

Coupled Finite Element Simulation of Shape Memory Bending Microactuator

Georgino Kaleng Tshikwand¹, Lena Seigner², Frank Wendler¹ and Manfred Kohl²

The following document will explain how to utilize the implemented coupled material model for 3D simulations of the displacement-thermal-electrical behavior of a component. The provided input files could also be used to verify the presented results in the paper.

To run a coupled simulation in ABAQUS (Standard) using User subroutine UMAT describing the material model, the following steps need to be followed:

- **Property Module**

While creating the material property, Under the General tab, select User Material. Select the User Material type "Thermomechanical".

This type gives the opportunity to define the "mechanical constants" to be used accessed as in the UMAT by the variable PROPS in the given order from 1 to n number of required parameters; in our model, we require 18 parameters. Under the "Thermal Constants" the parameters to be Accessed in the subroutine UMATH through the variable PROPS, here the first parameter PROP(1) is the thermal conductivity and the second PROP(2) the heat capacity.

It can seen that the two subroutines UMAT and UMATH are combined in the same .f file and Abaqus call them at each material calculation point. For the mechanical constants, the following parameters, As arranged in the paper (see Appendix C), need to be provided, most of these are obtained from experimental testing results:

PROP(1) = σ_{scr} the critical start stress ;
PROP(2) = σ_{fcr} the critical finish stress;
PROP(3) = C^{PM} the Clausius Clapeyron coefficient from the parent to the martensite phase;
PROP(4) = C^{MP} the Clausius Clapeyron coefficient from the martensite to the parent phase;
PROP(5) = ε_{max}^{tr} The maximum transformation strain;
PROP(6) = E^A the young modulus of the Austenite phase;
PROP(7) = E^M the young modulus of the Martensite phase;
PROP(8) = μ^A the poisson ratio of the Austenite phase;
PROP(9) = μ^M the poisson ratio of the Martensite phase;
PROP(10) = M_s martensite start temperature;
PROP(11) = M_f martensite finish temperature;
PROP(12) = A_s Austenite start temperature;
PROP(13) = A_f Austenite finish temperature;
PROP(14) = c^A Heat capacity of the austenite phase;
PROP(15) = c^M Heat capacity of the martensite phase;
PROP(16) = Δs_0 the difference between the effective specific entropy at the reference state of martensite s_0^M and austenite s_0^A ;

PROP(17) = Δu_o the difference between the effective specific internal energy at the reference state of martensite u_o^M and austenite u_o^A ;
 PROP(18) = ρ material density.
 These parameters are obtained using the diagrams and the equations below,

$$\sigma_{scr} = \sigma_{Ms} - C^{PM} * (T_{ref} - M_s) \quad Eq \ 1$$

$$\sigma_{fsc} = \sigma_{Mf} - C^{MP} * (T_{ref} - M_s) \quad Eq \ 2$$

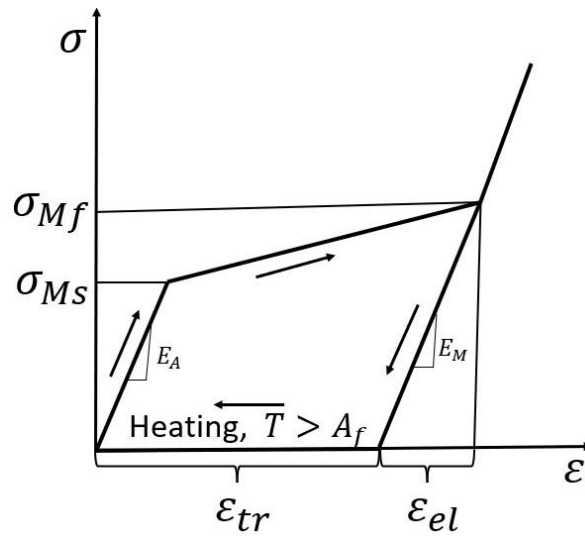


Figure 1 Stress-strain diagram of SMA (One-way shape memory effect).

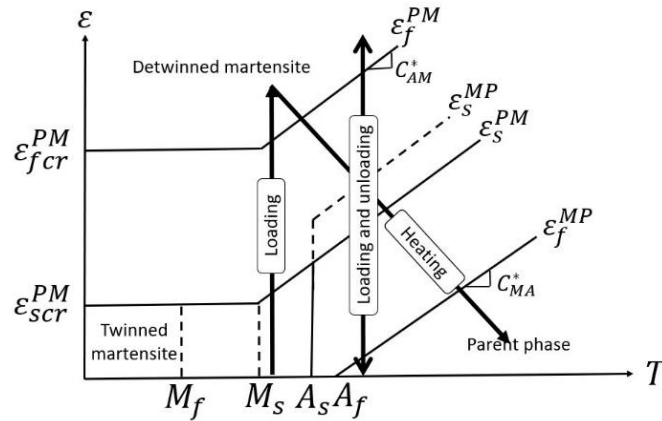


Figure 2 Modified SMA phase diagram adapted from [1]

- **Step module**

Here, the type of analysis is set. For a simulation coupling the displacement, thermal and electrical fields, an analysis of the type Coupled thermal-electrical-structural is chosen.

- **Interaction Module**

Here, data relative to the free convection are provided. The convection coefficient and the environment temperature are provided through the surface film condition option from the interaction manager.

- **Mesh Module**

Here, data relative to the mesh size and type are provided. For a fully coupled, bending dominated simulation, we recommend the Standard, quadratic 3D thermal Electrical Structural elements. A reduced integration method is also recommended.

- **Running the simulation on a Linux based system**

The following commands are executed to run simulations with the provided input files and material model:

Note: For the compilation of the Fortran code we used an Intel compiler (intel/2016.2.181). Abaqus 2018 was used for the simulations.

- 1. Example of tensile test at 299 K**

```
>> abaqus job=run input=tensile299.inp cpus=15  
user=TSHIKWAND_JABER_FULLY_COUPLED_STRAIN_BASED_SMA_MODEL.f
```

- 2. Example of bendint simulation at 299 K**

```
>> abaqus job=run input=bending299.inp cpus=15  
user=TSHIKWAND_JABER_FULLY_COUPLED_STRAIN_BASED_SMA_MODEL.f
```

The link below provided details on executing Abaqus jobs from the terminal:

<https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usb/default.htm?startat=pt01ch03s02abx02.html>

- **Reference**

[1] Jaber, M.B., Smaoui, H., Terriault, P.: Finite element analysis of a shape memory alloy three-dimensional beam based on a finite strain description. Smart materials and structures 17(4), 045005 (2008)