24.3.1 DAMAGE AND FAILURE FOR FIBER-REINFORCED COMPOSITES: OVERVIEW

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

References

- “Progressive damage and failure,” Section 24.1.1
- “Damage initiation for fiber-reinforced composites,” Section 24.3.2
- “Damage evolution and element removal for fiber-reinforced composites,” Section 24.3.3
- *DAMAGE INITIATION
- *DAMAGE EVOLUTION
- *DAMAGE STABILIZATION
- “Hashin damage” in “Defining damage,” Section 12.9.3 of the Abaqus/CAE User’s Manual, in the online HTML version of this manual

Overview

Abaqus offers a damage model enabling you to predict the onset of damage and to model damage evolution for elastic-brittle materials with anisotropic behavior. The model is primarily intended to be used with fiber-reinforced materials since they typically exhibit such behavior.

This damage model requires specification of the following:

- the undamaged response of the material, which must be linearly elastic (see “Linear elastic behavior,” Section 22.2.1);
- a damage initiation criterion (see “Progressive damage and failure,” Section 24.1.1, and “Damage initiation for fiber-reinforced composites,” Section 24.3.2); and
- a damage evolution response, including a choice of element removal (see “Progressive damage and failure,” Section 24.1.1, and “Damage evolution and element removal for fiber-reinforced composites,” Section 24.3.3).

General concepts of damage in unidirectional lamina

Damage is characterized by the degradation of material stiffness. It plays an important role in the analysis of fiber-reinforced composite materials. Many such materials exhibit elastic-brittle behavior; that is, damage in these materials is initiated without significant plastic deformation. Consequently, plasticity can be neglected when modeling behavior of such materials.

The fibers in the fiber-reinforced material are assumed to be parallel, as depicted in Figure 24.3.1–1. You must specify material properties in a local coordinate system defined by the user. The lamina is in the 1–2 plane, and the local 1 direction corresponds to the fiber direction. You must specify the undamaged material response using one of the methods for defining an orthotropic linear elastic material (“Linear elastic behavior,” Section 22.2.1); the most convenient of which is the method for defining an orthotropic material in plane stress (“Defining orthotropic elasticity in plane stress” in “Linear elastic behavior,”

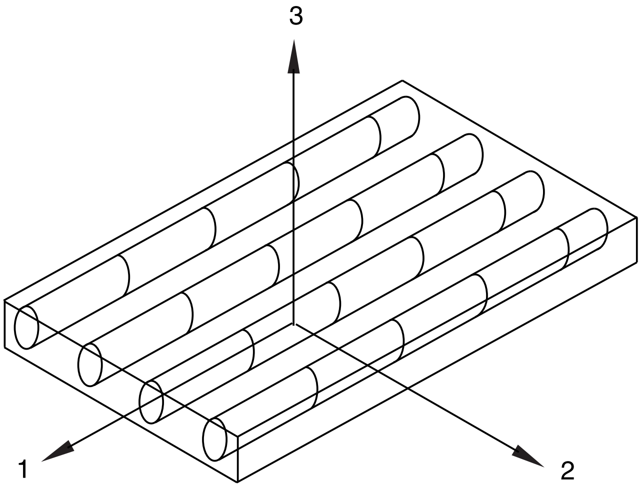

Figure 24.3.1–1 Unidirectional lamina.

Section 22.2.1). However, the material response can also be defined in terms of the engineering constants or by specifying the elastic stiffness matrix directly.

The Abaqus anisotropic damage model is based on the work of Matzenmiller et. al (1995), Hashin and Rotem (1973), Hashin (1980), and Camanho and Davila (2002).

Four different modes of failure are considered:

- fiber rupture in tension;
- fiber buckling and kinking in compression;
- matrix cracking under transverse tension and shearing; and
- matrix crushing under transverse compression and shearing.

In Abaqus the onset of damage is determined by the initiation criteria proposed by Hashin and Rotem (1973) and Hashin (1980), in which the failure surface is expressed in the effective stress space (the stress acting over the area that effectively resists the force). These criteria are discussed in detail in "Damage initiation for fiber-reinforced composites," Section 24.3.2.

