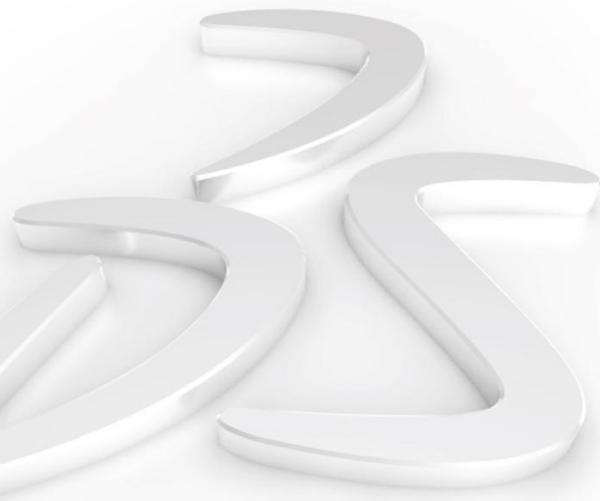


Analysis of Composite Materials with Abaqus

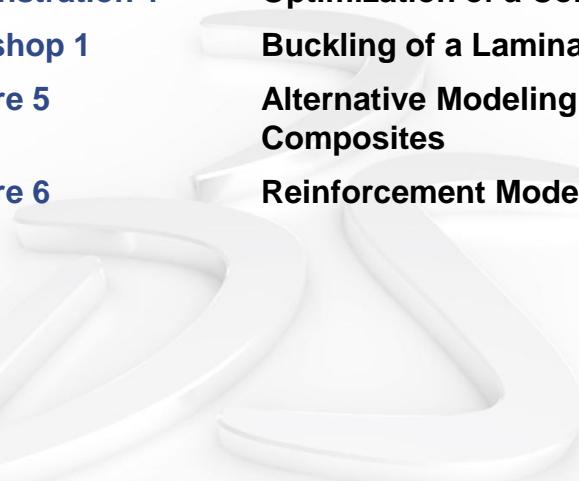
© DASSAULT SYSTEMES



Day 1

- | | |
|-------------------|-------------------------------------------------------|
| • Lecture 1 | Introduction |
| • Lecture 2 | Macroscopic Modeling |
| • Lecture 3 | Mixed Modeling |
| • Lecture 4 | Composite Modeling with Abaqus |
| • Demonstration 1 | Optimization of a Composite Tube |
| • Workshop 1 | Buckling of a Laminate Panel |
| • Lecture 5 | Alternative Modeling Techniques for Composites |
| • Lecture 6 | Reinforcement Modeling |

© DASSAULT SYSTEMES



Day 2

- | | |
|--------------|----------------------------------------------------------------|
| • Lecture 7 | Modeling of Sandwich Composites |
| • Lecture 8 | Modeling of Stiffened Panels |
| • Workshop 2 | Composite Yacht Hull |
| • Lecture 9 | Modeling Damage and Failure in Composites |
| • Lecture 10 | Cohesive Behavior |
| • Lecture 11 | Virtual Crack Closure Technique (VCCT) |
| • Lecture 12 | Low-cycle Fatigue |
| • Lecture 13 | Modeling Composite Material Impact with Abaqus/Explicit |
| • Workshop 3 | Perforation of a Composite Plate |

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Additional Material

- | | |
|--------------|---------------------------------------------------------------|
| • Appendix 1 | Crack Propagation Analysis using the Debond Capability |
| • Appendix 2 | Cohesive Element Modeling Techniques |
| • Appendix 3 | Modeling Issues for Continuum Shell Elements |

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Legal Notices

The Abaqus Software described in this documentation is available only under license from Dassault Systèmes and its subsidiary and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

© Dassault Systèmes, 2009.

Printed in the United States of America

Abaqus, the 3DS logo, SIMULIA and CATIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus 6.9 Release Notes and the notices at:

http://www.simulia.com/products/products_legal.html.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Revision Status

Lecture 1	3/09	Updated for 6.9
Lecture 2	3/09	Updated for 6.9
Lecture 3	3/09	Updated for 6.9
Lecture 4	3/09	Updated for 6.9
Lecture 5	3/09	Updated for 6.9
Lecture 6	3/09	Updated for 6.9
Lecture 7	3/09	Updated for 6.9
Lecture 8	3/09	Updated for 6.9
Lecture 9	3/09	Updated for 6.9
Lecture 10	3/09	Updated for 6.9
Lecture 11	3/09	Updated for 6.9
Lecture 12	3/09	Updated for 6.9
Lecture 13	3/09	Updated for 6.9
Appendix 1	3/09	Updated for 6.9
Appendix 2	3/09	Updated for 6.9
Appendix 3	3/09	Updated for 6.9
Demonstration 1	3/09	Updated for 6.9
Workshop 1	3/09	Updated for 6.9
Workshop 2	3/09	Updated for 6.9
Workshop 3	3/09	Updated for 6.9

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Notes

Notes

Introduction

Lecture 1

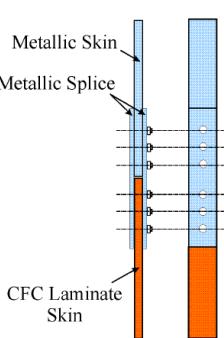
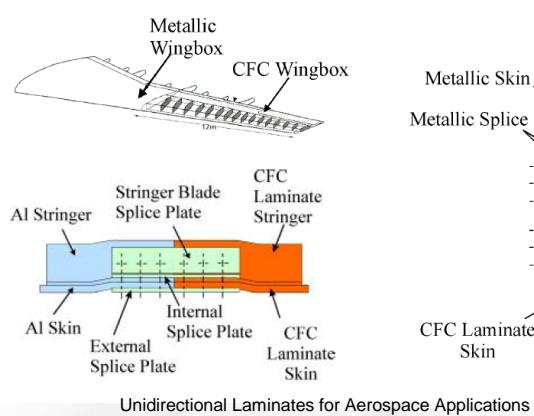
© DASSAULT SYSTEMES



L1.2

Overview

- Description of a Composite
- Some Typical Composites
- Finite Element Modeling of Composites



Ceramic Matrix Composites
for Turbomachinery



Woven Fabrics for Formula 1 Racing

© DASSAULT SYSTEMES

Description of a Composite

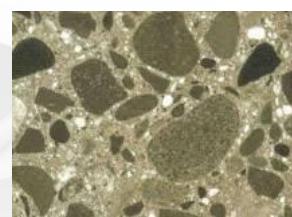
© DASSAULT SYSTEMES



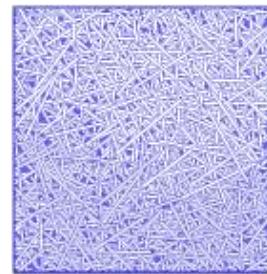
L1.4

Description of a Composite

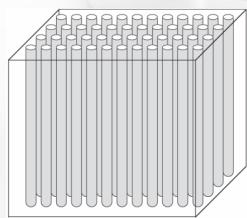
- In the context of this course a composite is a macroscopic mixture of at least two materials.
- One of the materials is the “matrix,” in which the other materials are embedded; the other materials are usually called “reinforcement.”
- The reinforcement can have many different forms:
 - Discrete, macroscopic particles
 - Randomly oriented fibers
 - Aligned fibers
 - Woven fabrics



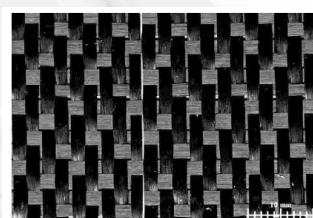
Discrete Particles



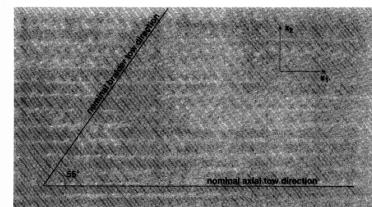
Random Fiber



Unidirectional Fiber



Woven Fiber



Braided Fiber



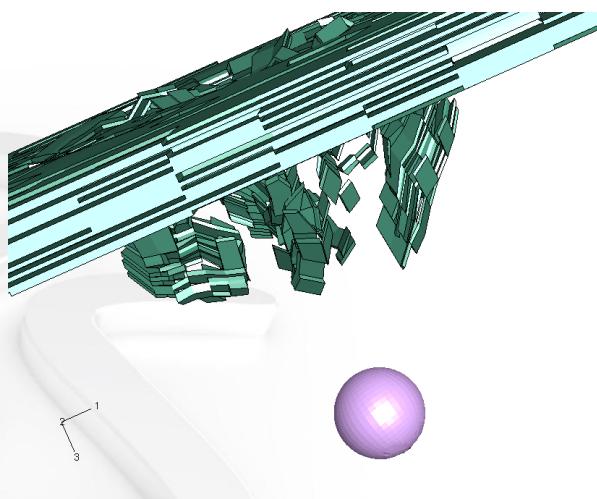
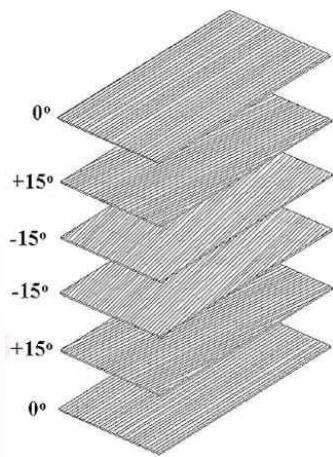
Some Typical Composites

