Created in COMSOL Multiphysics 6.2

Uniaxial Loading of a Shape Memory Alloy Using the Souza–Auricchio Model

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 applications in, for example, 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
- The shape memory effect is portrayed after increasing the temperature

This model is similar to the Uniaxial Loading of a Shape Memory Alloy model, but it uses the Souza–Auricchio model instead of the Lagoudas model.

The Souza–Auricchio model introduces a transformation strain tensor calculated from the limit function $\boldsymbol{F} = |\sigma_{tr}| - \sigma_0$, where the conjugated thermodynamic stress associated with the transformation strain has the following expression (Ref. 1–3):

$$\sigma_{tr} = \mathrm{dev}(\sigma) - H_k \varepsilon_{tr} - \beta \langle T - T^* \rangle \frac{\varepsilon_{tr}}{|\varepsilon_{tr}|} - \frac{\partial I}{\partial \varepsilon_{tr}}$$

The transformation strain, ε_{tr}, is computed from the flow rule

$$\dot{\varepsilon}_{tr} = \lambda_p \frac{\partial F}{\partial \sigma_{tr}}$$

and the Kuhn–Tucker condition, where λ_p is the plastic multiplier.

Model Definition

A cylinder made of nickel-titanium (NiTi) alloy is submitted to axial tension. The cylinder is modeled by a rectangle in a 2D axisymmetric geometry. The **Shape Memory Alloy** node is set to use the Souza–Auricchio model.

The phase transformation parameters are calculated from the parameters used for the Lagoudas model portrayed in the Uniaxial Loading of a Shape Memory Alloy model (Ref. 4).

TABLE 1: MATERIAL PARAMETERS, LAGOUDAS MODEL.

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

Note that the Lagoudas model contains more parameters than the Souza–Auricchio model. The latter model assumes that the slope of limit curves to martensite and austenite domains are equal, that is, $C_M = C_A$. In this example, the slope of limit curve β equals the slope of martensite limit curve C_M.

Similarly, the Souza–Auricchio model assumes that the width of the transition zones are equal, that is, the temperature difference $A_f - A_s$ and $M_s - M_f$ are the same, which is not the case in the Lagoudas model. The reference temperature is then defined as

$$T^* = \frac{M_s + A_f}{2} = T_\sigma - \frac{\sigma_{Ms} + \sigma_{Af}}{2\beta}$$

In this example, the hardening modulus H_k is derived from the difference in martensite start and finish temperatures, $M_s - M_f$, and the slope of martensite limit curve C_M.

$$H_k = \frac{C_M(M_s - M_f)}{\varepsilon_{tr,\,max}} = \frac{\sigma_{Mf} - \sigma_{Ms}}{\varepsilon_{tr,\,max}}$$

where $\varepsilon_{tr,max}$ is the maximum transformation strain.

It is also possible to map Souza–Auricchio parameters to the stresses at the start and finish transformations, see Ref. 3. The material properties are listed in Table 2.

TABLE 2: MATERIAL PROPERTIES OF PHASE CHANGE.

Material Parameter	**Symbol**	**Lagoudas parameters**	Ref. 4	**Value**
Maximum transformation strain	$\varepsilon_{tr,max}$	$\varepsilon_{tr,max}$	ε_L	0.056
Slope of limit curve	β	C_M	$\frac{\Delta\sigma}{\Delta T}$	7.4 MPa/K
Reference temperature	T^*	$\frac{M_s + A_f}{2}$	$T_\sigma - \frac{\sigma_{Ms} + \sigma_{Af}}{2\beta}$	265.5 K
Elastic domain radius	σ_0	$\frac{C_M(A_f - M_s)}{2}$	$\frac{\sigma_{Ms} - \sigma_{Af}}{2}$	129.5 MPa
Hardening modulus	H_k	$\frac{C_M(M_s - M_f)}{\varepsilon_{tr,\,max}}$	$\frac{\sigma_{Mf} - \sigma_{Ms}}{\varepsilon_{tr,\,max}}$	1.982 GPa

