ABAQUS

Lecture 3

Materials

Overview

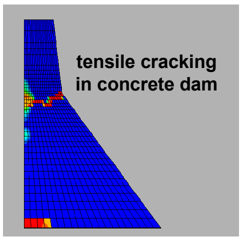
- Introduction
- Metals
- Rubber Elasticity
- Concrete
- Additional Materials

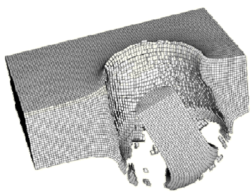
ABAQUS

Introduction

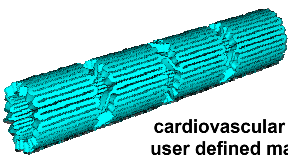
Introduction

- **ABAQUS has an extensive material library that can be used to model most engineering materials, including:**
  - Metals
  - Rubbers
  - Concrete
  - Damage and failure
  - Fabrics
  - Hydrodynamics
  - User defined

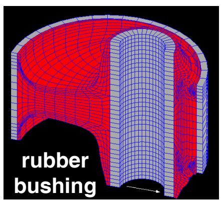




failure and erosion



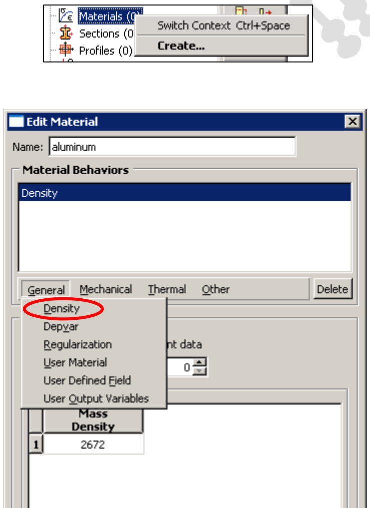
cardiovascular stent
user defined material
(Nitinol)



Introduction

- **Mass density**
  - In ABAQUS/Explicit a nonzero mass density must be defined for all elements.
  - Exceptions:
    - Fully constrained rigid bodies do not require a mass.
    - Mass density for hydrostatic fluid elements is defined as a fluid density.

```
*MATERIAL, NAME=aluminum
*DENSITY
2672.,
...
```



Introduction

- **Material damping**
  - Most models do not require material damping.
    - Energy dissipation mechanisms—dashpots, inelastic material behavior, etc.—are often included as part of the basic model.
  - Models that do not include other energy dissipation mechanisms, may require some general damping.
    - For example, a linear system with chattering contact.
    - ABAQUS provides Rayleigh damping for these situations.
  - There are two Rayleigh damping factors:
    - α for mass proportional damping and
    - β for stiffness proportional damping.
  - With these factors specified, the damping matrix C is added to the system:

$$C = \alpha M + \beta K.$$

Introduction

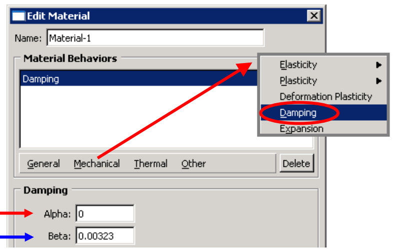
– For each natural frequency of the system, ω_α, the effective damping ratio is

$$\xi\left(\omega_\alpha\right) = \frac{\alpha}{2\omega_\alpha} + \frac{\beta\omega_\alpha}{2}.$$

– Thus, mass proportional damping dominates when the frequency is low, and stiffness proportional damping dominates when the frequency is high.

– Recall that increasing damping reduces the stable time increment.

```
*MATERIAL, NAME = ...
*DAMPING, ALPHA=α, BETA=β
```



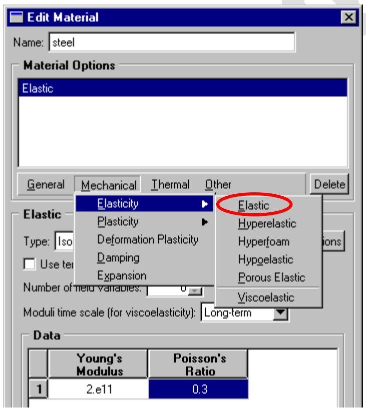
 ABAQUS

ABAQUS

Metals

Metals

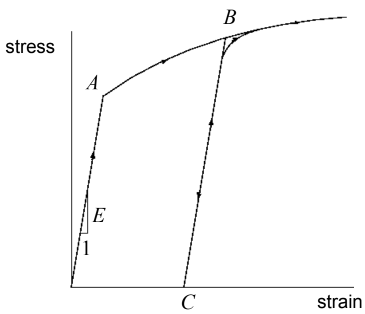
- **Elasticity**
  - The elastic response of metals can be modeled with either linear elasticity or an equation-of-state model.
  - Linear elasticity
    - Elastic properties can be specified as isotropic or anisotropic.
    - Elastic properties may depend on temperature (θ) and/or predefined field variables (f_i).
    - Linear elasticity should not be used if the elastic strains in the material are large.
  - The equation-of-state model is discussed later in the *Additional Materials* section.




```
*Material, name=steel
*Elastic
2.e11, 0.3
```


Metals

- **Metal plasticity overview**
  - Plasticity theories model the material's mechanical response under ductile nonrecoverable deformation.
  - A typical stress-strain curve for a metal is shown below.



Uniaxial stress-strain data for a metal

Features of the stress-strain curve:

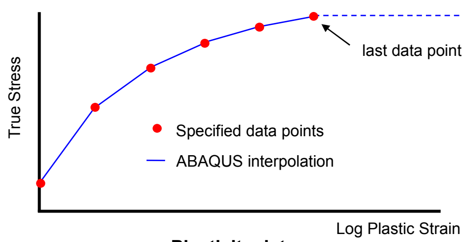
- Initially linear elastic
- Plastic yield begins at *A*
- Strain reversed at *B*
  - Material immediately recovers its elastic stiffness
- Complete unloading at *C*
  - Material has permanently deformed
- Reloading
  - Yield at, or very close to, *B*

Metals

– For most metals:
  - The yield stress is a small fraction, typically 1/10% to 1%, of the elastic modulus, which implies that the elastic strain is never more than this same fraction.
  - The elasticity can be modeled quite accurately as linear.

Metals

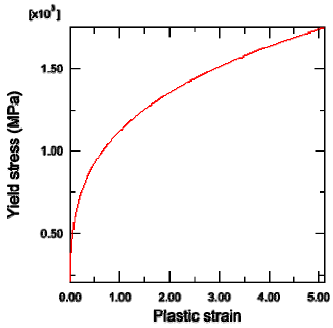
- **Classical metal plasticity**
  – The Mises yield surface is used in ABAQUS to model *isotropic* metal plasticity.
    - The plasticity data are defined as *true stress* vs. *logarithmic plastic strain*.
      – ABAQUS assumes no work hardening continues beyond the last entry provided.