The response of the material is computed from

$$\sigma = \mathbf{C_d}\varepsilon,$$

where ε is the strain and $\mathbf{C_d}$ is the elasticity matrix, which reflects any damage and has the form

$$\mathbf{C_d} = \frac{1}{D}\begin{bmatrix} (1-d_f)E_1 & (1-d_f)(1-d_m)\nu_{21}E_1 & 0 \\ (1-d_f)(1-d_m)\nu_{12}E_2 & (1-d_m)E_2 & 0 \\ 0 & 0 & (1-d_s)GD \end{bmatrix},$$

where $D = 1 - (1 - d_f)(1 - d_m)\nu_{12}\nu_{21}$, d_f reflects the current state of fiber damage, d_m reflects the current state of matrix damage, d_s reflects the current state of shear damage, E_1 is the Young's modulus in the fiber direction, E_2 is the Young's modulus in the direction perpendicular to the fibers, G is the shear modulus, and ν_{12} and ν_{21} are Poisson's ratios.

The evolution of the elasticity matrix due to damage is discussed in more detail in "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3; that section also discusses:

- options for treating severe damage ("Maximum degradation and choice of element removal" in "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3); and
- viscous regularization ("Viscous regularization" in "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3).

Elements

The fiber-reinforced composite damage model must be used with elements with a plane stress formulation, which include plane stress, shell, continuum shell, and membrane elements.

Additional references

- Camanho, P. P., and C. G. Davila, "Mixed-Mode Decohesion Finite Elements for the Simulation of Delamination in Composite Materials," NASA/TM-2002–211737, pp. 1–37, 2002.
- Hashin, Z., "Failure Criteria for Unidirectional Fiber Composites," Journal of Applied Mechanics, vol. 47, pp. 329–334, 1980.
- Hashin, Z., and A. Rotem, "A Fatigue Criterion for Fiber-Reinforced Materials," Journal of Composite Materials, vol. 7, pp. 448–464, 1973.
- Matzenmiller, A., J. Lubliner, and R. L. Taylor, "A Constitutive Model for Anisotropic Damage in Fiber-Composites," Mechanics of Materials, vol. 20, pp. 125–152, 1995.

24.3.2 DAMAGE INITIATION FOR FIBER-REINFORCED COMPOSITES

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

References

- "Progressive damage and failure," Section 24.1.1
- "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3
- *DAMAGE INITIATION
- "Hashin damage" in "Defining damage," Section 12.9.3 of the Abaqus/CAE User's Manual, in the online HTML version of this manual

Overview

The material damage initiation capability for fiber-reinforced materials:

- requires that the behavior of the undamaged material is linearly elastic (see "Linear elastic behavior," Section 22.2.1);
- is based on Hashin's theory (Hashin and Rotem, 1973, and Hashin, 1980);
- takes into account four different failure modes: fiber tension, fiber compression, matrix tension, and matrix compression; and
- can be used in combination with the damage evolution model described in "Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3 (see "Failure of blunt notched fiber metal laminates," Section 1.4.6 of the Abaqus Example Problems Manual).

Damage Initiation

Damage initiation refers to the onset of degradation at a material point. In Abaqus the damage initiation criteria for fiber-reinforced composites are based on Hashin's theory (see Hashin and Rotem, 1973, and Hashin, 1980). These criteria consider four different damage initiation mechanisms: fiber tension, fiber compression, matrix tension, and matrix compression.

The initiation criteria have the following general forms:

Fiber tension ($\hat{\sigma}_{11} \geq 0$):

$$F_f^t = \left(\frac{\hat{\sigma}_{11}}{X^T}\right)^2 + \alpha\left(\frac{\hat{\tau}_{12}}{S^L}\right)^2.$$

Fiber compression ($\hat{\sigma}_{11} < 0$):

$$F_f^c = \left(\frac{\hat{\sigma}_{11}}{X^C}\right)^2.$$

Matrix tension ($\hat{\sigma}_{22} \geq 0$):

$$F_m^t = \left(\frac{\hat{\sigma}_{22}}{Y^T}\right)^2 + \left(\frac{\hat{\tau}_{12}}{S^L}\right)^2.$$

Matrix compression ($\hat{\sigma_{22}} < 0$):

$$F_m^c = \left(\frac{\hat{\sigma}_{22}}{2S^T}\right)^2 + \left[\left(\frac{Y^C}{2S^T}\right)^2 - 1\right]\frac{\hat{\sigma}_{22}}{Y^C} + \left(\frac{\hat{\tau}_{12}}{S^L}\right)^2.$$