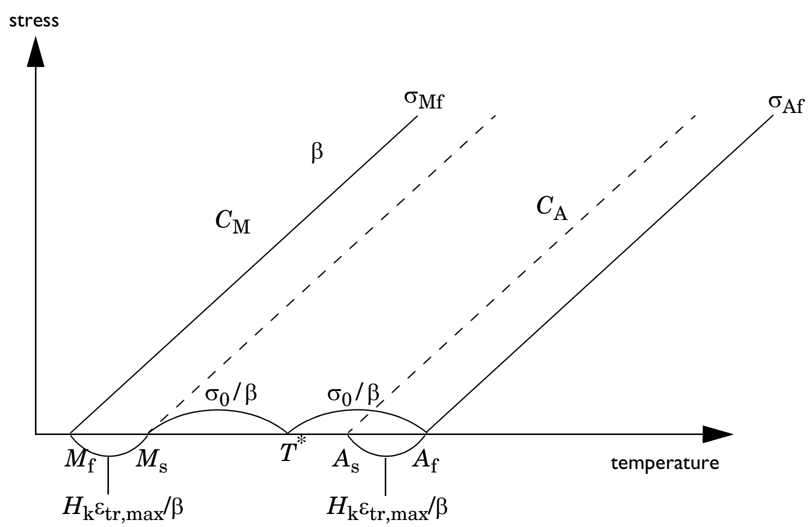


Figure 1: Phase diagram showing phase transformation parameters for the Lagoudas and Souza–Auricchio models.

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 temperatures, and one temperature (260 K) is below the austenite start temperature. The boundary load is a loading–unloading cycle up to 800 MPa for first study and up to 300 MPa for the third study, see Figure 2.

The second study runs a parametric sweep on the axial displacement for 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 3.

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 2, the temperature is uniformly increased to 300 K to achieve the reverse transformation to austenite following Figure 4.

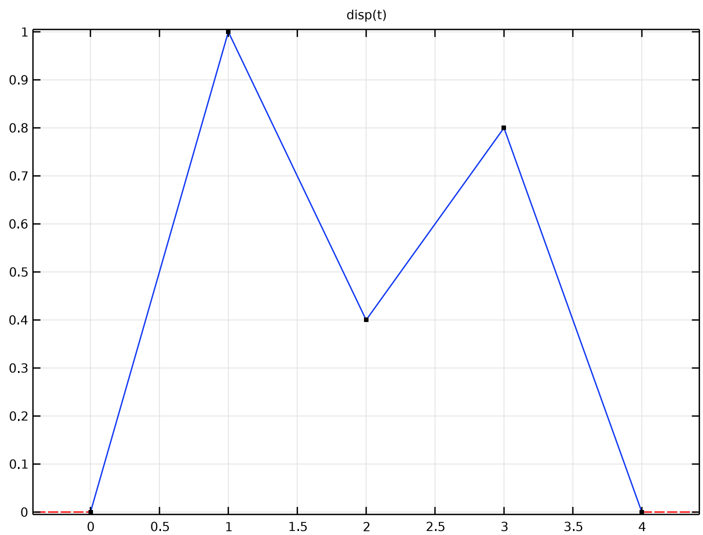


Figure 2: Boundary load for the first and third studies.

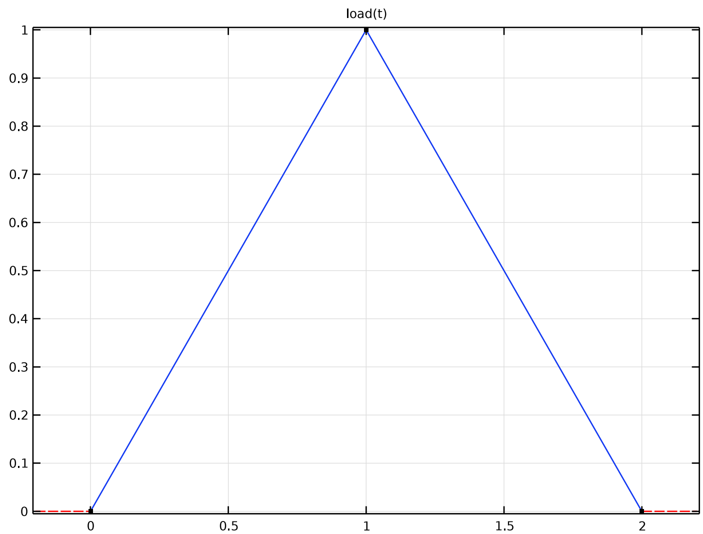


Figure 3: Prescribed displacement for the second study.