© DASSAULT SYSTEMES



L1.6

Some Typical Composites

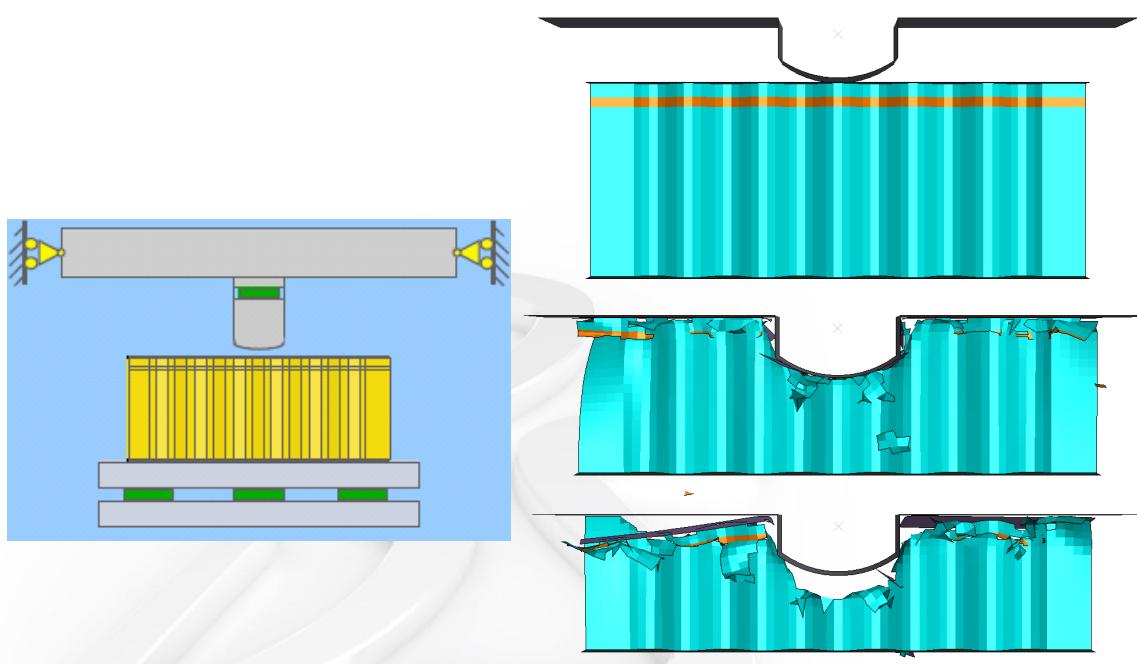


Unidirectional fiber composite plate - Ballistic impact analysis

© DASSAULT SYSTEMES

Some Typical Composites

© DASSAULT SYSTEMES



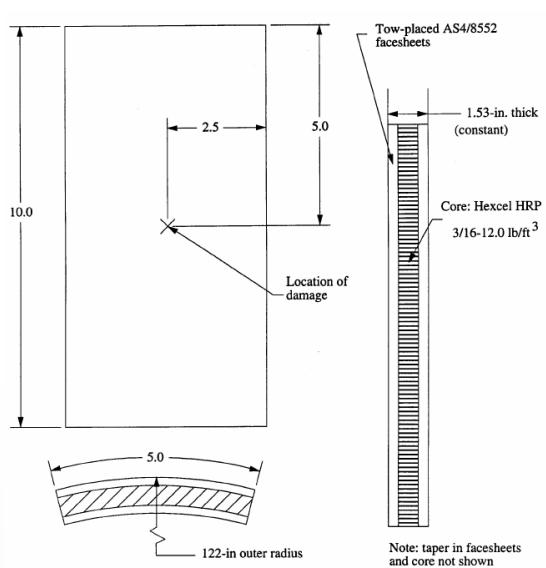
Woven fabric composite beam – Crushing analysis

SIMULIA

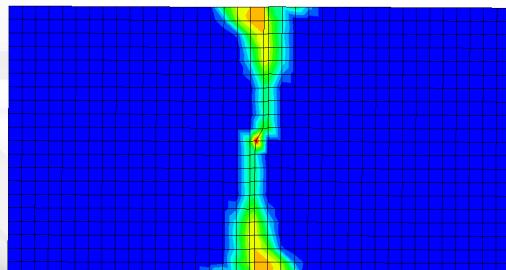
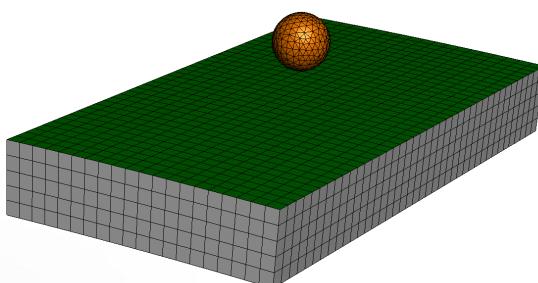
Analysis of Composite Materials with Abaqus

Some Typical Composites

© DASSAULT SYSTEMES



Geometry of test specimens. Dimensions are in inches.



Honeycomb core composite - Barely visible damage analysis

SIMULIA

Analysis of Composite Materials with Abaqus

Some Typical Composites

- The form and the material properties of the matrix and reinforcement have a strong influence on the characteristics of the composite.
- The purpose of varying matrix and reinforcement properties is to create a material that can exhibit:
 - Low Cost:
 - Prototypes
 - Mass production
 - Part consolidation
 - Maintenance
 - Maturity of technology
 - Desired Weight:
 - Light weight
 - Specific weight distribution
 - Improved Strength and Stiffness:
 - High strength-to-weight ratio
 - Directional strength and/or stiffness
 - Improved Surface Properties:
 - Corrosion resistance
 - Tailored surface finish
 - Desired Thermal Properties:
 - Low thermal conductivity
 - Low coefficient of thermal expansion
 - Unique Electric Properties:
 - High dielectric strength
 - Non-magnetic
 - Radar transparency
 - Dimensional Flexibility:
 - Large parts
 - Special geometry
- These goals are often competing and cannot always be realized simultaneously.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



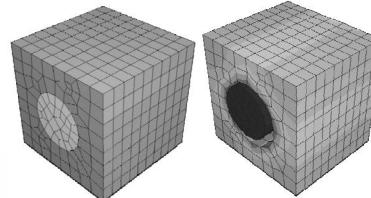
Finite Element Modeling of Composites

© DASSAULT SYSTEMES

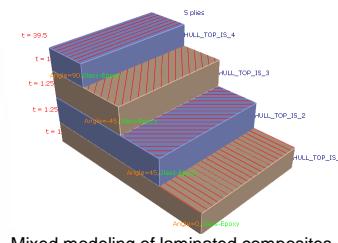


Finite Element Modeling of Composites

- Depending on the purpose of the analysis, different modeling techniques for composites can be used:
 - Microscopic modeling
 - The matrix and reinforcement material are both modeled separately as deformable continua.
 - This topic is not covered in this course.
 - Macroscopic modeling
 - The composite is modeled as a single orthotropic material or a single fully anisotropic material.
 - Mixed modeling
 - The composite is modeled by a number of discrete, macroscopically modeled reinforced layers.
 - Discrete reinforcement modeling
 - Reinforcement modeled with discrete elements or other modeling tools (e.g., rebar).
 - Submodeling
 - Useful for studying stress concentrations around the tips of reinforcing fibers



Unit cell model of fiber-matrix delamination



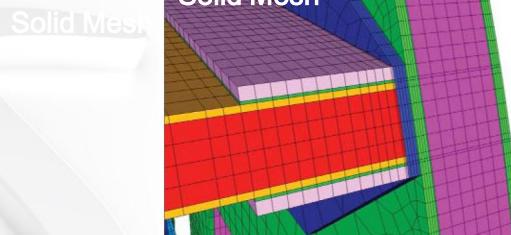
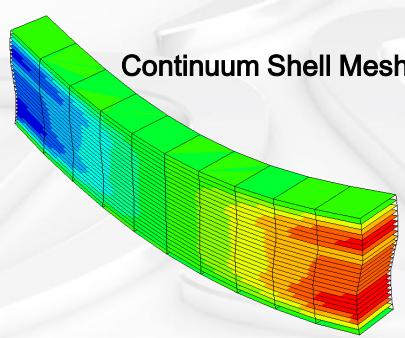
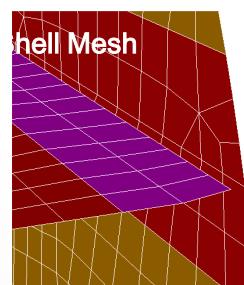
Mixed modeling of laminated composites



Analysis of Composite Materials with Abaqus

Finite Element Modeling of Composites

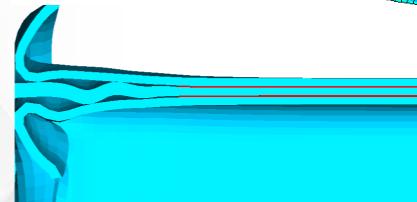
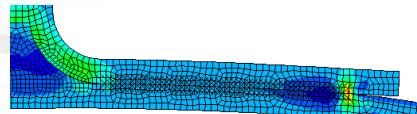
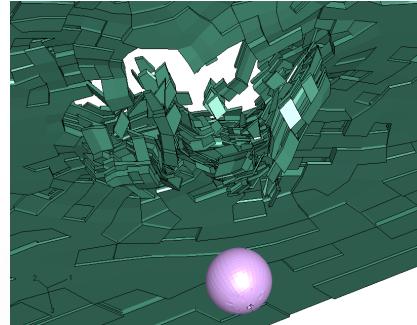
- For a large majority of finite element simulations, composites are modeled as:
 - Layered-shells, layered-solids
 - Stacked solid elements
 - Stacked or layered continuum shells



Analysis of Composite Materials with Abaqus

Finite Element Modeling of Composites

- In addition to the modeling of matrix and reinforcement, progressive damage and failure of the materials and their interfaces can be modeled as well.
 - Progressive Damage and Failure—Prediction of failure modes for both fiber and matrix materials
 - Hashin Criteria
 - UMAT (Abaqus/Standard)
 - VUMAT (Abaqus/Explicit)
 - Delamination—Separation of adhesively bonded sections of laminated composites
 - Virtual Crack Closure Technique (VCCT)
 - Cohesive Elements
 - Cohesive Contact



 SIMULIA

Analysis of Composite Materials with Abaqus

Notes

Notes

Macroscopic Modeling

Lecture 2



L2.2

Overview

- **Introduction**
- **Some Notes on Anisotropic Elasticity**
- **Thermal Expansion**
- **Material Orientation**
- **Almost Incompressible Behavior**

Introduction

© DASSAULT SYSTEMES



L2.4

Introduction

- In this technique the composite is modeled as a single orthotropic material or a single fully anisotropic material.
- The composite is usually considered elastic.
 - In addition, Hill's anisotropic plasticity model is sometimes used to model inelastic deformation.
 - The reinforcements and elements do not need to be aligned.
 - The deformation field is homogeneous.

© DASSAULT SYSTEMES



Introduction

- Macroscopic analysis is used to model the overall behavior of structural components built out of composites.
 - Nonlinear material behavior and local failure often are not considered because of the complex nature of modeling these effects
 - Note: Abaqus does have progressive damage and delamination/decohesion modeling capabilities, if these effects are important.
 - Structural failure (buckling and collapse) is commonly studied without taking into account material failure such as delamination.
 - Post-analysis checks are used to establish whether this approach is acceptable.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Some Notes on Anisotropic Elasticity

© DASSAULT SYSTEMES



Some Notes on Anisotropic Elasticity

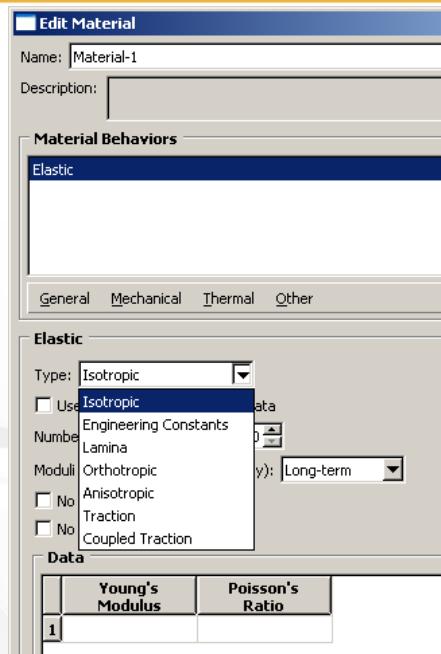
- For the macroscopic modeling of elastic composites, it is essential to define anisotropic elasticity coefficients accurately.
 - Improper specification leads to incorrect results or can even lead to loss of material stability.
- In Abaqus several types of anisotropic elastic behavior are available.
 - All anisotropic models have the general form

$$\sigma_{ij} = D_{ijkl} (\varepsilon_{kl} - \varepsilon_{kl}^{th}) \quad \text{or} \quad \boldsymbol{\sigma} = \mathbf{D} : (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{th}).$$

where $\mathbf{D} = \mathbf{D}(\theta, f_i)$ is a symmetric matrix with a maximum dimension of 6×6 , θ = temperature, f_i = predefined field variables, and $\boldsymbol{\varepsilon}^{th} = \boldsymbol{\varepsilon}^{th}(\theta)$ is the strain due to thermal expansion.

