Simulation of Implantable Nitinol Stents

Summary

The superelastic, shape memory, biocompatibility, and fatigue properties of Nitinol, a nickel-titanium alloy, have made the material attractive for medical devices such as cardiovascular stents. However, it is a complex material and difficult to process. Finite element modeling of Nitinol devices such as stents reduces testing and time-to-market by allowing the designer to simulate the stent manufacturing and deployment processes. The constitutive models for superelastic alloys are available as user subroutine libraries for both Abaqus/Standard and Abaqus/Explicit.

Background

Self-expanding micro devices (stents) are cylindrical metal mesh tubes, made of materials such as Nitinol, that are inserted into blood vessels to counteract the effects associated with vascular diseases, such as narrowing of the blood vessels due to plaque build-up.

Nitinol stents can be manufactured from thin tubes into which a pattern is electromachined. After a sequence of operations, a stent is mounted on a catheter and inserted into a blood vessel. After being released by this delivery system (Figure 1), the stent self-expands and exerts a radial force on the wall of the blood vessel.
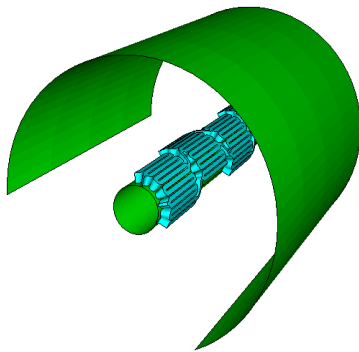

Figure 1: Stent and delivery system.

The behavior of Nitinol is extremely complex, as can be seen from its uniaxial behavior shown in Figure 2. The key characteristic of Nitinol is its superelastic material behavior, making it 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. When loaded beyond a certain stress, the austenite phase
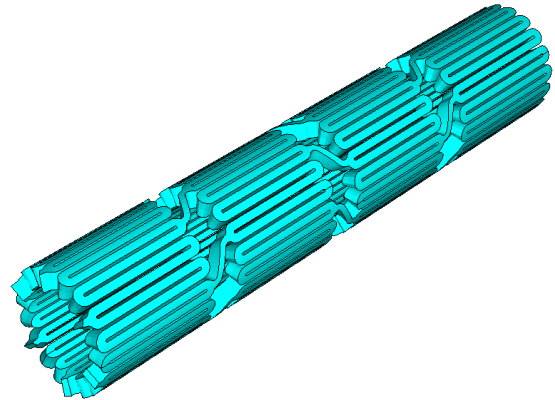
Key Abaqus Features and Benefits

- Constitutive models, provided as user subroutines, for accurately modeling the behavior of superelastic alloys such as Nitinol.

- Advanced modeling capabilities such as contact, large deformations, and annealing.

transforms into a martensite phase. The transformation produces a substantial amount of strain, which on unloading is reversible. Since the transformation strains are large (of the order of 6%) compared to elastic strains in typical metals, the material is said to be superelastic. Further loading beyond superelastic limit reveals plastic behavior in martensite. The material data required to calibrate the Abaqus material model can be obtained from the uniaxial behavior (Figure 2) in terms of loading, unloading, reverse loading, and temperature effects.
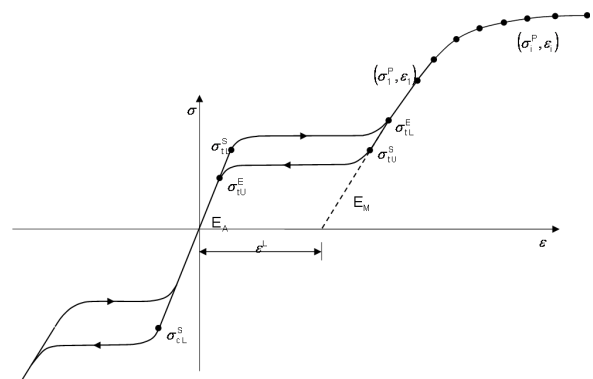

Figure 2: Uniaxial behavior of Nitinol.

Finite Element Analysis Approach

The manufacturing process of Nitinol stents starts from a thin tube in which a pattern is micro- machined. The finite element model is built from this machined tube. Since the pattern repeats itself symmetrically, only a part of the stent with appropriate symmetry boundary conditions needs to be considered (Figure 3).