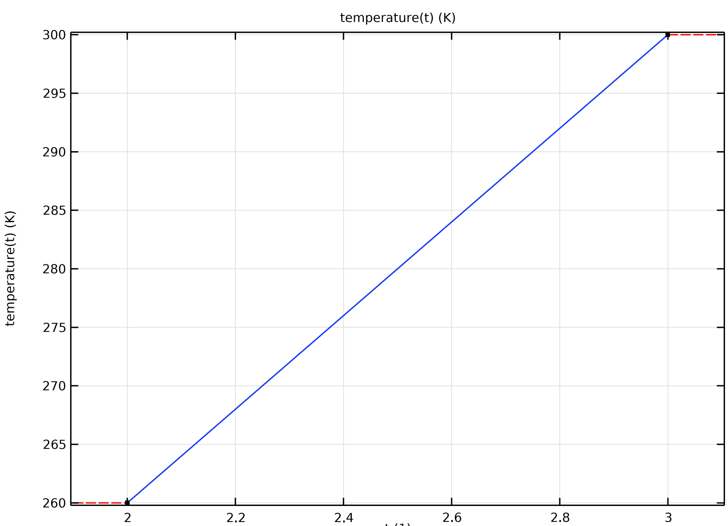


Figure 4: Temperature increase in the third study.

PSEUDOELASTICITY — SINGLE LOADING CYCLE

In Figure 5, 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, at which the microstructure is 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 6 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 cycle. 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 7 sums up all these observations on a phase diagram: at higher temperatures the forward transformation occurs at higher stress, and the reverse transformation is complete. At lower temperatures the forward transformation occurs at lower stress and the reverse transformation does not start, or it is incomplete.

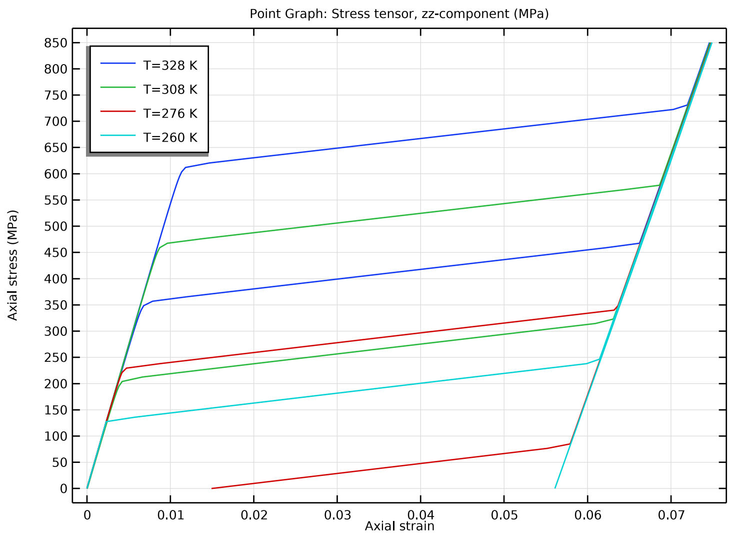


Figure 5: Stress vs strain curve at several temperatures.