In the above equations

X^T	denotes the longitudinal tensile strength;
X^C	denotes the longitudinal compressive strength;
Y^T	denotes the transverse tensile strength;
Y^C	denotes the transverse compressive strength;
S^L	denotes the longitudinal shear strength;
S^T	denotes the transverse shear strength;
α	is a coefficient that determines the contribution of the shear stress to the fiber tensile initiation criterion; and
$\hat{\sigma}_{11}, \hat{\sigma}_{22}, \hat{\tau}_{12}$	are components of the effective stress tensor, $\hat{\sigma}$, that is used to evaluate the initiation criteria and which is computed from:

$$\hat{\sigma} = \mathbf{M}\sigma,$$

where σ is the true stress and $\mathbf{M}$ is the damage operator:

$$\mathbf{M} = \begin{bmatrix} \frac{1}{(1-d_f)} & 0 & 0 \\ 0 & \frac{1}{(1-d_m)} & 0 \\ 0 & 0 & \frac{1}{(1-d_s)} \end{bmatrix}.$$

d_f, d_m, and d_s are internal (damage) variables that characterize fiber, matrix, and shear damage, which are derived from damage variables d_f^t, d_f^c, d_m^t, and d_m^c, corresponding to the four modes previously discussed, as follows:

$$d_f = \begin{cases} d_f^t & \text{if } \hat{\sigma}_{11} \geq 0, \\ d_f^c & \text{if } \hat{\sigma}_{11} < 0, \end{cases}$$

$$d_m = \begin{cases} d_m^t & \text{if } \hat{\sigma}_{22} \geq 0, \\ d_m^c & \text{if } \hat{\sigma}_{22} < 0, \end{cases}$$

$$d_s = 1 - (1 - d_f^t)(1 - d_f^c)(1 - d_m^t)(1 - d_m^c).$$

Prior to any damage initiation and evolution the damage operator, $\mathbf{M}$, is equal to the identity matrix, so $\hat{\sigma} = \sigma$. Once damage initiation and evolution has occurred for at least one mode, the damage operator becomes significant in the criteria for damage initiation of other modes (see “Damage evolution and element removal for fiber-reinforced composites," Section 24.3.3, for discussion of damage evolution). The effective stress, $\hat{\sigma}$, is intended to represent the stress acting over the damaged area that effectively resists the internal forces.

The initiation criteria presented above can be specialized to obtain the model proposed in Hashin and Rotem (1973) by setting $\alpha = 0.0$ and $S^T = Y^C/2$ or the model proposed in Hashin (1980) by setting $\alpha = 1.0$.

An output variable is associated with each initiation criterion (fiber tension, fiber compression, matrix tension, matrix compression) to indicate whether the criterion has been met. A value of 1.0 or higher indicates that the initiation criterion has been met (see "Output" for further details). If you define a damage initiation model without defining an associated evolution law, the initiation criteria will affect only output. Thus, you can use these criteria to evaluate the propensity of the material to undergo damage without modeling the damage process.

Input File Usage: Use the following option to define the Hashin damage initiation criterion:

*DAMAGE INITIATION, CRITERION=HASHIN, ALPHA=α
X^T, X^C, Y^T, Y^C, S^L, S^T

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Damage for Fiber-Reinforced Composites→Hashin Damage**

Elements

The damage initiation criteria must be used with elements with a plane stress formulation, which include plane stress, shell, continuum shell, and membrane elements.

Output

In addition to the standard output identifiers available in Abaqus ("Abaqus/Standard output variable identifiers," Section 4.2.1, and, "Abaqus/Explicit output variable identifiers," Section 4.2.2), the following variables relate specifically to damage initiation at a material point in the fiber-reinforced composite damage model:

DMICRT	All damage initiation criteria components.
HSNFTCRT	Maximum value of the fiber tensile initiation criterion experienced during the analysis.
HSNFCCRT	Maximum value of the fiber compressive initiation criterion experienced during the analysis.
HSNMTCRT	Maximum value of the matrix tensile initiation criterion experienced during the analysis.
HSNMCCRT	Maximum value of the matrix compressive initiation criterion experienced during the analysis.