The stent is expanded to its nominal dimensions, typically at a diameter much larger than the original tube diameter (Figure 4). The fraction of martensite after expansion is shown in Figure 5. It can be seen that the fraction of martensite as well as the stresses are higher at the corners of the stent. The stent is then annealed to provide its new unloaded configuration. It is then crimped from the outside (Figure 6) and inserted into the delivery system (usually a system of catheter tubes). Once inside the blood vessel, the delivery system pushes the stent out of its containment (Figure 7), expanding to exert radial forces on the blood vessel. The tools are considered rigid and cylindrical. In this simplified Nitinol stent simulation, only a fraction of the length of the stent is modeled and the expansion is unconstrained when the tool is released (Figure 7). For the simulation to be more realistic, the artery has to be included in the model and the full length of the stent needs to be modeled.
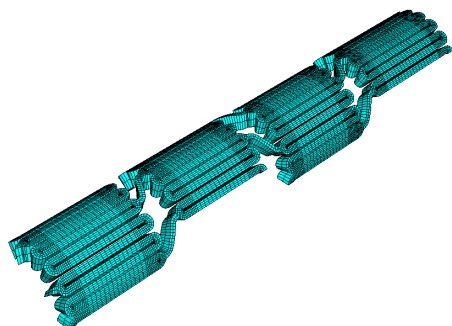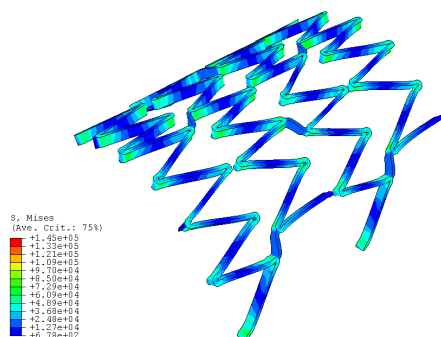

Figure 5: Fraction of transformed martensite.
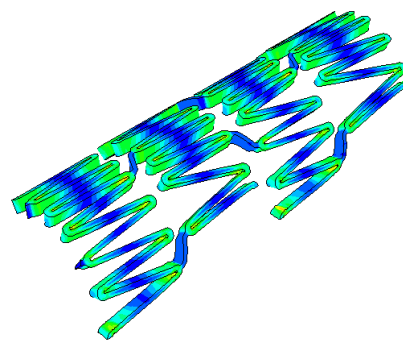

Figure 6: Crimped stent.

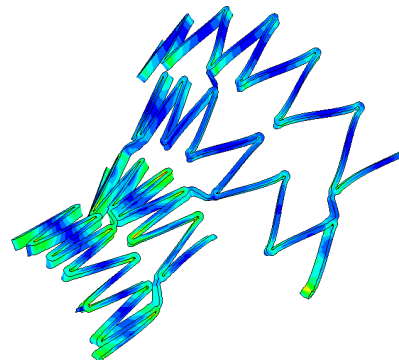*Figure 3: FE model of stent.*

Figure 7: Partially deployed stent.

Results and Conclusions

Nitinol exhibits extremely complex behavior and can be difficult to process; however, finite element modeling can hasten the time-to-market by reducing the design iterations required.

Finite element modeling can be used to reveal the stress or strain concentrations during manufacturing as well as deployment of the stent. This allows for optimization of stent designs. Additional pulsating loads after deployment allow prediction of the device life. The simulation can also reveal the amount of martensitic transformation that has taken place in the stent and, therefore, how close the design is to the limits of the material flexibility.

Figure 4: Expanded stent.

References

1. Rebelo, N., N. Walker, and H. Foadian, "Simulation of Implantable Nitinol Stents," 2001 Abaqus Users' Conference Proceedings.

2. Auricchio, F., and R. L. Taylor, "Shape Memory-Alloys: Modeling and Numerical Simulations of the Finite-Strain Superelastic Behavior," Computer Methods in Applied Mechanics and Engineering, vol. 143, pp. 175–194, 1997.

3. Auricchio F., R. L. Taylor, and J. Lubliner, "Shape Memory-Alloys: Macromodeling and Numerical Simulations of the Superelastic Behavior," Computer Methods in Applied Mechanics and Engineering, vol. 146, pp. 281–312, 1997.

Abaqus References

For additional information on the capabilities and techniques outlined above, see the following references to the Abaqus 6.11 documentation:

- Analysis User's Manual
  - "Annealing procedure," Section 6.12.1
  - "Defining contact pairs in Abaqus/Standard," Section 34.3.1
  - "Defining contact pairs in Abaqus/Explicit," Section 34.5.1
- User Subroutines Reference Manual
- Abaqus Answer 1658, "UMAT and VUMAT Routines for the Simulation of Nitinol"

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

3DS SIMULIA