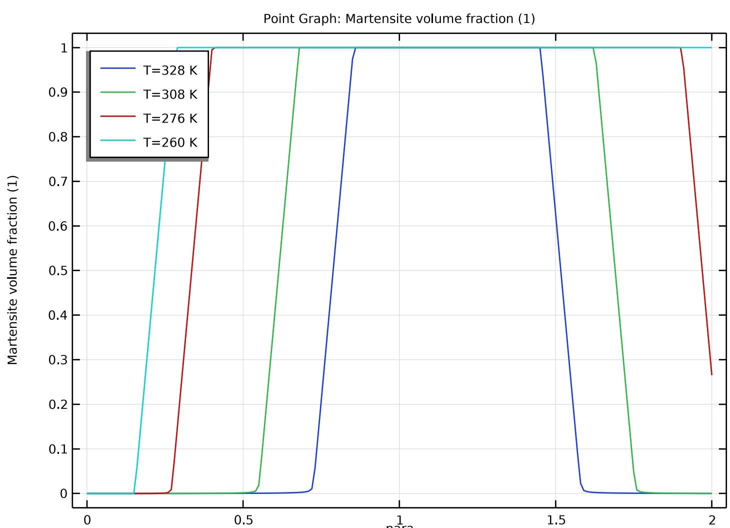
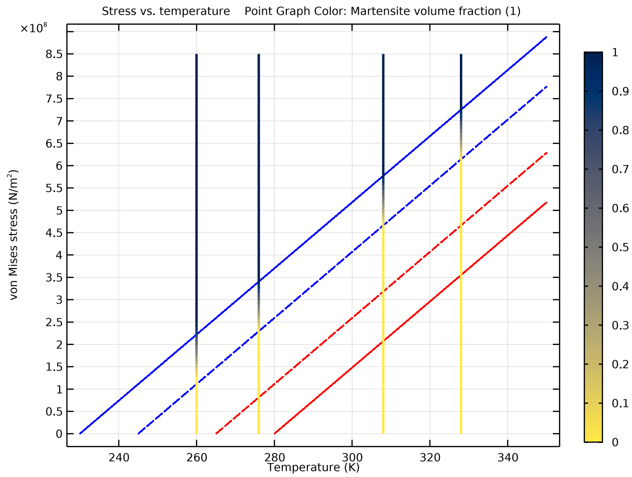


Figure 6: Evolution of martensite volume fraction at several temperatures.



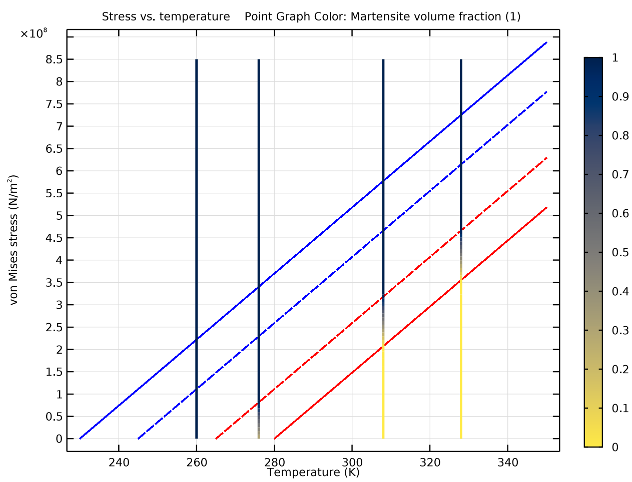


Figure 7: Stress–temperature paths reported on the phase diagram for the four temperatures. Top: loading. Bottom: unloading.

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 level of 0.028, the material is loaded again, which leads to a stress increase with high stiffness, see Figure 8. This part of the curve is represented by a plateau in Figure 9. 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 decreases and the material undergoes the reverse transformation to pure austenite. Finally the stress is decreased to zero. The four stages can be seen in Figure 10.