For the variables above that indicate whether an initiation criterion in a damage mode has been satisfied or not, a value that is less than 1.0 indicates that the criterion has not been satisfied, while a value of 1.0 or higher indicates that the criterion has been satisfied. If you define a damage evolution model, the maximum value of this variable does not exceed 1.0. However, if you do not define a damage evolution model, this variable can have values higher than 1.0, which indicates by how much the criterion has been exceeded.

Additional references

- Hashin, Z., "Failure Criteria for Unidirectional Fiber Composites," Journal of Applied Mechanics, vol. 47, pp. 329–334, 1980.
- Hashin, Z., and A. Rotem, "A Fatigue Criterion for Fiber-Reinforced Materials," Journal of Composite Materials, vol. 7, pp. 448–464, 1973.

24.3.3 DAMAGE EVOLUTION AND ELEMENT REMOVAL FOR FIBER-REINFORCED COMPOSITES

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

References

- "Progressive damage and failure," Section 24.1.1
- "Damage initiation for fiber-reinforced composites," Section 24.3.2
- *DAMAGE EVOLUTION
- "Damage evolution" in "Defining damage," Section 12.9.3 of the Abaqus/CAE User's Manual, in the online HTML version of this manual

Overview

The damage evolution capability for fiber-reinforced materials in Abaqus:

- assumes that damage is characterized by progressive degradation of material stiffness, leading to material failure;
- requires linearly elastic behavior of the undamaged material (see "Linear elastic behavior," Section 22.2.1);
- takes into account four different failure modes: fiber tension, fiber compression, matrix tension, and matrix compression;
- uses four damage variables to describe damage for each failure mode;
- must be used in combination with Hashin's damage initiation criteria ("Damage initiation for fiber-reinforced composites," Section 24.3.2);
- is based on energy dissipation during the damage process;
- offers options for what occurs upon failure, including the removal of elements from the mesh; and
- can be used in conjunction with a viscous regularization of the constitutive equations to improve the convergence rate in the softening regime.

Damage evolution

The previous section ("Damage initiation for fiber-reinforced composites," Section 24.3.2) discussed the damage initiation in plane stress fiber-reinforced composites. This section will discuss the post-damage initiation behavior for cases in which a damage evolution model has been specified. Prior to damage initiation the material is linearly elastic, with the stiffness matrix of a plane stress orthotropic material. Thereafter, the response of the material is computed from

$$\sigma = \mathbf{C_d}\varepsilon,$$

where ε is the strain and $\mathbf{C_d}$ is the damaged elasticity matrix, which has the form

$$\mathbf{C_d} = \frac{1}{D}\begin{bmatrix} (1-d_f)E_1 & (1-d_f)(1-d_m)\nu_{21}E_1 & 0 \\ (1-d_f)(1-d_m)\nu_{12}E_2 & (1-d_m)E_2 & 0 \\ 0 & 0 & (1-d_s)GD \end{bmatrix},$$

where $D = 1-(1-d_f)(1-d_m)\nu_{12}\nu_{21}$, d_f reflects the current state of fiber damage, d_m reflects the current state of matrix damage, d_s reflects the current state of shear damage, E_1 is the Young's modulus in the fiber direction, E_2 is the Young's modulus in the matrix direction, G is the shear modulus, and ν_{12} and ν_{21} are Poisson's ratios.

The damage variables d_f, d_m, and d_s are derived from damage variables d_f^t, d_f^c, d_m^t, and d_m^c, corresponding to the four failure modes previously discussed, as follows:

$$d_f = \begin{cases} d_f^t & \text{if } \hat{\sigma}_{11} \geq 0, \\ d_f^c & \text{if } \hat{\sigma}_{11} < 0, \end{cases}$$

$$d_m = \begin{cases} d_m^t & \text{if } \hat{\sigma}_{22} \geq 0, \\ d_m^c & \text{if } \hat{\sigma}_{22} < 0, \end{cases}$$

$$d_s = 1-(1-d_f^t)(1-d_f^c)(1-d_m^t)(1-d_m^c),$$

$\hat{\sigma}_{11}$ and $\hat{\sigma}_{22}$ are components of the effective stress tensor. The effective stress tensor is primarily used to evaluate damage initiation criteria; see "Damage initiation for fiber-reinforced composites," Section 24.3.2, for a description of how the effective stress tensor is computed.