Some Notes on Anisotropic Elasticity

- The anisotropic elastic moduli are defined in Abaqus using linear elasticity.
- The elasticity matrix \mathbf{D} is defined for:
 - The various material symmetries: lamina, orthotropic, and anisotropic
 - Different temperatures and field variables
- Since Abaqus provides a convenient material orientation option, material symmetries generally are used to the analyst's advantage by specifying the material's elastic properties, even though the symmetries are not aligned with the global axes.



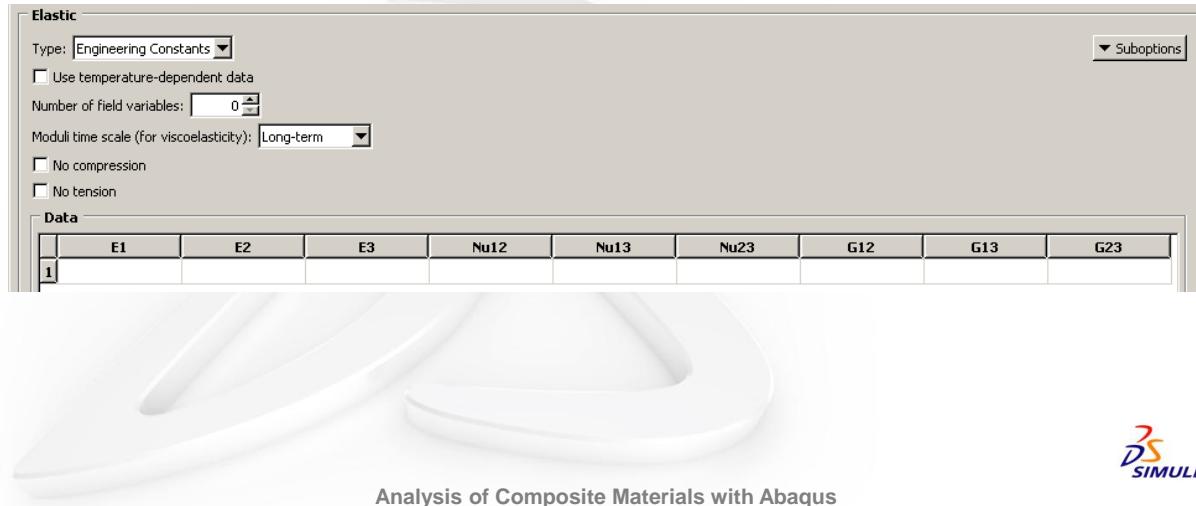
```
*MATERIAL, name=Material-1
*ELASTIC, TYPE=...
```

Some Notes on Anisotropic Elasticity

- The different anisotropic input options for linear elastic behavior are:

***ELASTIC, TYPE=ENGINEERING CONSTANTS**

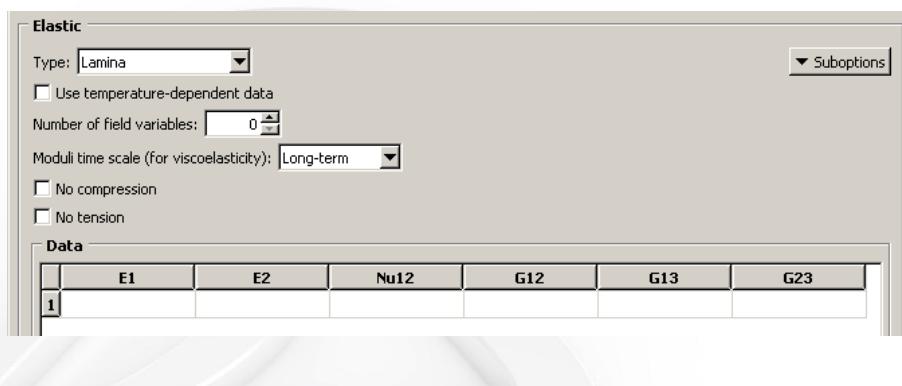
- This option is for orthotropic materials.
- It requires $E_1, E_2, E_3, \nu_{12}, \nu_{13}, \nu_{23}, G_{12}, G_{13}, G_{23}$.



Some Notes on Anisotropic Elasticity

***ELASTIC, TYPE=LAMINA**

- This is the same as TYPE=ENGINEERING CONSTANTS but is used specifically for plane stress, such as in laminated shells.
- It requires specification of $E_1, E_2, \nu_{12}, G_{12}, G_{13}, G_{23}$.

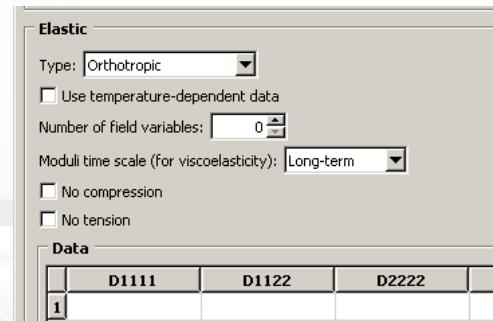


Some Notes on Anisotropic Elasticity

*ELASTIC, TYPE=ORTHOtROPIC

- Requires direct specification of all nonzero terms in the D matrix for the orthotropic case:

$$\begin{bmatrix} D_{1111} & D_{1122} & D_{1133} & 0 & 0 & 0 \\ D_{2222} & D_{2233} & 0 & 0 & 0 & 0 \\ D_{3333} & 0 & 0 & 0 & 0 & 0 \\ & D_{1212} & 0 & 0 & 0 & 0 \\ \text{sym} & & D_{1313} & 0 & 0 & 0 \\ & & & D_{2323} & 0 & 0 \end{bmatrix}$$



© DASSAULT SYSTEMES



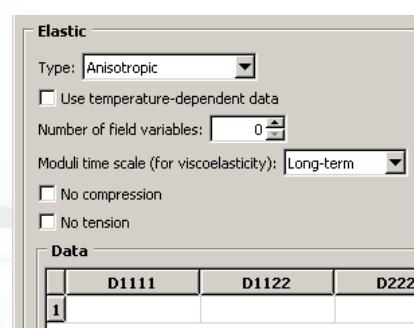
Analysis of Composite Materials with Abaqus

Some Notes on Anisotropic Elasticity

*ELASTIC, TYPE=ANISOTROPIC

- Requires direct specification of all nonzero terms in the D matrix for the completely anisotropic case:

$$\begin{bmatrix} D_{1111} & D_{1122} & D_{1133} & D_{1112} & D_{1113} & D_{1123} \\ D_{2222} & D_{2233} & D_{2212} & D_{2213} & D_{2223} & \\ D_{3333} & D_{3312} & D_{3313} & D_{3323} & & \\ & D_{1212} & D_{1213} & D_{1223} & & \\ \text{sym} & & D_{1313} & D_{1323} & & \\ & & & D_{2323} & & \end{bmatrix}$$



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Some Notes on Anisotropic Elasticity

- For an orthotropic material with engineering constants, the Poisson's ratios, ν_{ij} , obey the following relations:

$$\frac{\nu_{ij}}{E_i} = \frac{\nu_{ji}}{E_j}$$

or in expanded form,

$$\frac{\nu_{12}}{E_1} = \frac{\nu_{21}}{E_2}, \quad \frac{\nu_{13}}{E_1} = \frac{\nu_{31}}{E_3}, \quad \frac{\nu_{23}}{E_2} = \frac{\nu_{32}}{E_3},$$

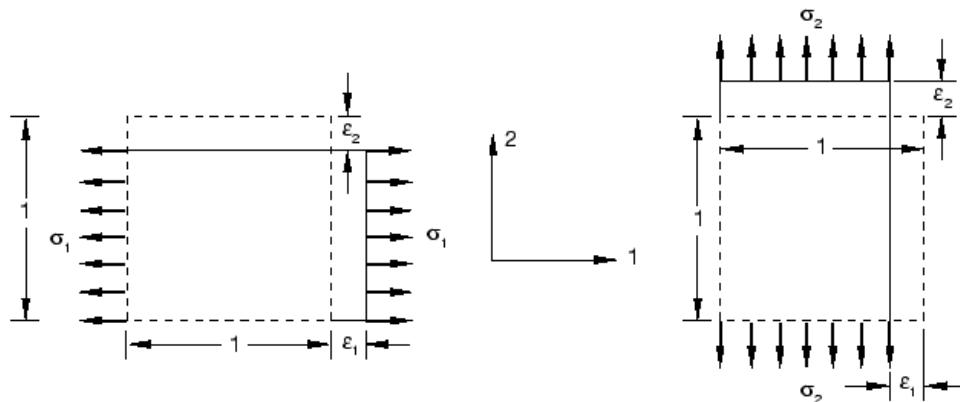
where

$$\nu_{ij} = -\frac{\varepsilon_j}{\varepsilon_i}$$

is the Poisson's ratio that defines the transverse strain in the j -direction when the material is stressed in the i -direction.