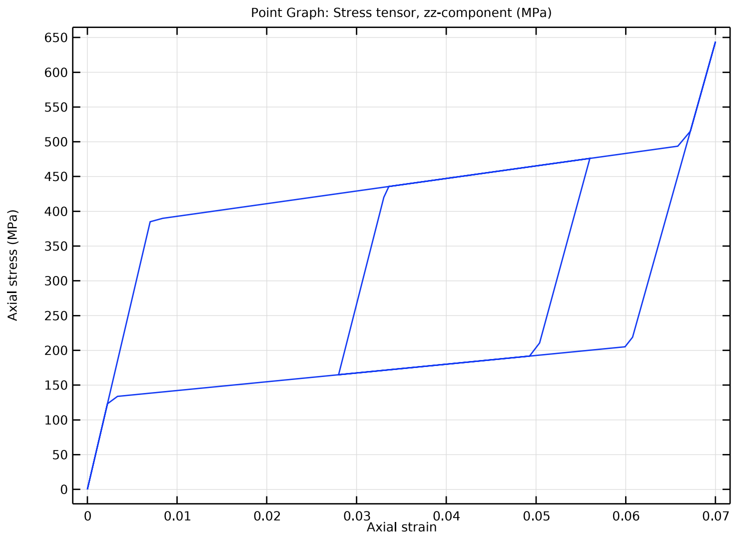


Figure 8: Stress versus strain curve for loading-unloading cycle with an internal loop.

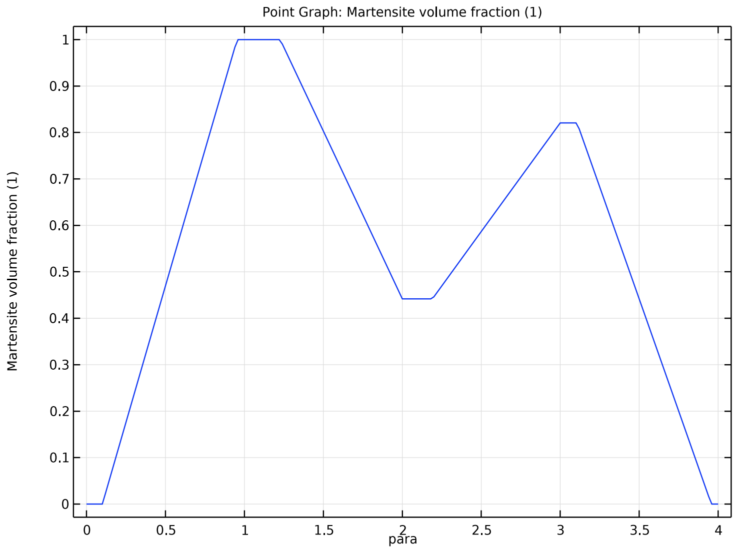


Figure 9: Evolution of martensite during the loading-unloading cycle with an internal loop.

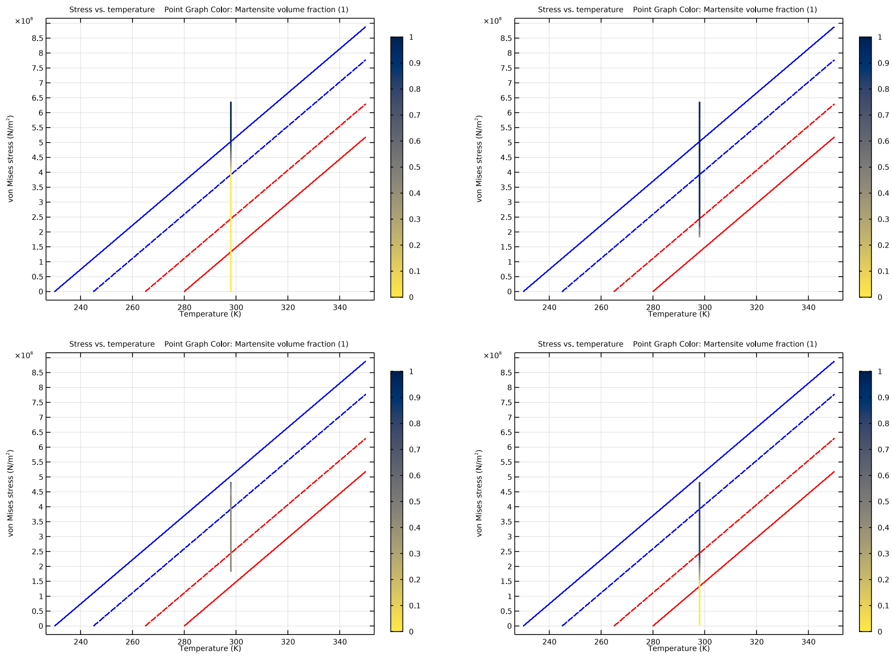


Figure 10: Stress–temperature paths reported on the phase diagram. Top left: loading. Top right: partial unloading. Bottom left: partial loading. Bottom right: unloading.