Evolution of damage variables for each mode

To alleviate mesh dependency during material softening, Abaqus introduces a characteristic length into the formulation, so that the constitutive law is expressed as a stress-displacement relation. The damage variable will evolve such that the stress-displacement behaves as shown in Figure 24.3.3–1 in each of the four failure modes. The positive slope of the stress-displacement curve prior to damage initiation corresponds to linear elastic material behavior; the negative slope after damage initiation is achieved by evolution of the respective damage variables according to the equations shown below.

Equivalent displacement and stress for each of the four damage modes are defined as follows:

Fiber tension ($\hat{\sigma}_{11} \geq 0$):

$$\delta_{eq}^{ft} = L^c\sqrt{\langle\varepsilon_{11}\rangle^2 + \alpha\varepsilon_{12}^2},$$

$$\sigma_{eq}^{ft} = \frac{\langle\sigma_{11}\rangle\langle\varepsilon_{11}\rangle + \alpha\tau_{12}\varepsilon_{12}}{\delta_{eq}^{ft}/L^c},$$

Fiber compression ($\hat{\sigma}_{11} < 0$):

$$\delta_{eq}^{fc} = L^c\langle-\varepsilon_{11}\rangle,$$

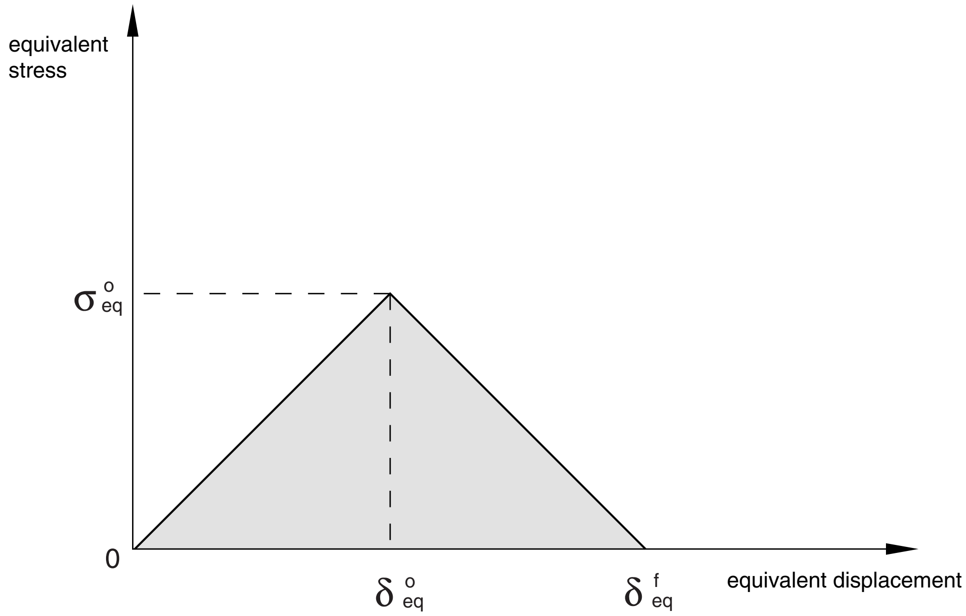

Figure 24.3.3–1 Equivalent stress versus equivalent displacement.

$$\sigma_{eq}^{fc} = \frac{\langle -\sigma_{11}\rangle\langle -\varepsilon_{11}\rangle}{\delta_{eq}^{fc}/L^c}.$$

Matrix tension ($\hat{\sigma}_{22} \geq 0$):

$$\delta_{eq}^{mt} = L^c\sqrt{\langle\varepsilon_{22}\rangle^2 + \varepsilon_{12}^2},$$

$$\sigma_{eq}^{mt} = \frac{\langle\sigma_{22}\rangle\langle\varepsilon_{22}\rangle + \tau_{12}\varepsilon_{12}}{\delta_{eq}^{mt}/L^c}.$$

Matrix compression ($\hat{\sigma}_{22} < 0$):

$$\delta_{eq}^{mc} = L^c\sqrt{\langle -\varepsilon_{22}\rangle^2 + \varepsilon_{12}^2},$$

$$\sigma_{eq}^{mc} = \frac{\langle -\sigma_{22}\rangle\langle -\varepsilon_{22}\rangle + \tau_{12}\varepsilon_{12}}{\delta_{eq}^{mc}/L^c}.$$

The characteristic length, L^c, is based on the element geometry and formulation: it is a typical length of a line across an element for a first-order element; it is half of the same typical length for a second-order element. For membranes and shells it is a characteristic length in the reference surface, computed as the square root of the area. The symbol $\langle\ \rangle$ in the equations above represents the Macaulay bracket operator, which is defined for every $\alpha \in \Re$ as $\langle \alpha \rangle = (\alpha + |\alpha|)/2$.