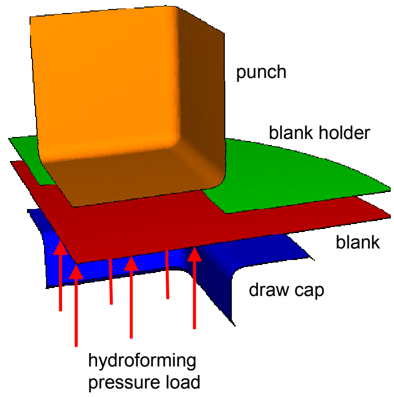
Plasticity data

Metals

– Example: Hydroforming of a box – Mises plasticity model



Blank plasticity data



Exploded view of initial configuration

Metals

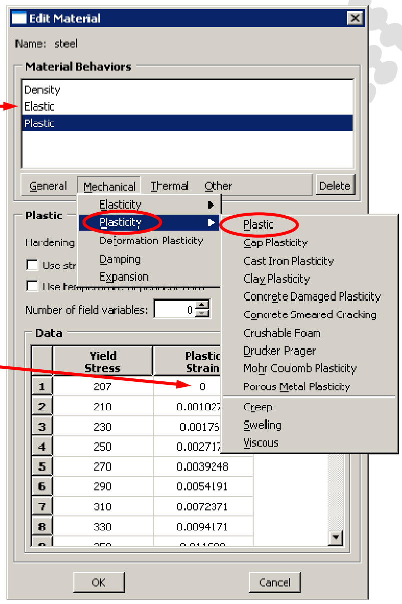
– Example (cont'd): Hydroforming of a box – Mises plasticity model

```
*Material, name=steel
*Density
 7.85e-09,
*Elastic
194000., 0.29
*Plastic
 207., 0.
 210., 0.0010279
 230., 0.001763
 250., 0.0027177
 270., 0.0039248
 :
 :
```

Linear elasticity

Plastic strain at initial yield = 0.0

True stress and log plastic strain



Metals

– In ABAQUS/Explicit, the table giving values of yield stress as a function of plastic strain (or any other material data given in tabular form) should be specified using equal intervals on the plastic strain axis.

- If this is not done, ABAQUS will *regularize* the data to create such a table with equal intervals.
  – The table lookups occur frequently in ABAQUS/Explicit and are most economical if the interpolation is from regular data.
- It is not always desirable to regularize the input data so that they are reproduced exactly in a piecewise linear manner;
  – in some cases this would require in an excessive number of data subdivisions.
- If ABAQUS/Explicit cannot regularize the data within a given tolerance using a reasonable number of intervals, an error is issued.

Metals

– Hill's yield potential is an extension of the Mises yield function used to model *anisotropic* metal plasticity:

- A reference yield stress (σ_0) is defined using the Mises plasticity definition syntax.
- Anisotropy is introduced through the definition of stress ratios:

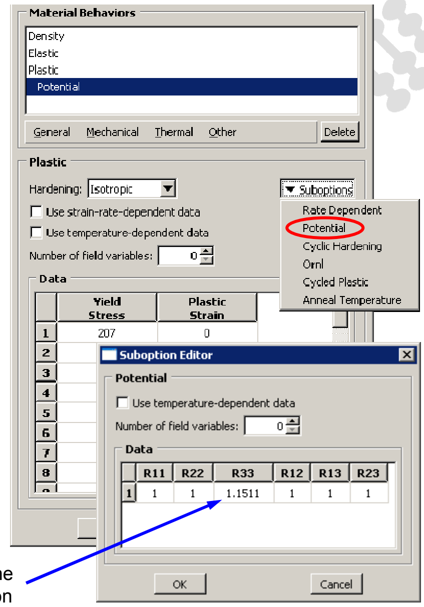
$$R_{11} = \frac{\bar{\sigma}_{11}}{\sigma_0}, \qquad R_{22} = \frac{\bar{\sigma}_{22}}{\sigma_0}, \quad \ldots$$

  – The R_{ij} values are determined from pure uniaxial and pure shear tests.
- This model is suitable for cases where the anisotropy has already been induced in the metal.
  – It is not suitable for situations in which the anisotropy develops with the plastic deformation.

Metals

– Example (cont'd): Hydroforming of a box – Hill's plasticity model

```
*Material, name=steel
*Density
 7.85e-09,
*Elastic
194000., 0.29
*Plastic
 207., 0.
 210., 0.0010279
 230., 0.001763
 250., 0.0027177
 270., 0.0039248
 :
 :
*Potential
1.0, 1.0, 1.1511, 1.0, 1.0, 1.0
```

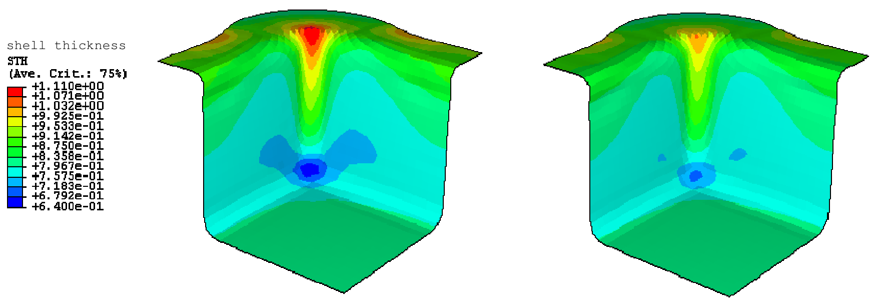


increased strength in the blank thickness direction

Metals

– Example (cont'd): Hydroforming of a box

- The effect of the anisotropy on the thickness is readily apparent, as the increased strength in the thickness direction results in less thinning of the blank.



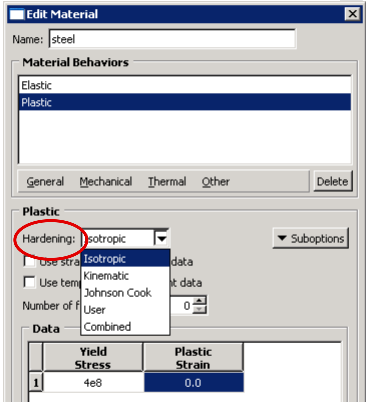
Isotropic (Mises plasticity)

Anisotropic (Hill's plasticity)

Effect of transverse anisotropy on blank thickness

Metals

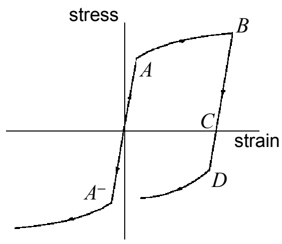
- **ABAQUS/Explicit offers four hardening options:**
  - Isotropic hardening (default).
    - The yield stress increases (or decreases) uniformly in all stress directions as plastic straining occurs.



 ABAQUS

Metals

- Linear kinematic hardening.
  - This is used in cases where simulation of the Bauschinger effect is relevant.
  - Applications include low cycle fatigue studies involving small amounts of plastic flow and stress reversals.
- Combined nonlinear isotropic/kinematic hardening.
  - This model is more general than the linear model
    - It will give better predictions.
    - However, it requires more detailed calibration.
  - This is typically used in cases involving cyclic loading.