Some Notes on Anisotropic Elasticity

- The user should be careful in distinguishing between ν_{ij} and ν_{ji} .
- For example, to determine ν_{12} and ν_{21} we do the following two simple uniaxial tests:



$$\nu_{12} = -\frac{\varepsilon_2}{\varepsilon_1}$$

$$\nu_{21} = -\frac{\varepsilon_1}{\varepsilon_2}$$

Some Notes on Anisotropic Elasticity

- For an orthotropic material the “engineering constants” define the D matrix as:

$$\begin{aligned} D_{1111} &= E_1(1 - \nu_{23}\nu_{32})\Upsilon \\ D_{2222} &= E_2(1 - \nu_{13}\nu_{31})\Upsilon \\ D_{3333} &= E_3(1 - \nu_{12}\nu_{21})\Upsilon \\ D_{1122} &= E_1(\nu_{21} + \nu_{31}\nu_{23})\Upsilon = E_2(\nu_{12} + \nu_{32}\nu_{13})\Upsilon \\ D_{1133} &= E_1(\nu_{31} + \nu_{21}\nu_{32})\Upsilon = E_3(\nu_{13} + \nu_{12}\nu_{23})\Upsilon \\ D_{2233} &= E_2(\nu_{32} + \nu_{12}\nu_{31})\Upsilon = E_3(\nu_{23} + \nu_{21}\nu_{13})\Upsilon \\ D_{1212} &= G_{12} \\ D_{1313} &= G_{13} \\ D_{2323} &= G_{23} \end{aligned}$$

where

$$\Upsilon = \frac{1}{1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{31}\nu_{13} - 2\nu_{21}\nu_{32}\nu_{13}}.$$

Some Notes on Anisotropic Elasticity

- Certain restrictions apply to the material specification to obtain valid elastic behavior.
- The following are sufficient conditions to ensure material stability (that is, for D to be positive-definite):
 - Orthotropic, plane stress material (TYPE=LAMINA):

$$E_1, E_2, G_{12}, G_{13}, G_{23} > 0$$

$$|\nu_{12}| < \left(\frac{E_1}{E_2} \right)^{1/2}$$

Some Notes on Anisotropic Elasticity

- General orthotropic material (TYPE=ENGINEERING CONSTANTS):

$$E_1, E_2, E_3, G_{12}, G_{13}, G_{23} > 0$$

$$|\nu_{12}| < \left(\frac{E_1}{E_2} \right)^{1/2}, \quad |\nu_{13}| < \left(\frac{E_1}{E_3} \right)^{1/2}, \quad |\nu_{23}| < \left(\frac{E_2}{E_3} \right)^{1/2}$$

$$1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{31}\nu_{13} - 2\nu_{21}\nu_{32}\nu_{13} > 0$$

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Some Notes on Anisotropic Elasticity

- Orthotropic material (TYPE=ORTHOTROPIC):

$$D_{1111}, D_{2222}, D_{3333}, D_{1212}, D_{1313}, D_{2323} > 0$$

$$|D_{1122}| < (D_{1111}D_{2222})^{1/2}$$

$$|D_{1133}| < (D_{1111}D_{3333})^{1/2}$$

$$|D_{2233}| < (D_{2222}D_{3333})^{1/2}$$

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Some Notes on Anisotropic Elasticity

- Anisotropic material (TYPE=ANISOTROPIC):
 - The conditions are too complex to express in simple relations.
 - The requirement that D is positive-definite means that all six eigenvalues of D must be positive.
 - This should be ascertained numerically before the material description is used in an Abaqus analysis.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Thermal Expansion

© DASSAULT SYSTEMES



Thermal Expansion

- In Abaqus isotropic as well as anisotropic thermal expansion can be specified. The anisotropic input cases are:

***EXPANSION, TYPE=ORTHOTROPIC**

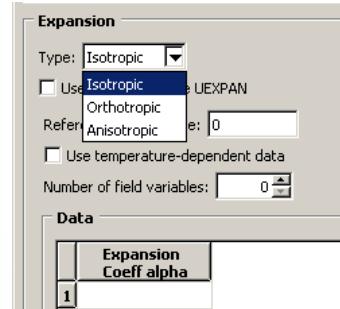
for orthotropic materials. The thermal expansion in the principal material directions must be specified: α_{11} , α_{22} , α_{33} .

***EXPANSION, TYPE=ANISOTROPIC**

for fully anisotropic materials. The thermal expansion in all directions must be specified:

α_{11} , α_{22} , α_{33} , α_{12} , α_{13} , α_{23} .

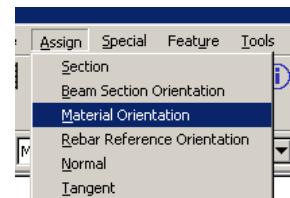
- There are no restrictions for the thermal expansion coefficients: negative values are allowed.
- For plane stress elements and shells α_{33} is not used.



Material Orientation

Material Orientation

- When anisotropic material behavior is defined, local material directions (*ORIENTATION) **must** be defined.
 - If local material directions are used with isotropic materials, they will only affect the element output.



Property module

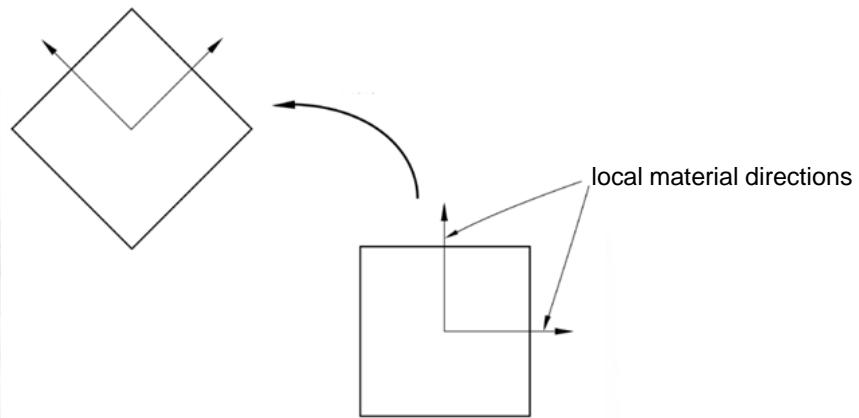
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Material Orientation

- In geometrically nonlinear analysis the local material directions rotate with the average spin of the material.
- Element output such as stress and strain will be in the (corotational) material directions.



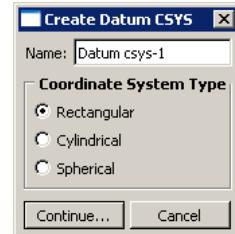
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Material Orientation

- Abaqus offers several options to specify the orientation of the material directions.
- The available coordinate systems are
 - RECTANGULAR,
 - CYLINDRICAL,
 - SPHERICAL, and
 - Z RECTANGULAR.



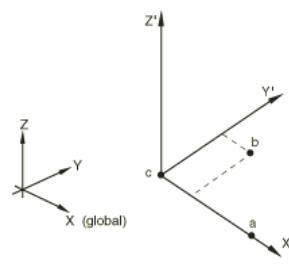
*ORIENTATION, SYSTEM=...

© DASSAULT SYSTEMES

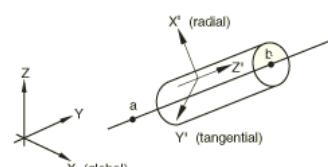


Analysis of Composite Materials with Abaqus

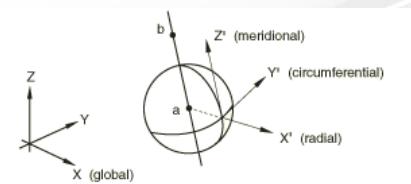
Material Orientation



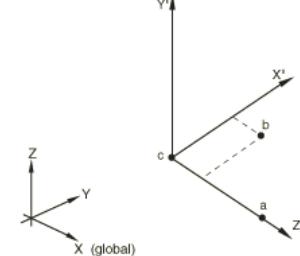
Rectangular



Cylindrical



Spherical



Z-rectangular

Coordinate systems for material orientation

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Material Orientation

- Additional comments on the keywords interface

- The orientation and position of the local coordinate system is defined by the coordinates of two points, *a* and *b*.
 - The DEFINITION parameter specifies how the coordinates of these points are defined.
 - The options are NODES, COORDINATES, and OFFSET TO NODES.
- If no further data are given, the principal axes of the material will be aligned with the local coordinate system.
 - Optionally, the principal axes of the material can be rotated around one of the local coordinate directions.
- If the orientation is too complex to be specified with these options, user subroutine ORIENT can be invoked with the parameter SYSTEM=USER.

© DASSAULT SYSTEMES

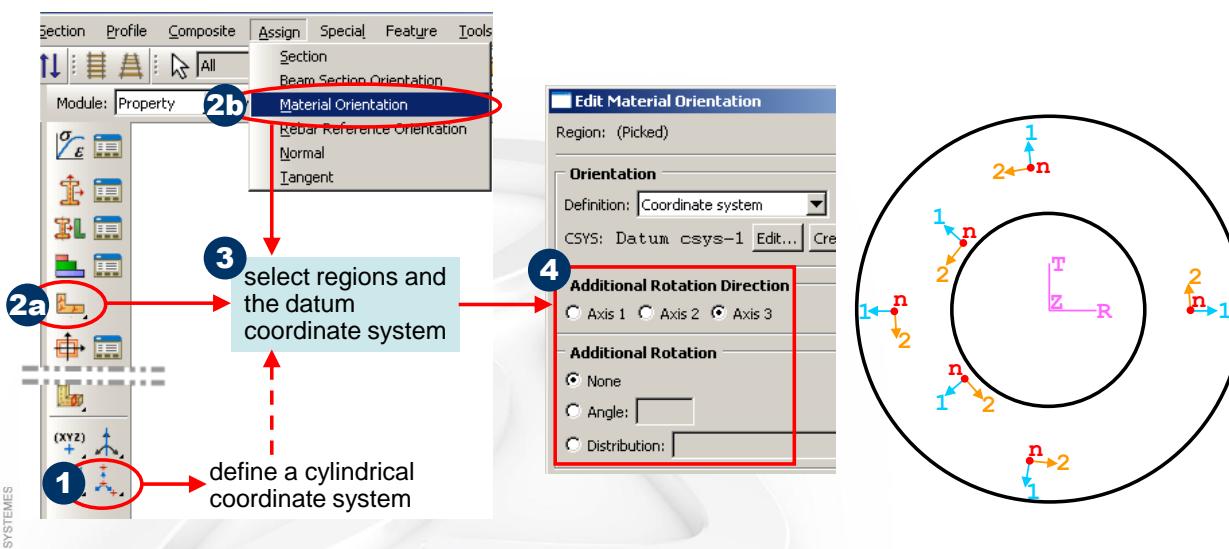


Analysis of Composite Materials with Abaqus

Material Orientation

- Example: Cylindrical orientation

- Abaqus/CAE interface



© DASSAULT SYSTEMES

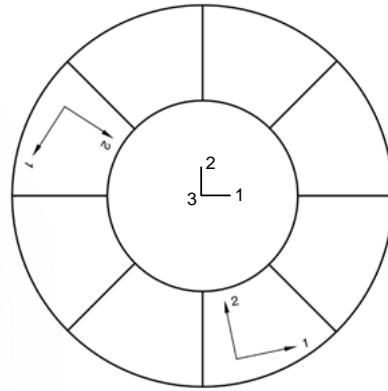


Analysis of Composite Materials with Abaqus

Material Orientation

- Keywords interface
 - The pertinent input data are:

```
*MATERIAL, NAME=MAT1
*ELASTIC, TYPE=LAMINA
  30.E6, 10.E6, .3,15.E6, 20.E6, 20.E6
*SHELL SECTION, ELSET=SHELLS1,
  MATERIAL=MAT1, ORIENTATION=CYL1
  1., 5
*ORIENTATION, NAME=CYL1,
  SYSTEM=CYLINDRICAL
  0., 0., 0., 0., 0., 1.
  3, 90.
```