SHAPE MEMORY EFFECT

The mechanical loading-unloading cycle is the same as in the first study (temperature of 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 11. This is also visible on the phase diagram, Figure 12: after unloading the material does not reach the austenite transition zone, then temperature increase is needed to recover the austenite phase.

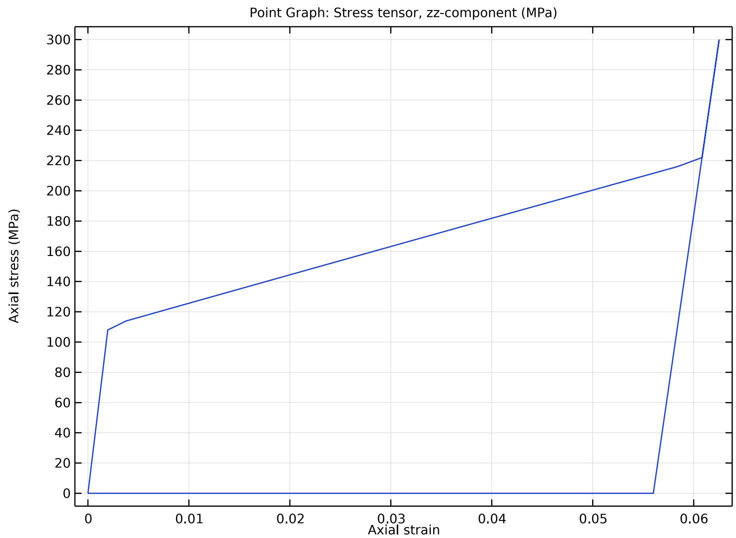


Figure 11: Stress versus strain curve showing the shape memory effect.

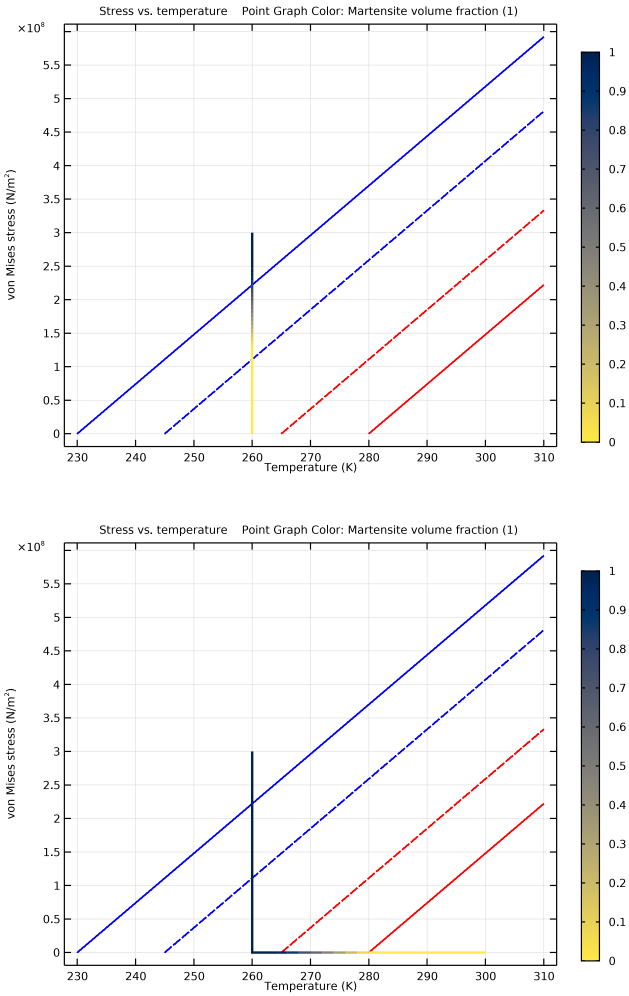


Figure 12: Stress–temperature paths reported on the phase diagram. Top: loading. Bottom: unloading and heating.