The Bauschinger effect
$(D < B)$

 ABAQUS

Metals

– Johnson-Cook hardening.

- The Johnson-Cook plasticity model is suitable for high-strain-rate deformation of many materials, including most metals.
- This model is a particular type of Mises plasticity that includes analytical forms of the hardening law and rate dependence.
- It is generally used in adiabatic transient dynamic simulations.
- The elastic part of the response can be either linear elastic or defined by an equation of state model with linear elastic shear behavior.
- It is only available in ABAQUS/Explicit.

Metals

– The Johnson-Cook yield stress is of the form:

$$\bar{\sigma} = \left[A + B\left(\bar{\varepsilon}^{pl}\right)^n \right] \left[1 + C \ln\left(\frac{\dot{\bar{\varepsilon}}^{pl}}{\dot{\varepsilon}_0} \right) \right] \left(1 - \hat{\theta}^m\right),$$

optional strain rate dependence term

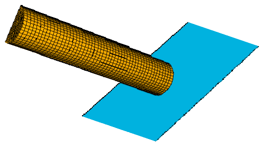
where $\hat{\theta}$ is the nondimensional temperature, defined as

$$\hat{\theta} = \begin{cases} 0 & \theta < \theta_{transition} \\ \dfrac{\theta - \theta_{transition}}{\theta_{melt} - \theta_{transition}} & \theta_{transition} \le \theta \le \theta_{melt} \\ 1 & \theta > \theta_{melt} \end{cases}$$

– The values of A, B, n, m, θ_{melt}, $\theta_{transition}$, and optionally C, and $\dot{\varepsilon}_0$ are defined as part of the material definition.

Metals

– Example: Oblique impact of copper rod



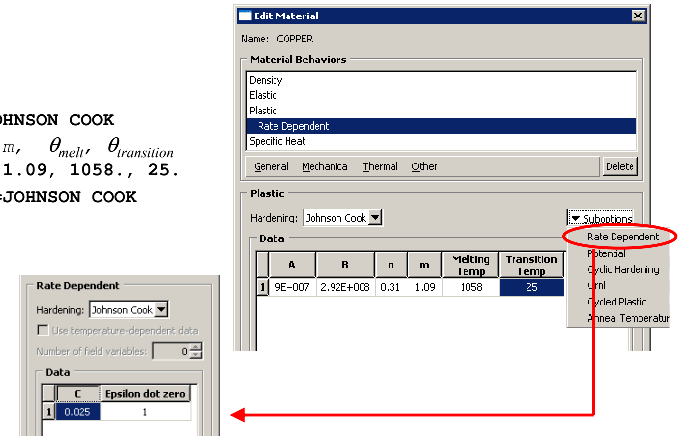
```
*MATERIAL,NAME=COPPER
*DENSITY
 8.96E3,
*ELASTIC
 124.E9, 0.34
*PLASTIC,HARDENING=JOHNSON COOK
```

** A, B, n, m, θ_{melt}, $\theta_{transition}$

```
90.E6, 292.E6, 0.31, 1.09, 1058., 25.
*RATE DEPENDENT,TYPE=JOHNSON COOK
```

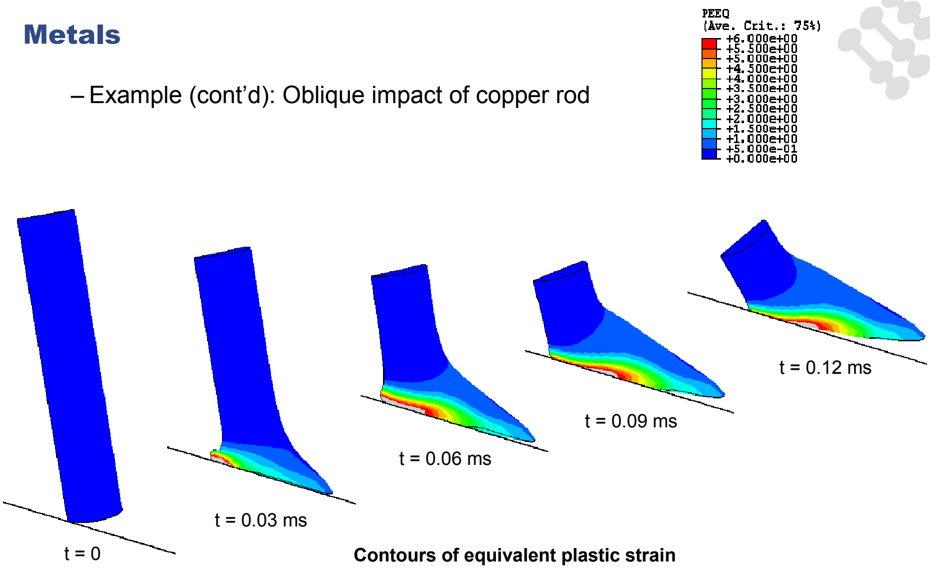
** C, $\dot{\varepsilon}_0$

```
0.025, 1.0
*SPECIFIC HEAT
:
```



Metals

– Example (cont'd): Oblique impact of copper rod

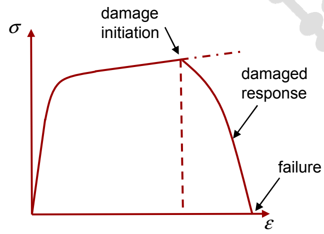


Contours of equivalent plastic strain

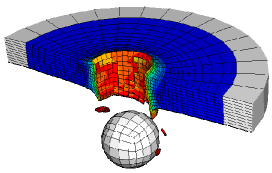
Metals

- **Progressive Damage and Failure**
  - allows for the modeling of:
    - damage initiation,
    - damage progression, and
    - failure

    in the Mises, Johnson-Cook, Hill, and Drucker-Prager plasticity models.
  - A combination of multiple failure mechanisms may act simultaneously on the same material.
  - These models are suitable for both quasi-static and dynamic situations.
  - These options will be discussed later in Lecture 9, *Material Damage and Failure*.



Typical material response showing progressive damage



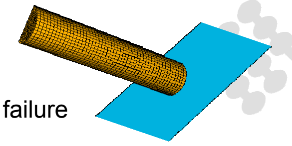
Projectile penetrates eroding plate

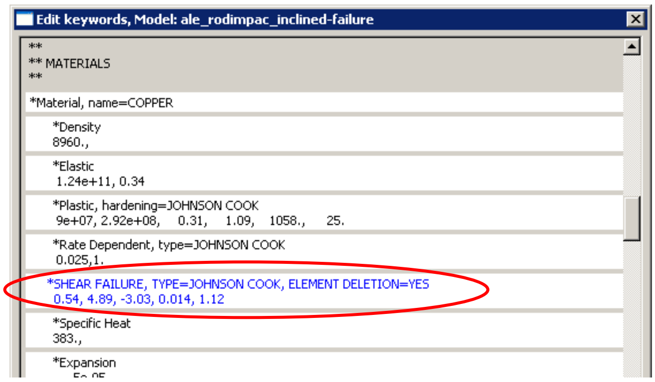
Metals

- **Dynamic failure models**
  - The following failure models are available for **high-strain-rate dynamic problems**:
    - the shear failure model driven by plastic yielding
    - the tensile failure model driven by tensile loading.
  - These models can be used with Johnson-Cook or Mises plasticity.
  - By default, when the failure criterion is met the element is deleted.
    - i.e. all stress components are set to zero and remain zero for the rest of the analysis.
  - If you choose not to delete failed elements, they will continue to support compressive pressure stress.