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Almost Incompressible Behavior

© DASSAULT SYSTEMES



Almost Incompressible Behavior

- For certain types of composites the elastic bulk modulus, K , is much larger than the effective elastic shear modulus, G_{eff} : the material is essentially incompressible.
- We should check this since, except for plane stress cases, almost incompressible behavior requires the use of special element formulations.
- First, we need values for K and G_{eff} , or we can estimate the effective Poisson's ratio.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Almost Incompressible Behavior

- For a general, anisotropic, linear elastic stress-strain law we can compute the bulk modulus from its definition:

$$K = -\partial p / \partial \varepsilon_{vol},$$

where the equivalent pressure stress, p , is

$$p \stackrel{\text{def}}{=} -\frac{1}{3} \mathbf{I} : \boldsymbol{\sigma} = -\frac{1}{3} \mathbf{I} : \mathbf{D} : \boldsymbol{\varepsilon}$$

and the elastic strain, $\boldsymbol{\varepsilon}$, is decomposed into its volumetric and deviatoric parts:

$$\boldsymbol{\varepsilon} = \frac{1}{3} \mathbf{I} \varepsilon_{vol} + \boldsymbol{\varepsilon}_{dev},$$

where ε_{vol} is a scalar quantity.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Almost Incompressible Behavior

- Therefore,

$$K = -\partial p / \partial \epsilon_{vol} = \frac{1}{3} \mathbf{I} : \mathbf{D} : \partial \boldsymbol{\epsilon} / \partial \epsilon_{vol},$$

and since $\partial \boldsymbol{\epsilon} / \partial \epsilon_{vol} = \frac{1}{3} \mathbf{I}$, the bulk modulus is

$$K = \frac{1}{9} \mathbf{I} : \mathbf{D} : \mathbf{I}.$$

- For example, for TYPE=ENGINEERING CONSTANTS

$$K = \frac{1}{\left(\frac{1-\nu_{12}-\nu_{13}}{E_1} + \frac{1-\nu_{21}-\nu_{23}}{E_2} + \frac{1-\nu_{31}-\nu_{32}}{E_3} \right)}.$$

- A conservative (low) estimate of the effective elastic shear modulus can be obtained by taking the minimum of G_{12} , G_{13} , and G_{23} .

Almost Incompressible Behavior

- The ratio of bulk modulus to effective shear modulus is related to the effective Poisson's ratio, ν_{eff} , as:

ν_{eff}	K/G_{eff}
0.3	2
0.4	5
0.45	10
0.49	50
0.495	100
0.499	500
0.4995	1000
0.4999	5000

- Alternatively, a conservative (high) estimate of Poisson's ratio can be obtained by taking the maximum of $\sqrt{\nu_{12}\nu_{21}}$, $\sqrt{\nu_{13}\nu_{31}}$, and $\sqrt{\nu_{23}\nu_{32}}$.
- We can use these values to determine whether the material is approximately incompressible.

Almost Incompressible Behavior

- If $K/G_{eff} > 100$ or $\nu_{eff} > 0.495$, the stiffness ratio between volumetric and shear deformation is too high to use standard displacement formulation elements: mesh locking is likely to occur.
- Reduced integration elements can be used to avoid locking effects.
 - Abaqus uses selectively reduced integration for the standard lower-order quadrilateral and brick elements (CPE4, CAX4, C3D8, etc.), which also prevents locking.
- The alternative is to use “hybrid” (mixed formulation) elements (element types CxxH, such as C3D20H).
 - Use of hybrid elements is necessary if the material is essentially incompressible ($K \rightarrow \infty$).
- For any plane stress application (beams, shells, and continuum plane stress elements), standard displacement elements can always be used for any value of ν_{eff} .

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Notes

Notes

Mixed Modeling

Lecture 3

© DASSAULT SYSTEMES



L3.2

Overview

- Introduction
- Laminated Composite Shells
- Continuum Shell Elements
- Continuum Shell Meshing
- Continuum Solid Elements
- Symmetry Conditions and Laminated Structures

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES

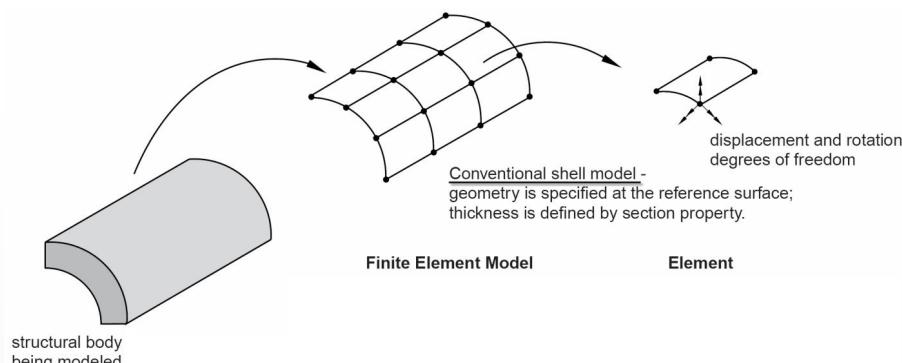


 SIMULIA

L3.4

Introduction

- With the mixed modeling technique, the composite is modeled by a number of discrete layers, each with orthotropic or fully anisotropic material properties.
 - Commonly use laminated shells for this purpose.

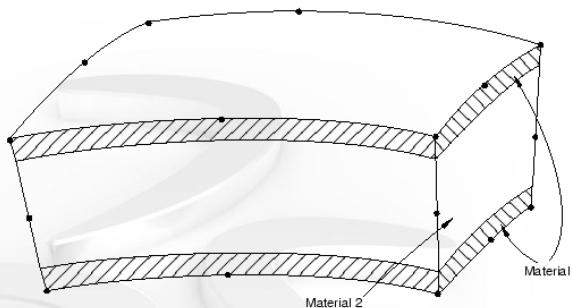


- In composites, the layers usually are considered to be elastic, although inelastic material properties can be used.

© DASSAULT SYSTEMES

Introduction

- The mixed modeling technique can also be used for layered continua.
 - Three-dimensional hexahedral elements with displacement degrees of freedom (Abaqus/Standard only) can be defined with any number of different layers in any direction.

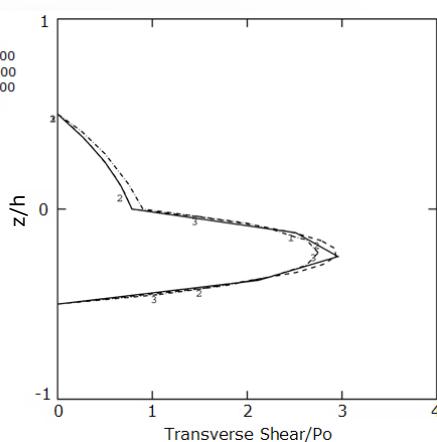


- However, this layered solid element capability cannot estimate transverse shear stresses as accurately as the thick shell elements.
 - Its usage generally is confined to cases where bending effects (and, hence, delamination caused by shearing) are not significant.

Introduction

- The highest-order solid elements in Abaqus provide quadratic displacement interpolation.
 - Hence, the strains vary linearly in any direction.
- But in a laminated shell section the transverse shear stress is zero on free surfaces and may vary rapidly through the section.

LINE VARIABLE	SCALE Factor
1 SBR	+1.00E+00
2 CPT	+1.00E+00
3 Elasticity	+1.00E+00



Transverse shear stress distribution through a two-layer plate

Introduction

- Clearly a linear strain/stress variation through the thickness cannot provide any reasonable approximation to this response.
- If transverse shear stress effects must be modeled and solid elements are required, there must be enough elements through the thickness to capture the stress variation.
- Thus, the recommended method for modeling a composite material is with shell elements.
 - The basic features of shell elements are the focus of this lecture.
 - Detailed modeling techniques (i.e., assigning layups, properties, etc. to the elements) will be discussed in subsequent lectures.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



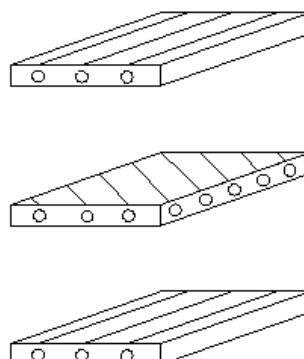
Laminated Composite Shells

© DASSAULT SYSTEMES



Laminated Composite Shells

- The shells in Abaqus support multilayer construction. The user may define:
 - A number of layers or laminae
 - Independent material and orientation in each layer
 - A different number of integration points used for Simpson's integration through each layer



Typical laminate construction

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Composite Shells

- Abaqus offers two types of shell elements
 - Conventional (where only the shell reference surface is discretized)
 - Continuum (where a 3D volume is discretized but the kinematic behavior of the element is based on shell theory).
 - Both can be used to model composite constructions.
- In addition, Abaqus offers meshed beam cross sections that can be used to model composite beam structures.
 - Not discussed further in this course.
 - See the “Element Selection in Abaqus” lecture notes for more information.

© DASSAULT SYSTEMES

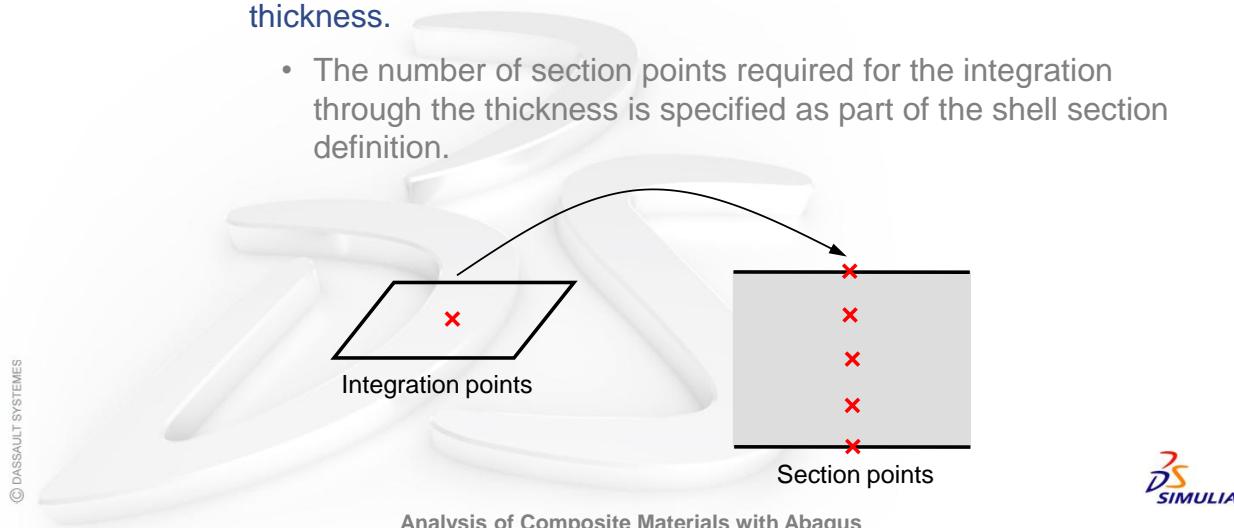


Analysis of Composite Materials with Abaqus

Laminated Composite Shells

- Some comments regarding shell elements in Abaqus
 - Section points vs. integration points in shell elements
 - Integration points refer to the integration positions in the plane of the shell.
 - Section points refer to the integration positions through the shell thickness.
 - The number of section points required for the integration through the thickness is specified as part of the shell section definition.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Composite Shells

- Pre-integration vs. runtime integration of shell section properties
 - Pre-integrated shell section properties are computed by Abaqus prior to the analysis and are not updated during the analysis.
 - No nonlinear material properties can be included.
 - Runtime integration of the shell section properties implies they are calculated by numerical integration through the shell thickness during the analysis, thus providing complete generality in material modeling.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Composite Shells

- Material modeling
 - Anisotropic linear elasticity is the most commonly used material model for laminated composite shells (for example, to simulate high-stiffness fibers macroscopically along different orientations in the layers of the shell).