References

1. A. Souza and others, "Three-dimensional model for solids undergoing stress-induced phase transformations," *European J. Mech. A Solids*, vol. 17, pp. 789–806, 1988.

2. F. Auricchio and L. Petrini, “A three-dimensional model describing stress-temperature induced solid phase transformations. Part I: solution algorithm and boundary value problems,” *Int. J. Numer. Meth. Eng.*, vol. 61, pp. 807–836. 2004.

3. F. Auricchio and others, “SMA numerical modeling versus experimental results: Parameter identification and model prediction capabilities,” *J. Mater. Eng. Perform.*, vol. 18, pp. 649–654, 2009

4. D. Lagoudas (ed.), *Shape Memory Alloys: Modeling and Engineering Applications*, Springer, 2008.

Application Library path: `Nonlinear_Structural_Materials_Module/Shape_Memory_Alloys/uniaxial_loading_of_shape_memory_alloy_souza_auricchio`

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`.

5 Locate the **Shape Memory Alloy** section. From the **Material model** list, choose **Souza–Auricchio**.

6 Find the **Phase transformation** subsection. From the **Specify** list, choose **Temperature**.

Roller 1

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

2 Select Boundary 2 only.

Boundary Load 1

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

Create an interpolation function to apply the load cycle.

DEFINITIONS

Load

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

2 In the **Settings** window for **Interpolation**, type `Load` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `load`.

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 **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Boundary Load 1**.

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]*load(para)	z

MATERIALS

NiTi Alloy

1 In the **Materials** toolbar, click **Blank Material**.

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

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

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.33	1	Basic
Young's modulus	E_A	55[GPa]	Pa	Austenite phase

Property	Variable	Value	Unit	Property group
Young's modulus	E_M	`46[GPa]`	Pa	Martensite phase
Slope of limit curve	beta	`7.4[MPa/K]`	Pa/K	Souza-Auricchio model
Maximum transformation strain	etrmaxAuricchio	`0.056`	1	Souza-Auricchio model
Martensite start temperature	TMs_SA	`245[K]`	K	Souza-Auricchio model
Martensite finish temperature	TMf_SA	`230[K]`	K	Souza-Auricchio model
Austenite finish temperature	TAf_SA	`280[K]`	K	Souza-Auricchio 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 **Minimum step size** text field, type `0.0001`.

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

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

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

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

RESULTS

Stress vs. Strain

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` 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** and choose **Point Graph**.

2 Select Point 1 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

1 In the **Model Builder** window, click **Stress vs. Strain**.

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** toolbar, click **Plot**.

The resulting plot should look like Figure 5.

Martensite Volume Fraction

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` 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** and choose **Point Graph**.

2 Select Point 1 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

1 In the **Model Builder** window, click **Martensite Volume Fraction**.

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** toolbar, click **Plot**.

The resulting plot should look like Figure 6.

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: Pseudoelasticity, Single Loading Cycle/Parametric Solutions 1 (sol2)>Solid Mechanics>Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1)**.

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

RESULTS

Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1)

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

2 From the **Parameter selection (para)** list, choose **From list**.

3 In the parameter values list, select all solution steps between 0 and 1.

4 In the **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1)** toolbar, click **Plot** to get the paths during stress loading (Figure 7, top).

5 In the parameter values list, select all solution steps between 1 and 2.

6 In the **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1)** toolbar, click **Plot** to get the paths during stress unloading (Figure 7, bottom).

7 From the **Parameter selection (para)** list, choose **All**.

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

DEFINITIONS

Displacement

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

2 In the **Settings** window for **Interpolation**, type `Displacement` in the **Label** text field.

3 Locate the **Definition** section. In the **Function name** text field, type `disp`.

4 In the table, enter the following settings:

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

5 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*disp(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.