Metals

– Example (cont'd): Oblique impact of copper rod with failure

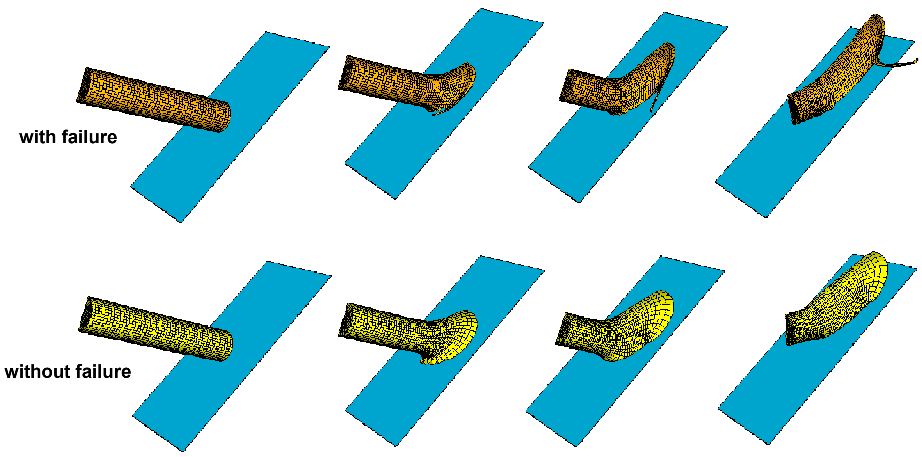



Edit keywords, Model: ale_rodimpac_inclined-failure

```
**
** MATERIALS
**
*Material, name=COPPER
  *Density
  8960.,
  *Elastic
   1.24e+11, 0.34
  *Plastic, hardening=JOHNSON COOK
   9e+07, 2.92e+08,    0.31,    1.09,    1058.,    25.
  *Rate Dependent, type=JOHNSON COOK
   0.025,1.
 *SHEAR FAILURE, TYPE=JOHNSON COOK, ELEMENT DELETION=YES
  0.54, 4.89, -3.03, 0.014, 1.12
  *Specific Heat
  383.,
  *Expansion
```

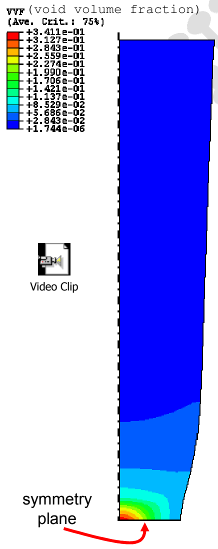

Metals

– Example (cont'd): Oblique impact of copper rod



Metals

- **Porous metal plasticity**
  - The porous metal plasticity model is intended for metals with relative densities greater than 90% (i.e., a dilute concentration of voids).
  - The model is based on Gurson's porous plasticity model with void nucleation and failure.
  - Inelastic flow is based on a potential function which characterizes the porosity in terms of a single state variable—the relative density.
  - The model is well-tuned for tensile applications, such as fracture studies with void coalescence, but it is also useful for compressive cases where the material densifies.
  - The details of this material model are discussed in the *Metal Inelasticity in ABAQUS* lecture notes.



necking of a round tensile bar

Metals

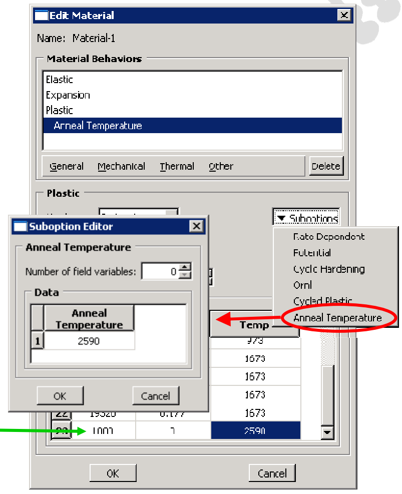
- **Annealing or Melting**
  - The effects of melting and resolidification in metals subjected to high-temperature deformation processes can be modeled.
    - The capability can also be used to model the effects of other forms of annealing, such as recrystallization.
  - If the temperature at a material point rises above the specified annealing temperature, the material point loses its hardening memory.
    - The effect of prior work hardening is removed by setting the equivalent plastic strain to zero.
    - For kinematic and combined hardening models the backstress tensor is also reset to zero.
  - Annealing is only available for the Mises, Johnson-Cook, and Hill plasticity models.

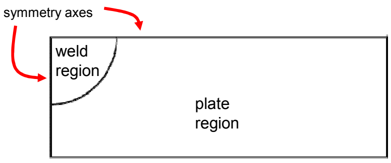
Metals

– Example: Spot weld

```
*MATERIAL ,NAME=MAT1
*ELASTIC
28.1E6,.2642
*PLASTIC
39440., 0.,      70
50170., .00473,  70
54950., .01264,  70
...
1000.,    0.,    2590
*ANNEAL TEMPERATURE
2590
```

No hardening at (and above) anneal temperature





model geometry

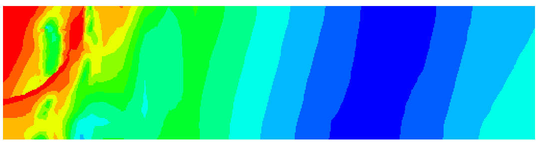
Metals

– Example (cont'd): Spot weld

- Residual stresses in the weld region are significantly reduced when annealing is included in the material definition.

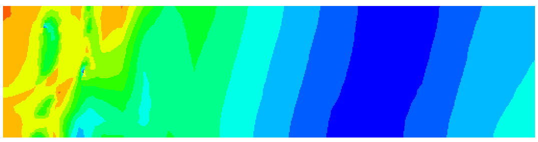
weld region

Residual stresses without annealing



weld region

Residual stresses with annealing



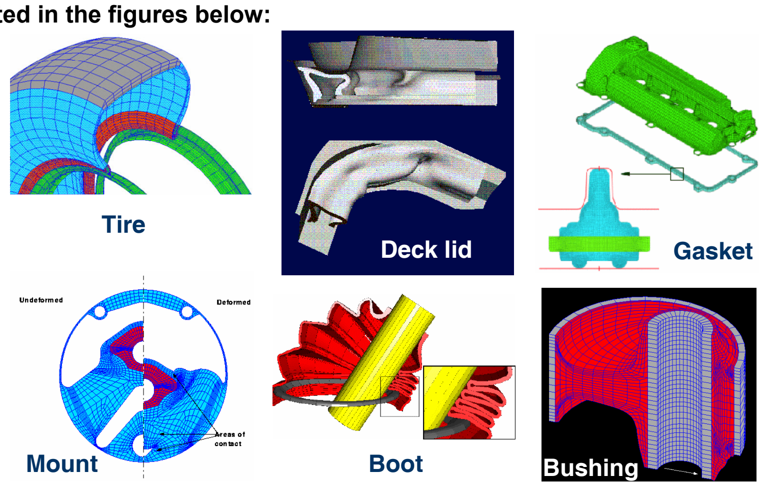
Metals

- If, during the deformation history, the temperature of the point falls below the annealing temperature, it can work harden again.
- Depending upon the temperature history, a material point may lose and accumulate memory several times.
- This annealing temperature material option is not related to the annealing analysis step procedure.
  - An annealing step can be defined to simulate the annealing process for the entire model, independent of temperature.