© DASSAULT SYSTEMES



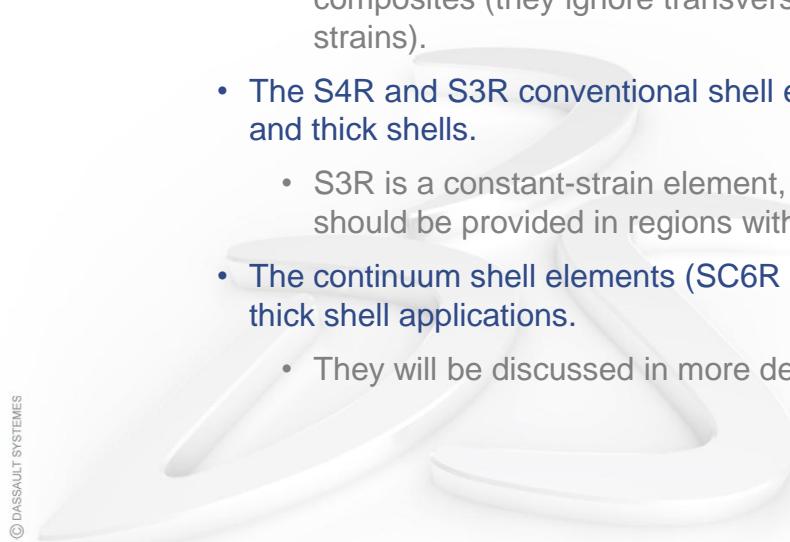
Analysis of Composite Materials with Abaqus



Laminated Composite Shells

- Element selection
 - Abaqus offers thin (STR13, S8R5, and S9R5) and thick (S8R) conventional shell elements.
 - The thin shell elements, however, are generally not suitable for composites (they ignore transverse flexibility and assume small strains).
 - The S4R and S3R conventional shell elements can model both thin and thick shells.
 - S3R is a constant-strain element, so sufficient mesh refinement should be provided in regions with high-strain gradients.
 - The continuum shell elements (SC6R and SC8R) are intended for thick shell applications.
 - They will be discussed in more detail later.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

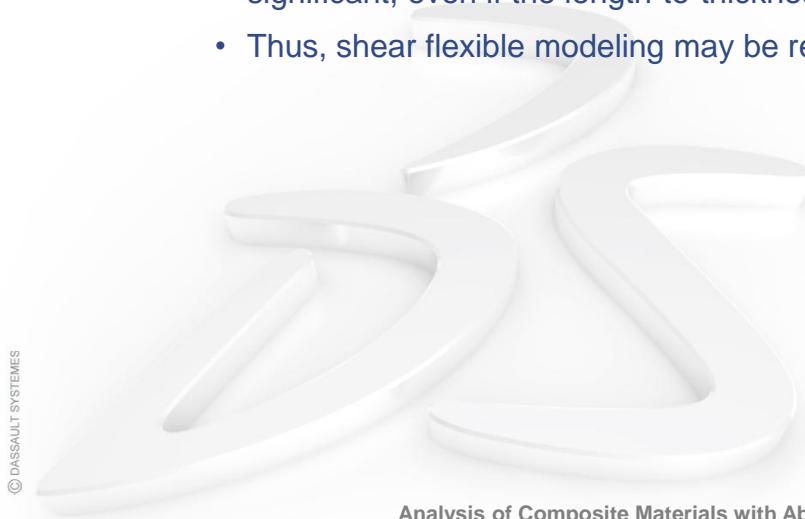


Laminated Composite Shells

- **Transverse shear**

- Shells made out of isotropic materials can be modeled without considering transverse shear effects, as long as the length-to-thickness ratios are large enough (i.e., greater than around 20).
 - In laminated shells, however, transverse shear effects can be significant, even if the length-to-thickness ratio is large.
 - Thus, shear flexible modeling may be required.

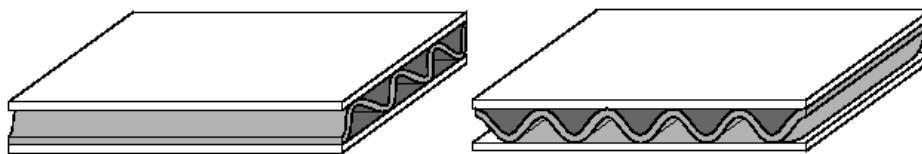
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

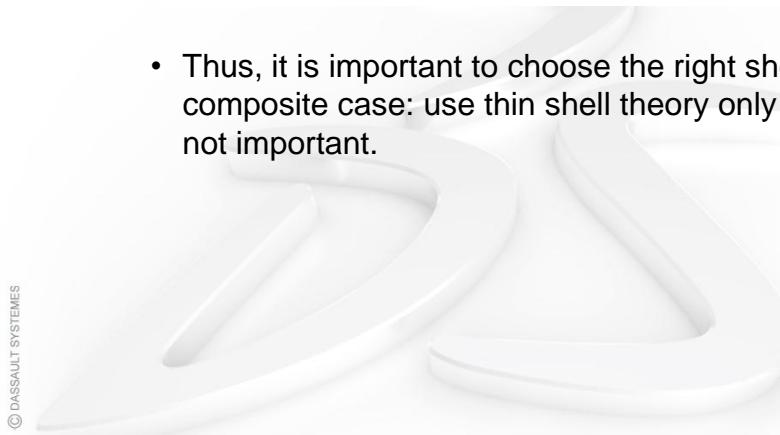
Laminated Composite Shells

- This is true especially for sandwich construction shells made from stiff skins with a soft core:



- Thus, it is important to choose the right shell formulation for a laminated composite case: use thin shell theory only if transverse shear flexibility is not important.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Composite Shells

- Abaqus uses the following strain distributions through the thickness of shell elements:
 - Membrane strains vary linearly.
 - This results in proper membrane and bending behavior.
 - Transverse shear strains are assumed constant through the shell's thickness.
 - The transverse shear stresses are zero at the shell surface. Furthermore, the transverse shear stresses between layers are continuous. Therefore, the constitutive equations alone cannot accurately model the true variation of transverse shear stress through the thickness.
 - The formulation (described next) for the transverse shear stiffness and stress calculations for laminates in element types S3R, S4R, S8R, SC6R, and SC8R properly accounts for all of these issues so long as elastic response is assumed.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Laminated Composite Shells

- **Basic theory**
 - The basis of this formulation is described in the Abaqus Theory Manual. It is summarized as follows:
 - We consider a plate in the $x-y$ plane, subject to bending and shear in the $x-z$ plane but with no primary (membrane) loading.
 - We assume there are no gradients of any function in the y -direction (so that any slice in any $x-z$ plane is the same).
 - This is an approximation; for example, the bending in the $x-z$ plane will induce bending in $y-z$ sections as a result of Poisson's effects.
 - We also assume that $M_{yy} = M_{xy} = 0$; that is, the bending in the $x-z$ sections do not induce any moments in the $y-z$ section or any twist.
 - Again, this is an approximation if we have an unbalanced section.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Laminated Composite Shells

- These assumptions simplify the elasticity problem sufficiently so that we have an expression for the gradient of the transverse shear stress anywhere through the shell's thickness as a function of the transverse shear force on the section, V_x :

$$\frac{\partial \tau_{xz}}{\partial z} = (B_{x1} - (z - z_0)B_{x2})V_x,$$

where B_{x1} and B_{x2} are defined from the shell section properties at the layer in which the thickness coordinate, z , is currently located. Thus, B_{x1} and B_{x2} are constants in each layer of the shell.

- This expression can be integrated through the thickness of the shell to obtain τ_{xz} as a function of z . Since the derivative is linear in z and the B_{x1} , B_{x2} are constant within each layer, we see that the transverse shear stress varies parabolically within each layer.

Laminated Composite Shells

- Likewise, the variation of the other component of transverse shear stress τ_{yz} can be found from the V_y component of transverse shear force, based on similar results written for deformation in the $y-z$ section.
- The final step is to define the transverse shear stiffness of the section.
 - This is done by equating the energy of the solution expressed in terms of transverse shear forces to that of the transverse shear stress distribution:

$$\frac{1}{2} [V_x \quad V_y] [F^s] \begin{Bmatrix} V_x \\ V_y \end{Bmatrix} = \frac{1}{2} \sum_{i=1}^N \int_{z_i}^{z_{i+1}} [\tau_{xz} \quad \tau_{yz}] [F^i] \begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} dz,$$

where $[F^s]$ and $[F^i]$ are the transverse shear flexibilities of the section and layer i , respectively.

Laminated Composite Shells

- This defines the section's 2×2 transverse shear flexibility,

$$\begin{bmatrix} F_{xx}^s & F_{xy}^s \\ F_{yx}^s & F_{yy}^s \end{bmatrix},$$

the inverse of which is the section's transverse shear stiffness.