After damage initiation (i.e., $\delta_{eq} \geq \delta_{eq}^0$) for the behavior shown in Figure 24.3.3–1, the damage variable for a particular mode is given by the following expression

$$d = \frac{\delta_{eq}^f(\delta_{eq} - \delta_{eq}^0)}{\delta_{eq}(\delta_{eq}^f - \delta_{eq}^0)},$$

where δ_{eq}^0 is the initial equivalent displacement at which the initiation criterion for that mode was met and δ_{eq}^f is the displacement at which the material is completely damaged in this failure mode. The above relation is presented graphically in Figure 24.3.3–2.

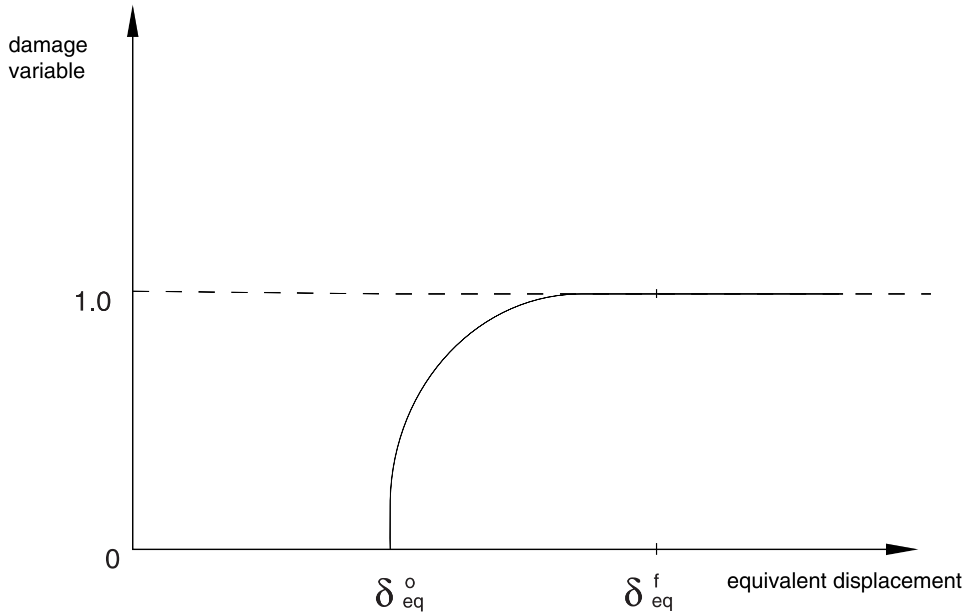

Figure 24.3.3–2 Damage variable as a function of equivalent displacement.

The values of δ_{eq}^0 for the various modes depend on the elastic stiffness and the strength parameters specified as part of the damage initiation definition (see "Damage initiation for fiber-reinforced composites," Section 24.3.2). For each failure mode you must specify the energy dissipated due to failure, G^c, which corresponds to the area of the triangle OAC in Figure 24.3.3–3. The values of δ_{eq}^f for the various modes depend on the respective G^c values.

Unloading from a partially damaged state, such as point B in Figure 24.3.3–3, occurs along a linear path toward the origin in the plot of equivalent stress vs. equivalent displacement; this same path is followed back to point B upon reloading as shown in the figure.