Rubber Elasticity

Rubber Elasticity

- **Rubber materials are widely used in many engineering applications, as indicated in the figures below:**



Rubber Elasticity

– The mechanical behavior of rubber (hyperelastic or hyperfoam) materials is expressed in terms of a strain energy potential

$$U = U(F), \quad \text{such that } S = \frac{\partial U(F)}{\partial F},$$

where S is a stress measure and F is a measure of deformation.

– Because the material is initially isotropic, we write the strain energy potential in terms of the strain invariants $\bar{I}_1, \bar{I}_2,$ and J_{el}:

$$U = U(\bar{I}_1,\ \bar{I}_2,\ J_{el}).$$

$\bar{I}_1$ and $\bar{I}_2$ are measures of deviatoric strain.

J_{el} is the volume ratio, a measure of volumetric strain.

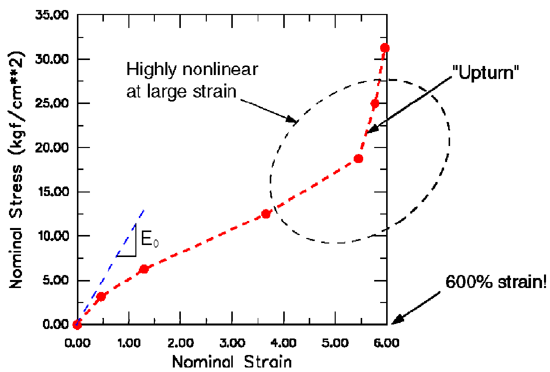
Rubber Elasticity

Physically motivated models	Material parameters (deviatoric behavior)
Arruda-Boyce	2
Van der Waals	4
Phenomenological models	
Polynomial (order N)	$\geq 2N$
Mooney-Rivlin (1st order)	2
Reduced polynomial (independent of $\bar{I}_2$)	N
Neo-Hookean (1st order)	1
Yeoh (3rd order)	3
Ogden (order N)	$2N$
Marlow (independent of $\bar{I}_2$)	N/A

 ABAQUS

Rubber Elasticity

- **Comparison of the solid rubber models**
  - Gum stock uniaxial data (Gerke):
    - Crude data but captures essential characteristics.



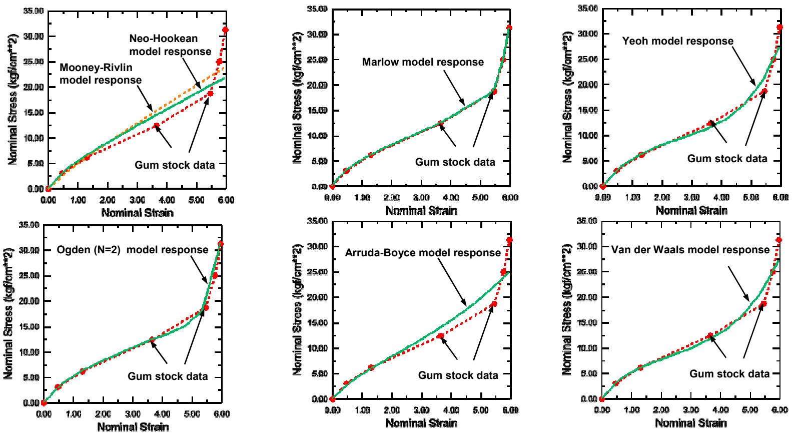
 ABAQUS

Rubber Elasticity

Video Clip

– Unit-element uniaxial tension tests are performed with ABAQUS.

- All material parameters are evaluated automatically by ABAQUS.

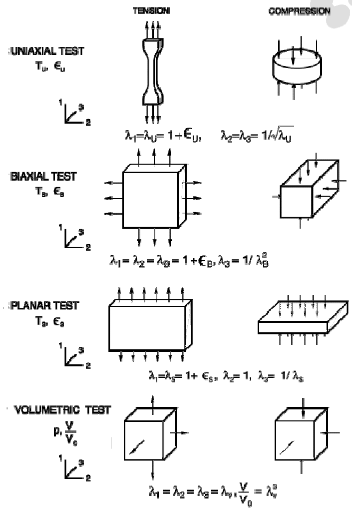


 ABAQUS

Rubber Elasticity

– Choosing a strain energy function in a particular problem depends on the availability of sufficient and “accurate” experimental data.

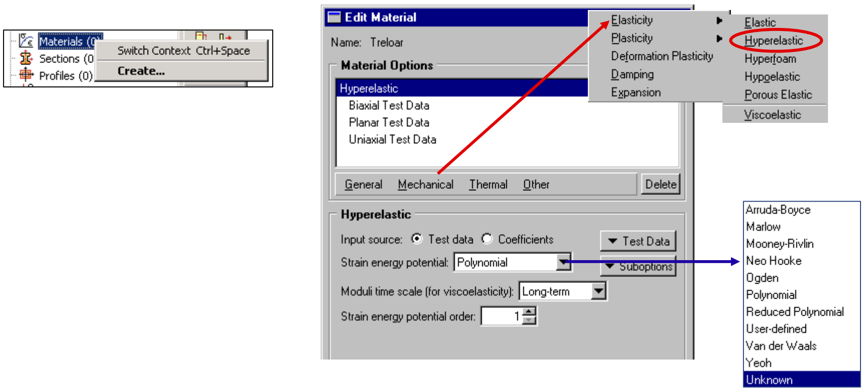
- Use data from experiments involving simple deformations:
  - – Uniaxial tension and compression
  - – Biaxial tension and compression
  - – Planar tension and compression
- If compressibility is important, volumetric test data must also be used.
  - – E.g., highly confined materials (such as an O-ring).



