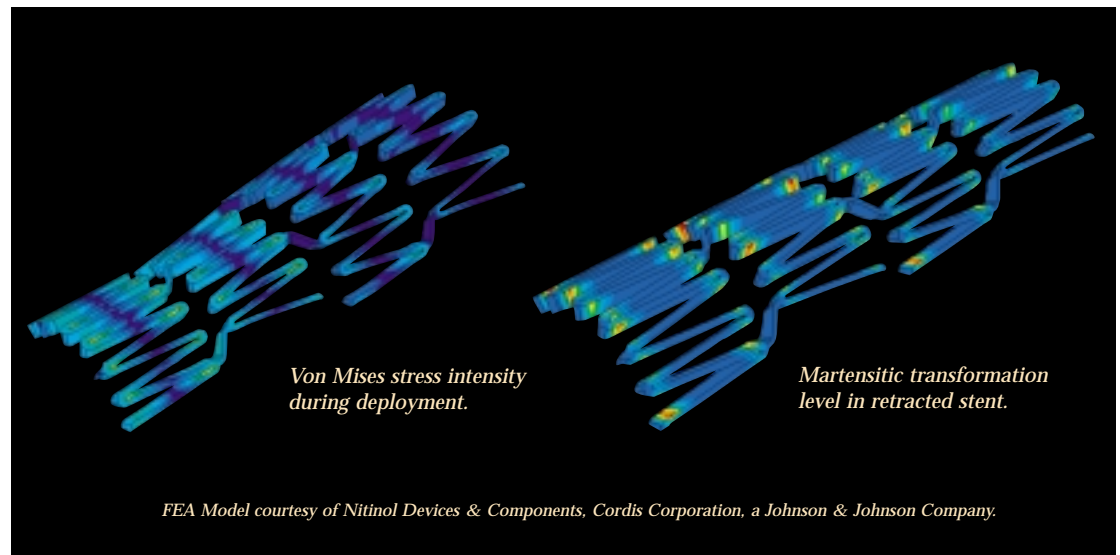


ABAQUS/Answers

ANSWERS TO COMMON ABAQUS QUESTIONS SPRING 2001

CONTENTS

1 Simulation of Implantable Nitinol Stents



Simulation of Implantable Nitinol Stents

Medical technology is advancing rapidly, particularly in the development of advanced medical implants. Traditionally new products were developed by prototyping and evaluating; however, this process is very time consuming and often does not fully reveal potential failures. Finite element modeling and analysis greatly reduces testing and time to market by allowing the designer to simulate product performance in advance of any prototyping.

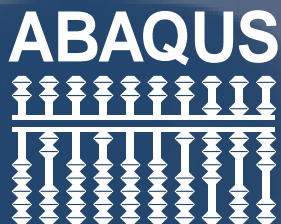
The superelastic and shape memory properties of Nitinol, a nickel-titanium alloy, along with its biocompatibility and fatigue properties, have made the material attractive for cardiovascular stents. However, it is a complex material and difficult to process. FEA hastens time to market of Nitinol products by reducing the number of design iterations required, hence expediting the design process.

Superelastic alloy material behavior

Nitinol is an extremely flexible metal alloy that can undergo very large deformations without losing the ability

to recover its original shape upon unloading. At rest, the material presents itself in an austenite phase, which behaves linear elastically. Upon loading, this austenite phase transforms into a martensite phase, which is also linear elastic; however, the elasticity of each phase has different constants. The transformation produces a substantial amount of strain and is triggered by stress over a relatively narrow range. Upon unloading, the transformation is reversible. However, the stress levels at which such reversible transformation occurs are smaller than the stresses that were required to produce the original transformation. Because the material recovers its original shape upon unloading, it is described as elastic. In addition, because the transformation strains are large (on the order of 6%) compared to elastic strains in typical metals (on the order of 0.1%), the material is said to be superelastic.

If a reverse loading is applied (for example, in compression instead of tension), a similar behavior is observed, with the exception that the stress levels required to produce the transformations are higher, while the transformation strain is lower.



Hibbitt, Karlsson &
Sorensen, Inc.