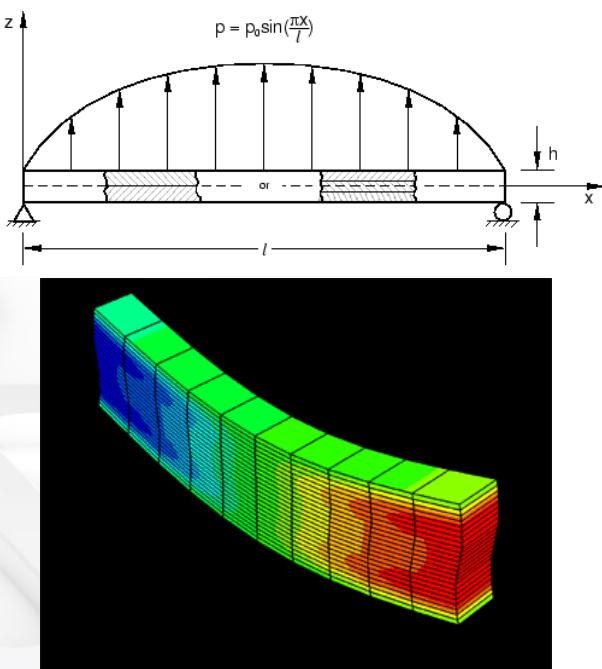
Laminated Composite Shells

- Transverse shear stress output**

- Estimates of transverse shear stress based on the theory outlined above are available. The output variable depends on the shell element used in the model.
 - Request variables TSHR13 and TSHR23 for conventional shell elements that allow shear flexibility:
 - S3(R)(S), S4(R)(S)(W), and S8R(T)
 - Request variables CTSHR13 and CTSHR23 for stacked continuum shell elements
 - SC6R and SC8R
 - TSHR13 and TSHR23 (or CTSHR13 and CTSHR23) must be requested at the relevant points through the section since, by default, Abaqus provides output only on the shell surfaces (where the transverse shear stresses are zero).

Laminated Composite Shells

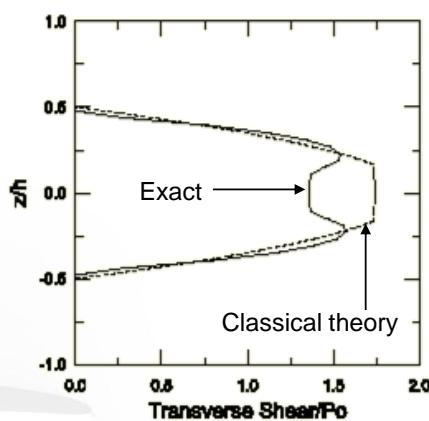
- Example: Pagano plate problem
 - Three-layer composite 0/90/0 subjected to a sinusoidal distributed load with minimal boundary conditions.
 - See Benchmark 1.1.3, “Composite shells in cylindrical bending.”
 - We are interested in the transverse shear stress distribution through the thickness of the shell at the extreme end.



Analysis of Composite Materials with Abaqus

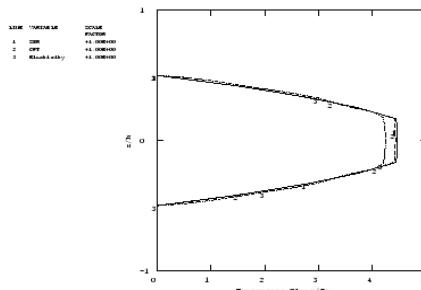
Laminated Composite Shells

- For a very thick plate ($l/h = 4$), the results for conventional shell elements (e.g., S8R) or a single continuum shell element (SC8R) through the thickness come close to classical laminated plate theory (dashed line).
 - They overpredict, however, the peak transverse shear stress compared to the exact elasticity solution.
- Results for multiple continuum shell elements stacked in the thickness direction come close to the exact elasticity solution (solid line).

Laminated shell: transverse shear stress, $l/h = 4$

Laminated Composite Shells

- At higher span-to-thickness ratios ($l/h = 10$, for example) the finite element and classical laminate plate theory results agree closely with the exact elasticity solution.
 - Accurate finite element results are obtained using either conventional or continuum shell elements (in the latter case a single continuum shell element through the thickness is sufficient).



Laminated shell: transverse shear stress, $l/h = 10$

- Thus, we see that for elastic behavior of the cross-section, we can obtain reasonably accurate estimates of both the section's transverse shear stiffness and the transverse shear stresses through the section.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Composite Shells

- Instead of allowing Abaqus to compute the transverse shear stiffness for a shell, the user can define the transverse shear stiffness values.
 - User-supplied values will override the default values.
 - For element types such as S4R5 and S8R5 (where the shear strain energy is used merely as a penalty), the average of the two stiffnesses is used along the edges.
- This option would be used if the section is such that the built-in computation would be inaccurate.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Continuum Shell Elements

© DASSAULT SYSTEMES

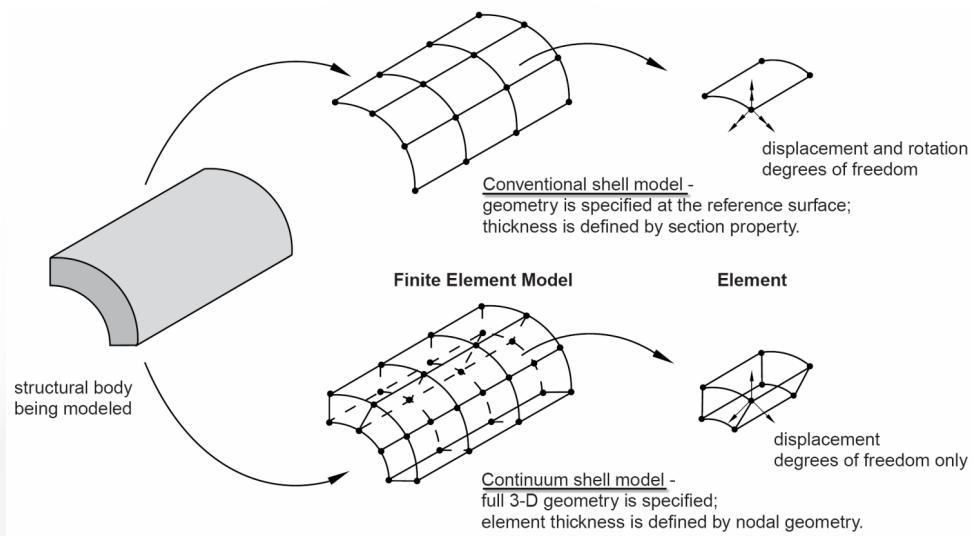


L3.28

Continuum Shell Elements

• Overview

- Continuum shell elements are three-dimensional stress/displacement elements for use in modeling structures that are generally slender, with a shell-like response but continuum element topology.



© DASSAULT SYSTEMES



Continuum Shell Elements

- The elements allow for:
 - Thick and thin shell applications.
 - Linear and nonlinear behavior (both large deformation and elastic-plastic material response).
 - Thickness tapering.
 - The elements derive from 3-D meshed geometry.
 - More accurate contact modeling than conventional shells.
 - They take into account two-sided contact and thickness changes.
 - Stacking.
 - They capture more accurately the through-thickness response for composite laminate structures.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

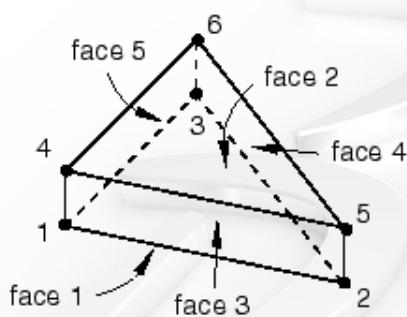
Continuum Shell Elements

• Element topology

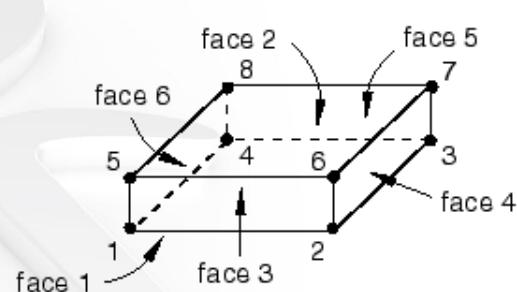
- Two continuum shell element topologies are available in Abaqus for general-purpose applications and finite membrane strains

SC6R 6-node triangular wedge

SC8R 8-node hexahedron



6-node continuum-shell



8-node continuum-shell

© DASSAULT SYSTEMES

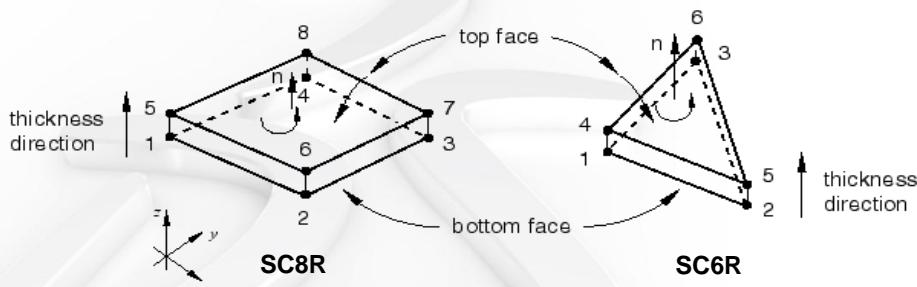


Analysis of Composite Materials with Abaqus

Continuum Shell Elements

- Default thickness (stack) direction

- The kinematic response in the thickness direction is different from that in the in-plane directions for the continuum shell.
- The thickness direction can be ambiguous for the SC8R element.
 - Any of the 6-faces could be the bottom face.
- The default behavior uses the nodal connectivity:



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Continuum Shell Elements

- Surfaces, contact, and coupling

- Surfaces are defined on the element in the same way that they are defined on continuum solid elements.
 - All surface-based loads can be activated (top, bottom, and edges).
- Contact takes place on the **actual** shell surface, not the reference surface.
- Double-sided contact is permitted.
- Coupling is fully supported. Matching meshes are not required for:
 - Continuum shell to continuum shell elements (Tie constraint).
 - Continuum shell to continuum solid elements (Tie constraint).
 - Can be directly connected for matching meshes, however.
 - Conventional shell to continuum shell coupling (Shell-to-solid coupling).

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Continuum Shell Elements

- The user interface looks like the interface for continuum solid elements (where appropriate) or conventional shell elements (where appropriate).

```
*ELEMENT, TYPE=SC6R, ELSET=triangles
*ELEMENT, TYPE=SC8R, ELSET=quads
*SHELL SECTION, ELSET=triangles,
  MATERIAL=steel, POISSON=ν,
  THICKNESS MODULUS=ε
*SHELL SECTION, ELSET=quads, COMPOSITE,
  ORIENTATION=orient,
  STACKING DIRECTION={1|2|3|orientation}
```

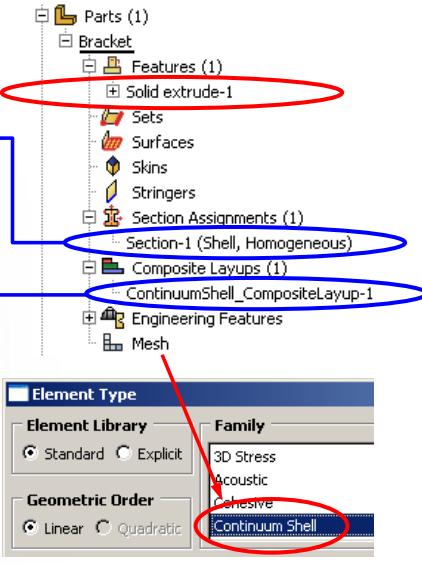
thickness, # sect pts, material, orientation

*MATERIAL, NAME=steel

*ELASTIC

*PLASTIC

.....