 ABAQUS

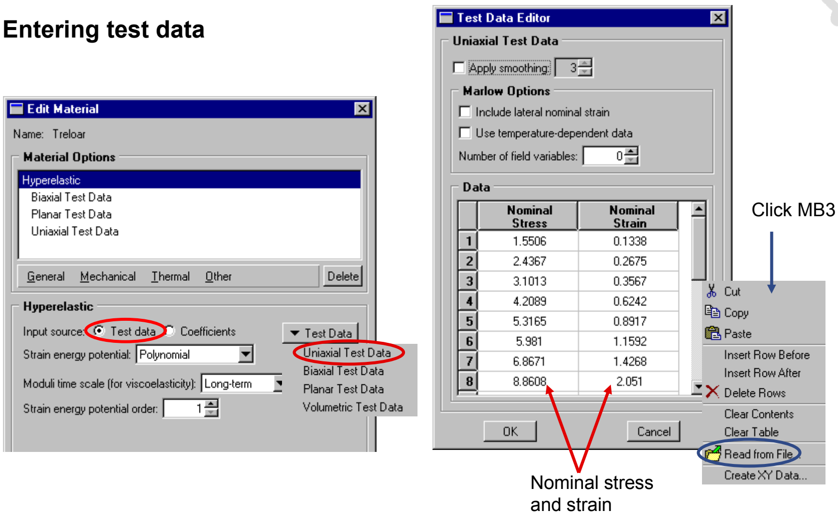
Rubber Elasticity

- **Defining rubber elasticity in ABAQUS/CAE: hyperelasticity**



Rubber Elasticity

- **Entering test data**



Rubber Elasticity

– Rubber elasticity keyword interface:

```
*MATERIAL, NAME=RUBBER
*HYPERELASTIC, NEO HOOKE, TEST DATA INPUT
*UNIAXIAL TEST DATA
 0.0,0.0
 0.03,0.02
 0.15,0.1
 0.23,0.2
 0.33,0.34
 0.41,0.57
 0.51,0.85
 ...
```

Omit to specify material coefficients directly (TEST DATA INPUT)

Specify one of the following energy functions:

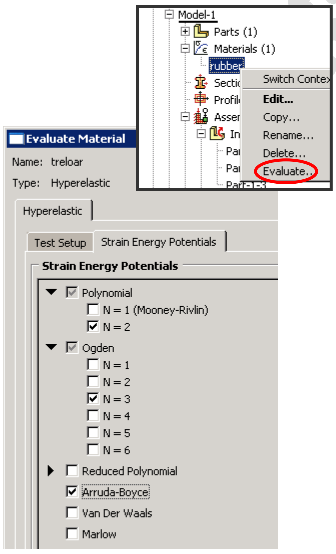
- **POLYNOMIAL** (default)
- **NEO HOOKE**
- **MOONEY-RIVLIN**
- **REDUCED POLYNOMIAL**
- **YEOH**
- **OGDEN**
- **ARRUDA-BOYCE**
- **VAN DER WAALS**
- **MARLOW**

Nominal stress and strain

With both polynomial models and Ogden model define the order, **N=**, of the series expansion.

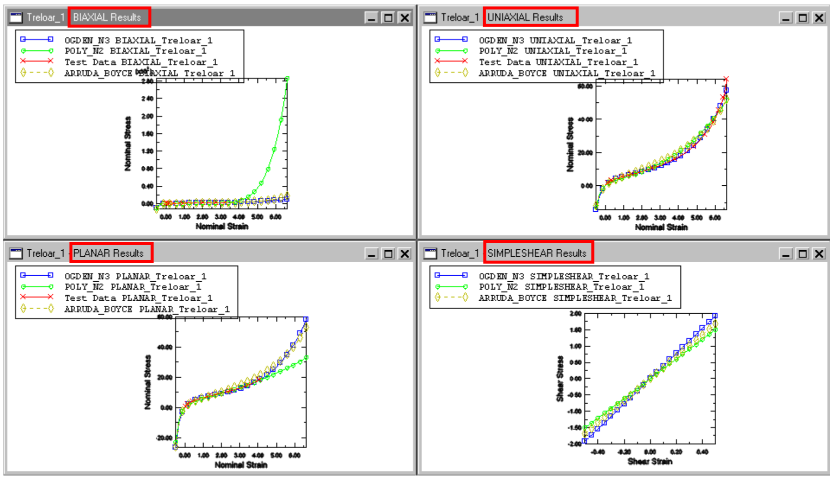
Rubber Elasticity

- **Automatic evaluation of the models using ABAQUS/CAE**
  – Verify correlation between predicted behavior and experimental data.
  – Use ABAQUS/CAE to perform standard unit-element tests.
    - Supply experimental test data.
    - Specify material models and deformation modes.
  – X–Y plots appear for each test.
    - Predicted nominal stress-strain curves plotted against experimental test data.



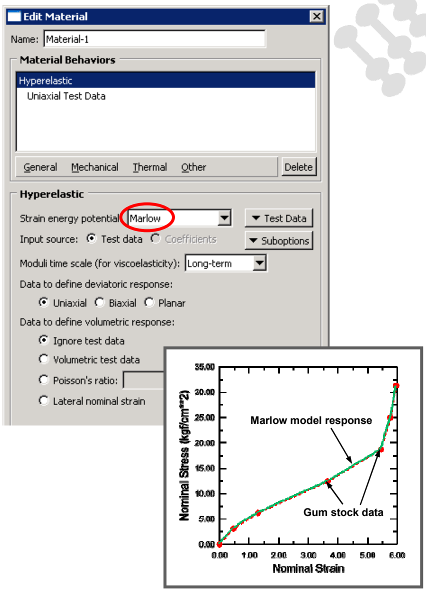
Rubber Elasticity

- **ABAQUS/CAE automatic evaluation results example**



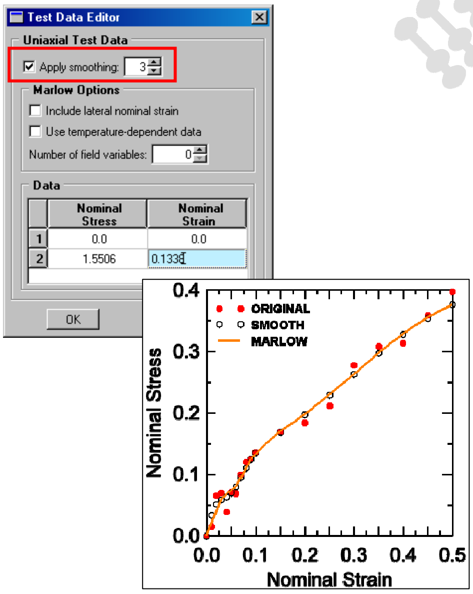
Rubber Elasticity

- **Marlow (General First Invariant) Model**
  - The Marlow model is a general first invariant model that can **exactly** reproduce the test data from one of the standard modes of loading (uniaxial, biaxial, or planar)
    - The responses for the other modes are also reasonably good.
  - This model should be used when limited test data are available.
    - The model works best when detailed data for one kind of test are available.



Rubber Elasticity

- **The test data input option provides a data-smoothing capability.**
  - This feature is useful in situations where the test data do not vary smoothly.
  - The user can control the smoothing process.
  - Smoothing is particularly important for the Marlow model.