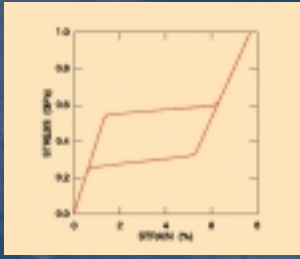
1080 Main Street
Pawtucket, RI
02860-4847 usa

phone 401.727.4200

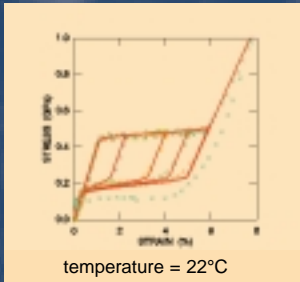
fax 401.727.4208

e-mail sales@abaqus.com

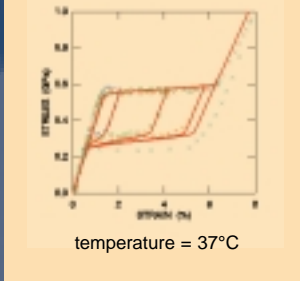
www.abaqus.com



Mechanical behavior of Nitinol.

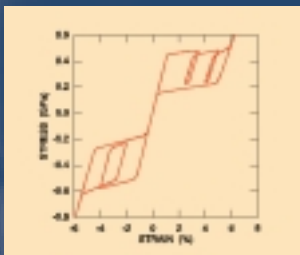


temperature = 22°C



temperature = 37°C

The ABAQUS algorithm for NiTi alloy modeling closely matches experimental measurements where successively larger strain loads are applied in the alloy at different temperatures.



Behavior of the model upon repeated loading and unloading of Nitinol.

Nitinol is a difficult material to characterize numerically. An approximation, capable of modeling the loading part of the model, is hyperelasticity, which is commonly used to represent rubber behavior. Other constitutive models have been proposed recently to address Nitinol's behavior. However, a thorough verification of these models has been difficult because the manufacturing requirements of Nitinol are complex. As a result, Nitinol is typically available as relatively thin wires and tubes, from which most of the testing data available are in the form of uniaxial data. It is relatively straightforward to produce a uniaxial model, but it is much more difficult to produce a model that represents Nitinol's three-dimensional stress-strain behavior.

At Hibbitt, Karlsson & Sorensen (West) a user material routine (UMAT/Nitinol) was written following the model proposed by Auricchio and Taylor. This theory is based on the concept of generalized plasticity and physical principles. The theory decomposes strain into two parts—a purely linear elastic component and a transformation component:

$$\Delta \epsilon = \Delta \epsilon^{\text{el}} + \Delta \epsilon^{\text{tr}}.$$

The austenite to twinned martensite transformation is driven by the resolution of shear forces, and it takes place within a range of stress levels that are characteristic of the material.

$$\Delta \epsilon^{\text{tr}} = a \Delta \zeta \frac{\partial F}{\partial \sigma},$$

$$F^S \leq F \leq F^F,$$

where ζ is the fraction of martensite and F is a transformation potential. The same is true for the reverse transformation but at different stress levels. The intensity of the transformation follows a stress potential law:

$$\Delta \zeta = f(\sigma, \zeta) \Delta F.$$

Any change in stress direction produces a reorientation of the martensite with negligible additional effort. Changes in temperature produce a shift in the stress levels at which the transformations take place. This shift is linear in temperature. Because there is a volume increase

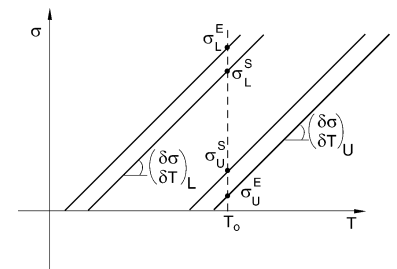
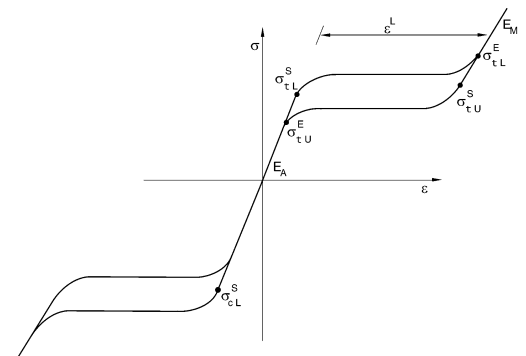
associated with the transformation, it requires less stress to produce the transformation in tension and more in compression. This is modeled with a linear Drucker-Prager approach for the transformation potential:

$$F = \bar{\sigma} - p \tan \beta + CT,$$

with $\bar{\sigma}$ being the Mises equivalent stress, p the pressure stress, and T the temperature.