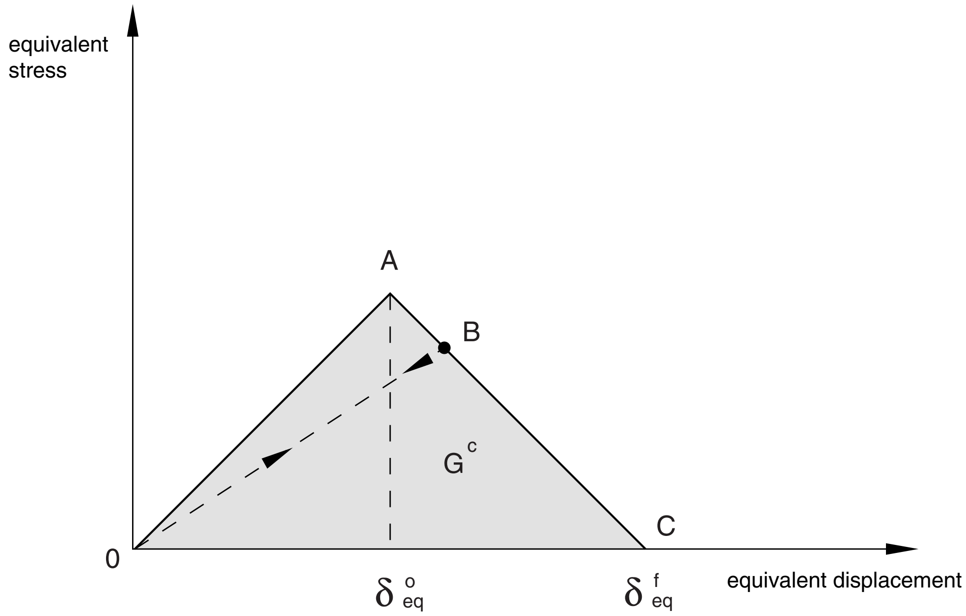

Figure 24.3.3–3 Linear damage evolution.

Input File Usage: Use the following option to define the damage evolution law:

*DAMAGE EVOLUTION, TYPE=ENERGY, SOFTENING=LINEAR

G^c_{ft}, G^c_{fc}, G^c_{mt}, G^c_{mc}

where G^c_{ft}, G^c_{fc}, G^c_{mt}, and G^c_{mc} are energies dissipated during damage for fiber tension, fiber compression, matrix tension, and matrix compression failure modes, respectively.

Abaqus/CAE Usage: Property module: material editor: **Mechanical→Damage for Fiber-Reinforced Composites→Hashin Damage**: **Suboptions→Damage Evolution**: **Type: Energy**: **Softening: Linear**

Maximum degradation and choice of element removal

You have control over how Abaqus treats elements with severe damage. By default, the upper bound to all damage variables at a material point is $d_{max} = 1.0$. You can reduce this upper bound as discussed in “Controlling element deletion and maximum degradation for materials with damage evolution” in “Section controls,” Section 27.1.4.

By default, in Abaqus/Standard an element is removed (deleted) once damage variables for all failure modes at all material points reach d_{max} (see “Controlling element deletion and maximum degradation for materials with damage evolution” in “Section controls,” Section 27.1.4). In Abaqus/Explicit a material point is assumed to fail when either of the damage variables associated with fiber failure modes (tensile or compressive) reaches d_{max} and the element is removed from the mesh when this condition is satisfied at all of the section points at any one integration location of an element; for example, in the case of shell elements all through-the-thickness section points at any one integration location of the element must fail before the element is removed from the mesh. If an element is removed, the output variable STATUS is set to zero for the element, and it offers no resistance to subsequent deformation. Elements that have been removed are not displayed when you view the deformed model in the Visualization module of Abaqus/CAE (Abaqus/Viewer). However, the elements still remain in the Abaqus model. You can choose to display removed elements by suppressing use of the STATUS variable (see "Selecting the status field output variable," Section 42.5.6 of the Abaqus/CAE User's Manual, in the online HTML version of this manual).

Alternatively, you can specify that an element should remain in the model even after all of the damage variables reach d_{max}. In this case, once all the damage variables reach the maximum value, the stiffness, $\mathbf{C_d}$, remains constant (see the expression for $\mathbf{C_d}$ earlier in this section).

Difficulties associated with element removal in Abaqus/Standard

When elements are removed from the model, their nodes will still remain in the model even if they are not attached to any active elements. When the solution progresses, these nodes might undergo non-physical displacements due to the extrapolation scheme used in Abaqus/Standard to speed up the solution (see "Convergence criteria for nonlinear problems," Section 7.2.3). These non-physical displacements can be prevented by turning off the extrapolation. In addition, applying a point load to a node that is not attached to an active element will cause convergence difficulties since there is no stiffness to resist the load. It is the responsibility of the user to prevent such situations.