Rubber Elasticity

- **Compressibility**
  - Most elastomers have very little compressibility compared to their shear flexibility.
  - Except for plane stress, ABAQUS/Explicit has no mechanism for enforcing strict incompressibility at the material points.
    - Some compressibility is always assumed.
    - If no value is given for the material compressibility, ABAQUS/Explicit assumes an initial Poisson's ratio of 0.475.
    - This default provides much more compressibility than is available in most elastomers.
      - However, if the material is relatively unconfined, this softer modeling of the bulk behavior provides accurate results.

Rubber Elasticity

- The material compressibility parameters may be entered directly to override the default setting.
  - Limit the initial Poisson's ratio to no greater than 0.495 to avoid high-frequency noise in the dynamic solution and very small time increments.

Suggested upper limit → (row 100)

K_0/μ_0	Poisson's ratio
10	0.452
20	0.475
50	0.490
100	0.495
1000	0.4995
10,000	0.49995

Rubber Elasticity

- **Modeling recommendations**
  - When using hyperelastic or hyperfoam materials in ABAQUS/Explicit, the following options are strongly recommended:
    - Distortion control with
    - Enhanced hourglass control.
  - Adaptive meshing is not recommended with hyperelastic or hyperfoam materials.
    - Distortion control provides the alternative to adaptive meshing.
    - These options are discussed in Lecture 6, *Adaptive Meshing and Distortion Control*.

Concrete

Concrete

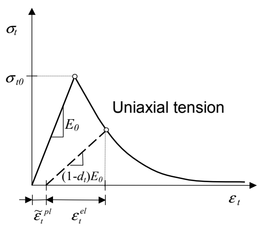
- **Brittle cracking model**
  - Intended for applications in which the concrete behavior is dominated by tensile cracking and compressive failure is not important.
  - Includes consideration of the anisotropy induced by cracking.
  - The compressive behavior is assumed to be always linear elastic.
  - A brittle failure criteria allows the removal of elements from a mesh.
  - This material model is not discussed further in this class.
    - For more information see “Cracking model for concrete,” section 11.5.2 of the ABAQUS Analysis User's Manual.

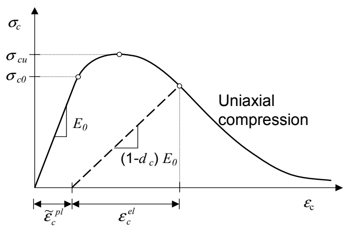
Concrete

- **Concrete Damaged Plasticity Model**
  - Intended as a general capability for the analysis of concrete structures under monotonic, cyclic, and/or dynamic loading
  - Scalar (isotropic) damage model, with tensile cracking and compressive crushing modes
  - Main features of the model:
    - The model is based on the scalar plastic damage models proposed by Lubliner et al. (1989) and by J. Lee & G.L. Fenves (1998).
    - The evolution of the yield surface is determined by two hardening variables, each of them linked to degradation mechanisms under tensile or compressive stress conditions.
    - The model accounts for the stiffness degradation mechanisms associated with each failure mode, as well as stiffness recovery effects during load reversals.

Concrete

- Mechanical response
  - The response is characterized by damaged plasticity
  - Two failure mechanisms: tensile cracking and compressive crushing
  - Evolution of failure is controlled by two hardening variables: $\tilde{\varepsilon}_t^{pl}$ and $\tilde{\varepsilon}_c^{pl}$





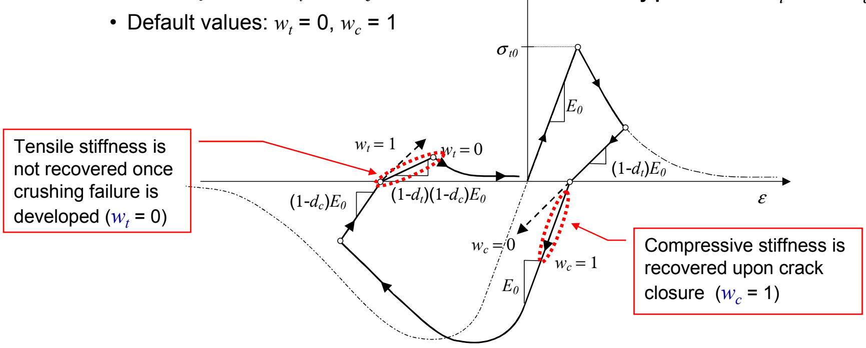
$$\sigma_t = \sigma_t(\tilde{\varepsilon}_t^{pl}, \dot{\tilde{\varepsilon}}_t^{pl}, \theta, f^\alpha)$$
$$d_t = d_t(\tilde{\varepsilon}_t^{pl}, \theta, f^\alpha); \quad 0 \le d_t \le 1$$
$$\bar{\sigma}_t = \sigma_t/(1 - d_t)$$

$$\sigma_c = \sigma_c(\tilde{\varepsilon}_c^{pl}, \dot{\tilde{\varepsilon}}_c^{pl}, \theta, f^\alpha)$$
$$d_c = d_c(\tilde{\varepsilon}_c^{pl}, \theta, f^\alpha); \quad 0 \le d_c \le 1$$
$$\bar{\sigma}_c = \sigma_c/(1 - d_c)$$

Concrete

– Cyclic loading conditions
  - Stiffness recovery is an important aspect of the mechanical response of concrete under cyclic conditions
  - User can specify the stiffness recovery factors w_t and w_c
  - Default values: w_t = 0, w_c = 1

Uniaxial load cycle (tension-compression-tension) assuming default values of the stiffness recovery parameters: w_t=0 and w_c=1

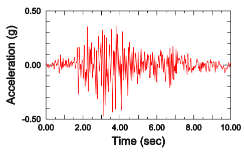


 ABAQUS

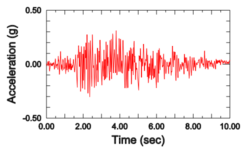
Concrete

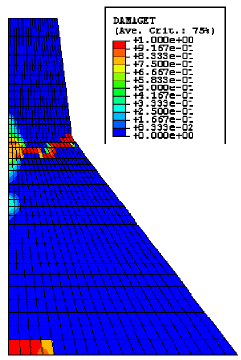
– Example: Seismic analysis of Koyna dam
  - Koyna dam (India), subjected to the December 11, 1967 earthquake of magnitude 6.5 on the Richter scale.
  - The dam undergoes severe damage but retains its overall structural stability.

