial strain`.

6 In the **y-axis label** text field, type `Axial stress (MPa)`.

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

8 In the **Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)** toolbar, click **Plot**.

The resulting plot should look like Figure 4.

Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)

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

2 In the **Settings** window for **1D Plot Group**, type `Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Pseudoelasticity, Single Loading Cycle/Parametric Solutions 1 (sol2)**.

Point Graph 1

1 Right-click **Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)** and choose **Point Graph**.

2 Select Point 4 only.

3 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>Shape memory alloy>solid.xiGp_M - Martensite volume fraction - 1**.

4 Locate the **Legends** section. Select the **Show legends** check box.

5 Find the **Include** subsection. Clear the **Point** check box.

Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)

1 In the **Model Builder** window, click **Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)**.

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

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

4 In the **Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)** toolbar, click **Plot**.

The resulting plot should look like Figure 5.

Now perform a study with a partial unloading–loading cycle. First define the load function.

DEFINITIONS

Interpolation 2 (int2)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

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

3 In the table, enter the following settings:

t	f(t)
0	0
1	1
2	0.4
3	0.8
4	0

4 Click **Plot**.

SOLID MECHANICS (SOLID)

Prescribed Displacement 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in z direction** list, choose **Prescribed**.

5 In the u_{0z} text field, type `20[cm]*0.07*int2(para)`.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

STUDY: PSEUDOELASTICITY, MULTIPLE LOADING CYCLES

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

2 In the **Settings** window for **Study**, type `Study: Pseudoelasticity, Multiple Loading Cycles` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Pseudoelasticity, Multiple Loading Cycles** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate 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
para (Continuation parameter)	`range(0,0.02,4)`	

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.

8 Click **Disable**.

9 In the **Model Builder** window, click **Study: Pseudoelasticity, Multiple Loading Cycles**.

10 In the **Settings** window for **Study**, locate the **Study Settings** section.

11 Clear the **Generate default plots** check box.

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

Duplicate the existing plots to reproduce Figure 6 and Figure 7.

RESULTS

Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)

In the **Model Builder** window, under **Results** right-click **Stress vs. Strain (Pseudoelasticity, Single Loading Cycle)** and choose **Duplicate**.

Stress vs. Strain (Pseudoelasticity, Multiple Loading Cycles)

1 In the **Model Builder** window, under **Results** click **Stress vs. Strain (Pseudoelasticity, Single Loading Cycle) 1**.

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain (Pseudoelasticity, Multiple Loading Cycles)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Pseudoelasticity, Multiple Loading Cycles/Solution 7 (sol7)**.

4 Locate the **Legend** section. Clear the **Show legends** check box.

5 In the **Stress vs. Strain (Pseudoelasticity, Multiple Loading Cycles)** toolbar, click **Plot**.

Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)

In the **Model Builder** window, right-click **Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle)** and choose **Duplicate**.

Martensite Volume Fraction (Pseudoelasticity, Multiple Loading Cycles)

1 In the **Model Builder** window, under **Results** click **Martensite Volume Fraction (Pseudoelasticity, Single Loading Cycle) 1**.

2 In the **Settings** window for **1D Plot Group**, type `Martensite Volume Fraction (Pseudoelasticity, Multiple Loading Cycles)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Pseudoelasticity, Multiple Loading Cycles/Solution 7 (sol7)**.

4 Locate the **Legend** section. Clear the **Show legends** check box.

5 In the **Martensite Volume Fraction (Pseudoelasticity, Multiple Loading Cycles)** toolbar, click **Plot**.

To show the shape memory effect you need to apply a sweep on the mechanical loading, then on the temperature.

DEFINITIONS

Interpolation 3 (int3)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

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

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

4 In the table, enter the following settings:

t	f(t)
2	260
3	300

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

Function	Unit
temperature	K

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

Argument	Unit
t	1

7 Click **Plot**.

SOLID MECHANICS (SOLID)

Duplicate the **Shape Memory Alloy 1** node to apply a temperature defined by the interpolation function.

Shape Memory Alloy 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Shape Memory Alloy 1** and choose **Duplicate**.

Shape Memory Alloy 2

1 In the **Model Builder** window, click **Shape Memory Alloy 2**.

2 In the **Settings** window for **Shape Memory Alloy**, locate the **Model Input** section.

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

Boundary Load 2

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	r
300[MPa]*int1(para)	z

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

STUDY: SHAPE MEMORY EFFECT

1. In the **Model Builder** window, click **Study 3**.
2. In the **Settings** window for **Study**, type `Study: Shape Memory Effect` in the **Label** text field.

Step 1: Stationary

1. In the **Model Builder** window, under **Study: Shape Memory Effect** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate 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
para (Continuation parameter)	`range(0,0.02,2) range(2.05,0.05,3)`	

6. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
7. In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.
8. Click **Disable**.
9. In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)> Prescribed Displacement 1**.
10. Click **Disable**.
11. In the **Model Builder** window, click **Study: Shape Memory Effect**.
12. In the **Settings** window for **Study**, locate the **Study Settings** section.
13. Clear the **Generate default plots** check box.
14. In the **Home** toolbar, click **Compute**.

Duplicate the third plot to reproduce Figure 8.

RESULTS

Stress vs. Strain (Pseudoelasticity, Multiple Loading Cycles)

In the **Model Builder** window, under **Results** right-click **Stress vs. Strain (Pseudoelasticity, Multiple Loading Cycles)** and choose **Duplicate**.

Stress vs. Strain (Shape Memory Effect)

1 In the **Model Builder** window, under **Results** click **Stress vs. Strain (Pseudoelasticity, Multiple Loading Cycles) 1**.

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain (Shape Memory Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Shape Memory Effect/ Solution 8 (sol8)**.

4 In the **Stress vs. Strain (Shape Memory Effect)** toolbar, click **Plot**.

You need to disable the following nodes in the studies if you want to run them again.

STUDY: PSEUDOELASTICITY, SINGLE LOADING CYCLE

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Pseudoelasticity, Single Loading Cycle** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)> Prescribed Displacement 1**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Shape Memory Alloy 2**.

7 Click **Disable**.

8 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 2**.

9 Click **Disable**.

STUDY: PSEUDOELASTICITY, MULTIPLE LOADING CYCLES

1 In the **Model Builder** window, under **Study: Pseudoelasticity, Multiple Loading Cycles** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Shape Memory Alloy 2**.

4 Click **Disable**.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 2**.

6 Click **Disable**.