Viscous regularization

Material models exhibiting softening behavior and stiffness degradation often lead to severe convergence difficulties in implicit analysis programs, such as Abaqus/Standard. You can overcome some of these convergence difficulties by using the viscous regularization scheme, which causes the tangent stiffness matrix of the softening material to be positive for sufficiently small time increments.

In this regularization scheme a viscous damage variable is defined by the evolution equation:

$$\dot{d}_v = \frac{1}{\eta}(d - d_v),$$

where η is the viscosity coefficient representing the relaxation time of the viscous system and d is the damage variable evaluated in the inviscid backbone model. The damaged response of the viscous material is given as

$$\sigma = \mathbf{C_d}\epsilon,$$

where the damaged elasticity matrix, $\mathbf{C_d}$, is computed using viscous values of damage variables for each failure mode. Using viscous regularization with a small value of the viscosity parameter (small compared to the characteristic time increment) usually helps improve the rate of convergence of the model in the softening regime, without compromising results. The basic idea is that the solution of the viscous system relaxes to that of the inviscid case as $t/\eta \rightarrow \infty$, where t represents time.

Viscous regularization is also available in Abaqus/Explicit. Viscous regularization slows down the rate of increase of damage and leads to increased fracture energy with increasing deformation rates, which can be exploited as an effective method of modeling rate-dependent material behavior.

In Abaqus/Standard the approximate amount of energy associated with viscous regularization over the whole model or over an element set is available using output variable ALLCD.

Defining viscous regularization coefficients

You can specify different values of viscous coefficients for different failure modes.

Input File Usage: Use the following option to define viscous coefficients:

*DAMAGE STABILIZATION

η_{ft}, η_{fc}, η_{mt}, η_{mc}

where η_{ft}, η_{fc}, η_{mt}, η_{mc} are viscosity coefficients for fiber tension, fiber compression, matrix tension, and matrix compression failure modes, respectively.

Abaqus/CAE Usage: Use the following input to define the viscous coefficients for fiber-reinforced materials:

Property module: material editor: **Mechanical→Damage for Fiber-Reinforced Composites→Hashin Damage**: **Suboptions→Damage Stabilization**

Applying a single viscous coefficient in Abaqus/Standard

Alternatively, in Abaqus/Standard you can specify the viscous coefficients as part of a section controls definition. In this case the same viscous coefficient will be applied to all failure modes. For more information, see “Using viscous regularization with cohesive elements, connector elements, and elements that can be used with the damage evolution models for ductile metals and fiber-reinforced composites in Abaqus/Standard” in “Section controls,” Section 27.1.4.

Material damping

If stiffness proportional damping is specified in combination with the damage evolution law for fiber-reinforced materials, Abaqus calculates the damping stresses using the damaged elastic stiffness.

Elements

The damage evolution law for fiber-reinforced materials must be used with elements with a plane stress formulation, which include plane stress, shell, continuum shell, and membrane elements.

Output

In addition to the standard output identifiers available in Abaqus (“Abaqus/Standard output variable identifiers,” Section 4.2.1), the following variables relate specifically to damage evolution in the fiber-reinforced composite damage model:

STATUS	Status of the element (the status of an element is 1.0 if the element is active, 0.0 if the element is not). The value of this variable is set to 0.0 only if damage has occurred in all the damage modes.
DAMAGEFT	Fiber tensile damage variable.
DAMAGEFC	Fiber compressive damage variable.
DAMAGEMT	Matrix tensile damage variable.
DAMAGEMC	Matrix compressive damage variable.
DAMAGESHR	Shear damage variable.
EDMDDEN	Energy dissipated per unit volume in the element by damage.
ELDMD	Total energy dissipated in the element by damage.
DMENER	Energy dissipated per unit volume by damage.
ALLDMD	Energy dissipated in the whole (or partial) model by damage.
ECDDEN	Energy per unit volume in the element that is associated with viscous regularization.
ELCD	Total energy in the element that is associated with viscous regularization.
CENER	Energy per unit volume that is associated with viscous regularization.
ALLCD	The approximate amount of energy over the whole model or over an element set that is associated with viscous regularization.