Solution 7 (sol7)

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

2 In the **Model Builder** window, expand the **Solution 7 (sol7)** node.

3 In the **Model Builder** window, expand the **Study: Pseudoelasticity, Multiple Loading Cycles>Solver Configurations>Solution 7 (sol7)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

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

6 In the **Minimum step size** text field, type `0.0001`.

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

Duplicate the two first plots to reproduce Figure 8 and Figure 9.

RESULTS

Martensite Volume Fraction, Stress vs. Strain

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress vs. Strain** and **Martensite Volume Fraction**.
2 Right-click and choose **Duplicate**.

Stress vs. Strain 1

1 In the **Model Builder** window, click **Stress vs. Strain 1**.
2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3 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 1** toolbar, click **Plot**.

Martensite Volume Fraction 1

1 In the **Model Builder** window, click **Martensite Volume Fraction 1**.
2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3 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 1** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.
2 In the tree, select **Study: Pseudoelasticity, Multiple Loading Cycles/Solution 7 (sol7)>Solid Mechanics>Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1)**.
3 Click **Add Plot** in the window toolbar.

RESULTS

Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1) 1

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
2 From the **Parameter selection (para)** list, choose **From list**.
3 In the parameter values list, select all solution steps between 0 and 1.
4 In the **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1) 1** toolbar, click **Plot** to get the path during stress loading (Figure 10, top).

5 You can do the same three times, selecting all solution steps between 1 and 2, between 2 and 3, and between 3 and 4 to plot path during partial unloading, partial loading, and total unloading, respectively.

6 From the **Parameter selection (para)** list, choose **All**.

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

DEFINITIONS

Temperature

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

2 In the **Settings** window for **Interpolation**, type `Temperature` in the **Label** text field.

3 Locate the **Definition** section. 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 In the T 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]*load(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.

Solution 8 (sol8)

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

2 In the **Model Builder** window, expand the **Solution 8 (sol8)** node.

3 In the **Model Builder** window, expand the **Study: Shape Memory Effect> Solver Configurations>Solution 8 (sol8)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

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

6 In the **Minimum step size** text field, type `0.0001`.

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

Duplicate the second stress-strain plot to reproduce Figure 11.

RESULTS

Stress vs. Strain 1

In the **Model Builder** window, under **Results** right-click **Stress vs. Strain 1** and choose **Duplicate**.

Stress vs. Strain 1.1

1 In the **Model Builder** window, click **Stress vs. Strain 1.1**.

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

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

4 In the **Stress vs. Strain 1.1** toolbar, click **Plot**.

ADD PREDEFINED PLOT

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

2 In the tree, select **Study: Shape Memory Effect/Solution 8 (sol8)>Solid Mechanics> Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2)**.

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

RESULTS

Parameters

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`dset4_solid_sma2_Tmax`	`310[K]`	310 K	`Maximum temperature of phase diagram`

Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2)

1 In the **Model Builder** window, click **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2)**.

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

3 From the **Parameter selection (para)** list, choose **From list**.

4 In the parameter values list, select all solution steps between 0 and 1.

5 In the **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2)** toolbar, click **Plot** to get the path during stress loading (Figure 12, top).

6 In the parameter values list, select all solution steps between 2 and 3.

7 In the **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2)** toolbar, click **Plot** to get the path during stress loading (Figure 12, bottom).

Martensite Volume Fraction, Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1), Stress vs. Strain

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress vs. Strain**, **Martensite Volume Fraction**, and **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1)**.

2 Right-click and choose **Group**.

Pseudoelasticity, Single Loading Cycle

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

Martensite Volume Fraction 1, Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1) 1, Stress vs. Strain 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress vs. Strain 1**, **Martensite Volume Fraction 1**, and **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 1) 1**.
2 Right-click and choose **Group**.

Pseudoelasticity, Multiple Loading Cycles

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

Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2), Stress vs. Strain 1.1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress vs. Strain 1.1** and **Shape Memory Alloy Phase Diagram (Shape Memory Alloy 2)**.
2 Right-click and choose **Group**.

Shape Memory Effect

In the **Settings** window for **Group**, type `Shape Memory Effect` in the **Label** text field.

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**.