A rule of mixtures is used to implement the change in linear elasticity from the austenite phase into the martensite phase.

In addition to the usual stresses and strains, the UMAT routine tracks variables specific to the model. These variables include the distribution of the fraction of martensite, transformation strains, and equivalent stresses and strains. These are points in the uniaxial tensile curve into which material points that have a three-dimensional state, specific fractions of martensite, and a specific loading/unloading history are mapped. The material data required by the model are obtained from straightforward observations of uniaxial tests in terms of loading, unloading, reverse loading, and temperature effects. The calibration consists of 13 values, as shown below.



Stent analysis

Stents are cylindrical metal mesh tubes made of materials such as Nitinol and inserted into blood vessels to counteract the effects associated with vascular diseases, such as narrowing of blood vessels due to plaque build up.

The present analysis considers both the manufacturing process and deployment of a stent. One of the key issues in medical implants is device lifetime or, in engineering terms, fatigue life. Finite element analysis provides quantitative measures of stress and strain needed to make fatigue estimates for these devices. This allows the optimization of designs and allows prediction of the device's life.

The large elastic strains possible in Nitinol reduce the risk of damage to the stent both during delivery into the body and due to accidents while in operation.

One manufacturing process of Nitinol stents starts from a thin tube into which a pattern is electromachined. The FEA model is built from this machined tube. Since the pattern repeats itself, only a part of the stent with appropriate symmetry is modeled. The stent is expanded to its nominal dimensions, typically at a diameter many times larger than the original tube diameter. While loaded, the stent is annealed to provide its new unloaded configuration. The stent is then crimped from the outside and inserted into the delivery system (usually a system of catheter tubes). Once inside a patient's body, the delivery system pushes the stent out of its containment. Once delivered, the response to blood pressure pulsing loads determines the fatigue life of the stent.

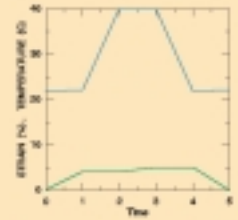
The calculation reveals stress-strain concentrations during manufacturing, deployment, and service loading. Auxiliary information is also available from the calculation; for example, how much martensitic transformation has occurred and, therefore, how close the design is to the limits of the material flexibility.

Summary

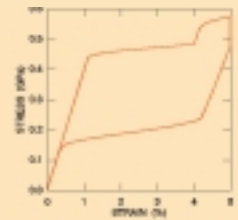
Finite element analysis is a method well suited to the needs of the product designer. The method allows designers to iterate designs on a computer before prototypes are made. The finite element method can

solve mechanical problems involving complex material behavior such as those of Nitinol alloys.

Complexity of stent designs and of the actions taken to deploy them requires sophistication in the finite element tool used. Proper material behavior is just one requirement. Advanced modeling capabilities such as contact with both rigid tools and other flexible bodies, large deformations, localized buckling, material removal, and simulation of complex loading sequences are needed. These basic robust capabilities in ABAQUS make the code particularly suitable for stent design.

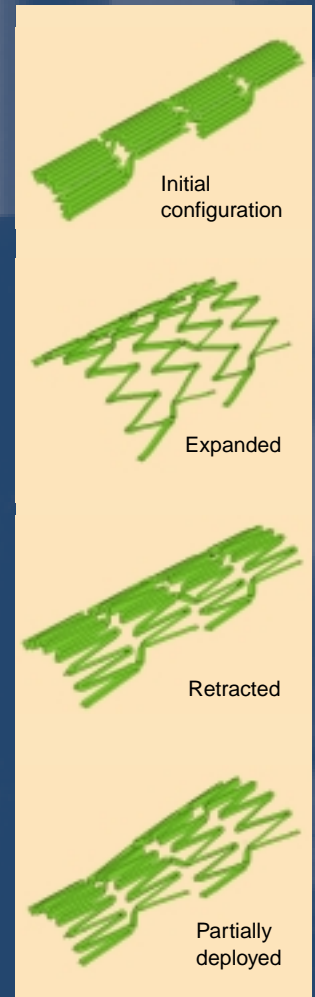


Loading in terms of temperature and strain.



Stress-strain curve results based on loading shown above.

Behavior of the model subjected to an 18°C temperature range while strains are applied.



Stent model.