Transverse ground acceleration



Vertical ground acceleration





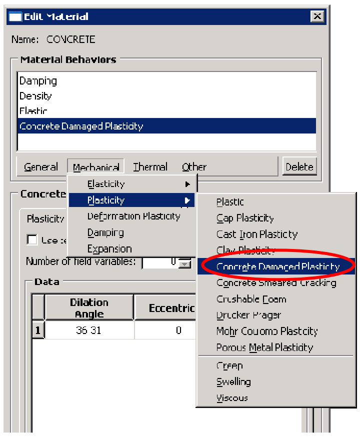
Structural damage due to tensile cracking failure (t=10 sec)

 ABAQUS

Concrete

– Example (cont'd): Seismic analysis of Koyna dam

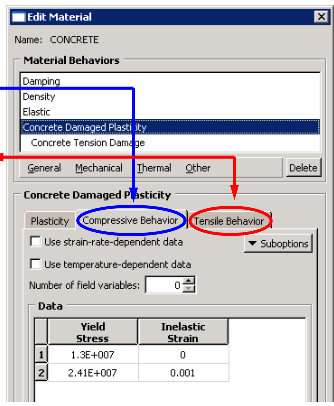
```
*MATERIAL, NAME=CONCRETE
*ELASTIC
 3.1027E+10, 0.2
*CONCRETE DAMAGED PLASTICITY
 36.31
*CONCRETE COMPRESSION HARDENING
 13.0E+6, 0.000
 24.1E+6, 0.001
*CONCRETE TENSION STIFFENING, TYPE=DISPLACEMENT
 2.9E+6          ,0
 1.94393E+6      ,0.000066185
 1.30305E+6      ,0.00012286
 0.873463E+6     ,0.000173427
 ...
*CONCRETE TENSION DAMAGE, TYPE=DISPLACEMENT,
 COMPRESSION RECOVERY=1
 0           ,0
 0.381217    ,0.000066185
 0.617107    ,0.00012286
 0.763072    ,0.000173427
 ...
```



Concrete

– Example (cont'd): Seismic analysis of Koyna dam

```
*MATERIAL, NAME=CONCRETE
*ELASTIC
 3.1027E+10, 0.2
*CONCRETE DAMAGED PLASTICITY
 36.31
*CONCRETE COMPRESSION HARDENING
 13.0E+6, 0.000
 24.1E+6, 0.001
*CONCRETE TENSION STIFFENING, TYPE=DISPLACEMENT
 2.9E+6          ,0
 1.94393E+6      ,0.000066185
 1.30305E+6      ,0.00012286
 0.873463E+6     ,0.000173427
 ...
*CONCRETE TENSION DAMAGE, TYPE=DISPLACEMENT,
 COMPRESSION RECOVERY=1
 0           ,0
 0.381217    ,0.000066185
 0.617107    ,0.00012286
 0.763072    ,0.000173427
 ...
```

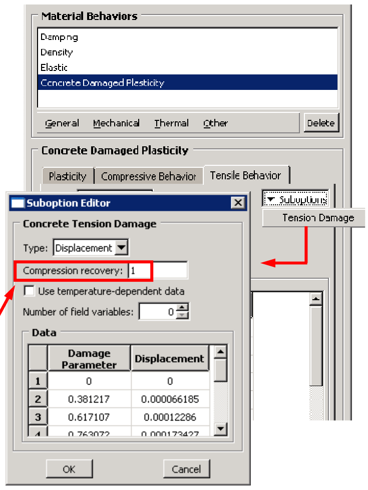


Concrete

– Example (cont'd): Seismic analysis of Koyna dam

```
*MATERIAL, NAME=CONCRETE
*ELASTIC
 3.1027E+10, 0.2
*CONCRETE DAMAGED PLASTICITY
 36.31
*CONCRETE COMPRESSION HARDENING
 13.0E+6, 0.000
 24.1E+6, 0.001
*CONCRETE TENSION STIFFENING, TYPE=DISPLACEMENT
 2.9E+6        ,0
 1.94393E+6    ,0.000066185
 1.30305E+6    ,0.00012286
 0.873463E+6   ,0.000173427
 ...
*CONCRETE TENSION DAMAGE, TYPE=DISPLACEMENT,
 COMPRESSION RECOVERY=1
 0          ,0
 0.381217   ,0.000066185
 0.617107   ,0.00012286
 0.763072   ,0.000173427
 ...
```

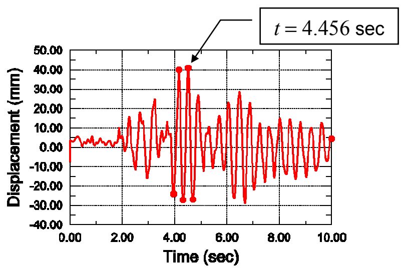
$W_c = 1$



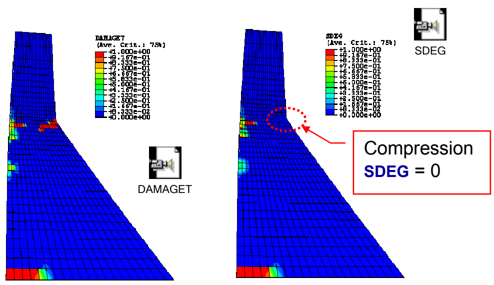
Concrete

– The tensile damage variable, DAMAGET, is a nondecreasing quantity associated with tensile (cracking) failure of the material.

– The stiffness degradation variable, SDEG, can increase or decrease, reflecting the stiffness recovery effects associated with the opening/closing of cracks.



Horizontal crest displacement
(relative to ground displacement)



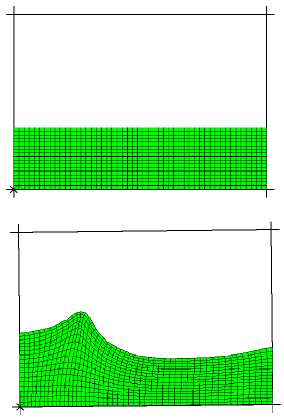
Contour plot of **DAMAGET** (left) and **SDEG** (right) at time $t = 4.456$ sec, corresponding to the largest excursion of the crest in the down-stream direction.

Additional Materials

Additional Materials

Video Clip

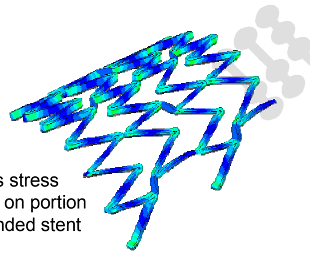
- **Hydrodynamic materials**
  - Equations of state material model
    - Provides a hydrodynamic material model in which the material's volumetric strength is determined by an equation of state
    - Applications include:
      - Fluids
      - Ideal gasses
      - Explosives
      - Compaction of granular materials
  - For more information see “Equation of state,” section 10.10.1 in the ABAQUS Analysis User's Manual.



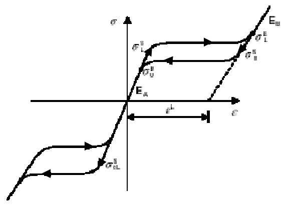
Water sloshing in a tank

Additional Materials

- **User-defined materials**
  - You can create additional material models through the VUMAT user subroutine.
  - This feature is very general and powerful;
    - any mechanical constitutive model can be added.
  - However, programming a VUMAT requires considerable effort and expertise.
  - For more information on user-defined materials refer to Appendix 3.



Mises stress contours on portion of expanded stent



complex uniaxial behavior of Nitinol modeled in a VUMAT subroutine

Technology Brief example:
Simulation of Implantable Nitinol Stents
ABAQUS Answer 1959