Analysis of Composite Materials with Abaqus

Continuum Shell Elements

• Limitations

- Continuum shell elements **cannot be used with the hyperelastic or hyperfoam material models.**
- Although continuum shells provide robust and accurate solutions to most shell applications, these elements may show **slow convergence for very thin shell applications.**
- In Abaqus/Explicit the element stable time increment can be controlled by the continuum shell element thickness.
 - This may increase significantly the number of increments taken to complete the analysis when compared to the same problem modeled with conventional shell elements.
 - The small stable time increment size may be mitigated by specifying a lower stiffness in the thickness direction when appropriate.

Continuum Shell Meshing

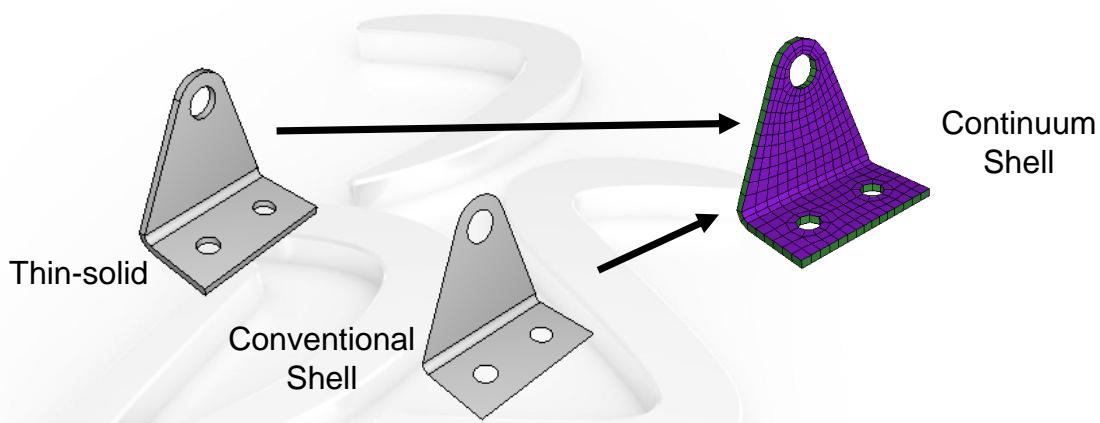
© DASSAULT SYSTEMES



L3.36

Continuum Shell Meshing

- There are a number of tools available in Abaqus/CAE to facilitate the creation of properly-oriented continuum shell meshes using the default thickness (stack) direction.
 - These tools facilitate the conversion of thin solids and conventional shells to continuum shells.

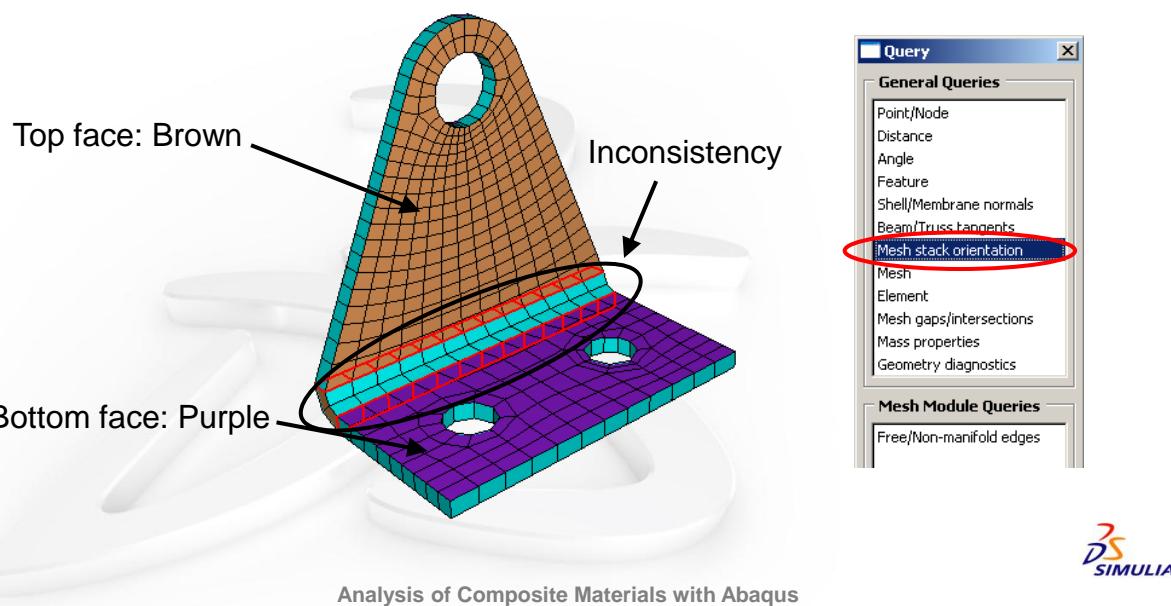


© DASSAULT SYSTEMES

Continuum Shell Meshing

- **Query stack direction orientation for native and orphan meshes**
 - Element faces are color coded.
 - Inconsistencies in element orientations are highlighted.

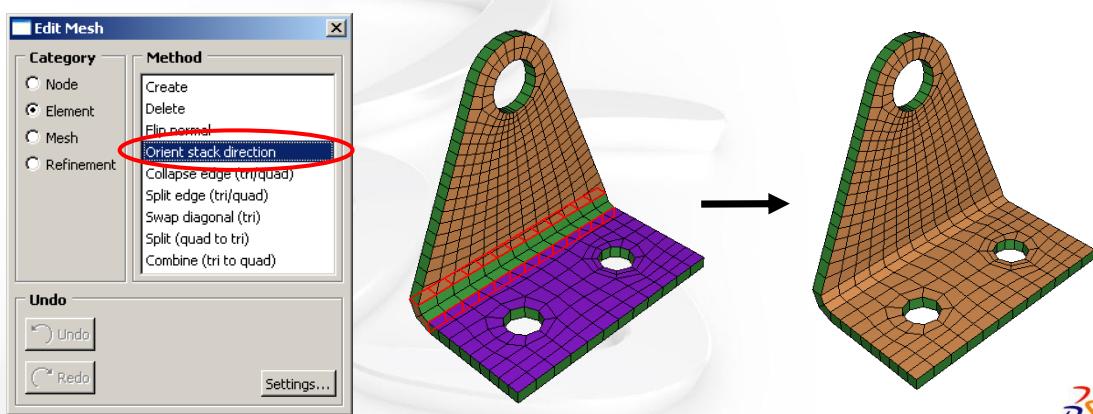
© DASSAULT SYSTEMES



Continuum Shell Meshing

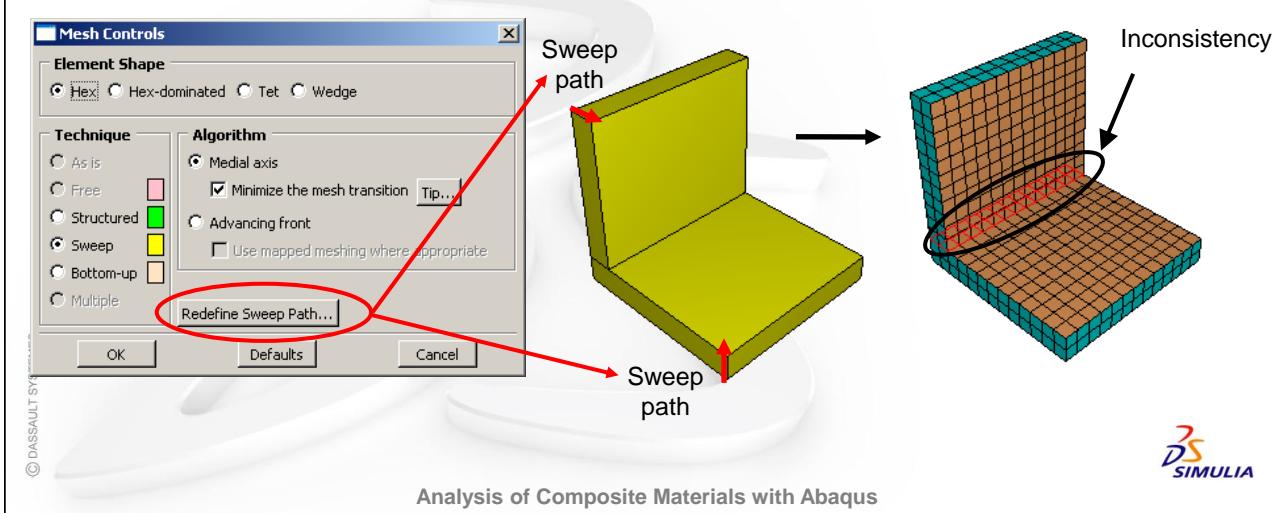
- **Edit orphan mesh stack orientation**
 - Selected elements are oriented with respect to a reference top face.
 - Node labels, element labels, and node coordinates are not altered.
 - Surfaces are managed during the transformation.
 - The tool is only available for orphan meshes.
 - Orphan mesh parts can be created from meshed native geometry.

© DASSAULT SYSTEMES



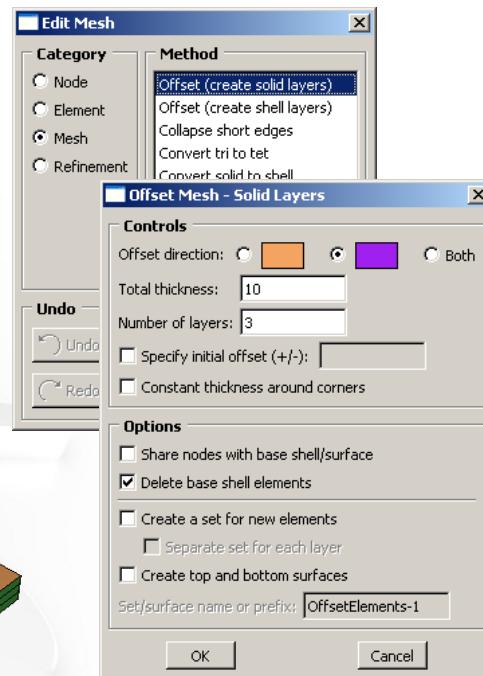
Continuum Shell Meshing

- It may be difficult to build a layered mesh on native geometry.
 - Swept meshing technique does build the mesh in layers.
 - It is possible to control the mesh stack orientation by choosing the desired sweep path.
 - However, orientation inconsistencies often occur between regions.



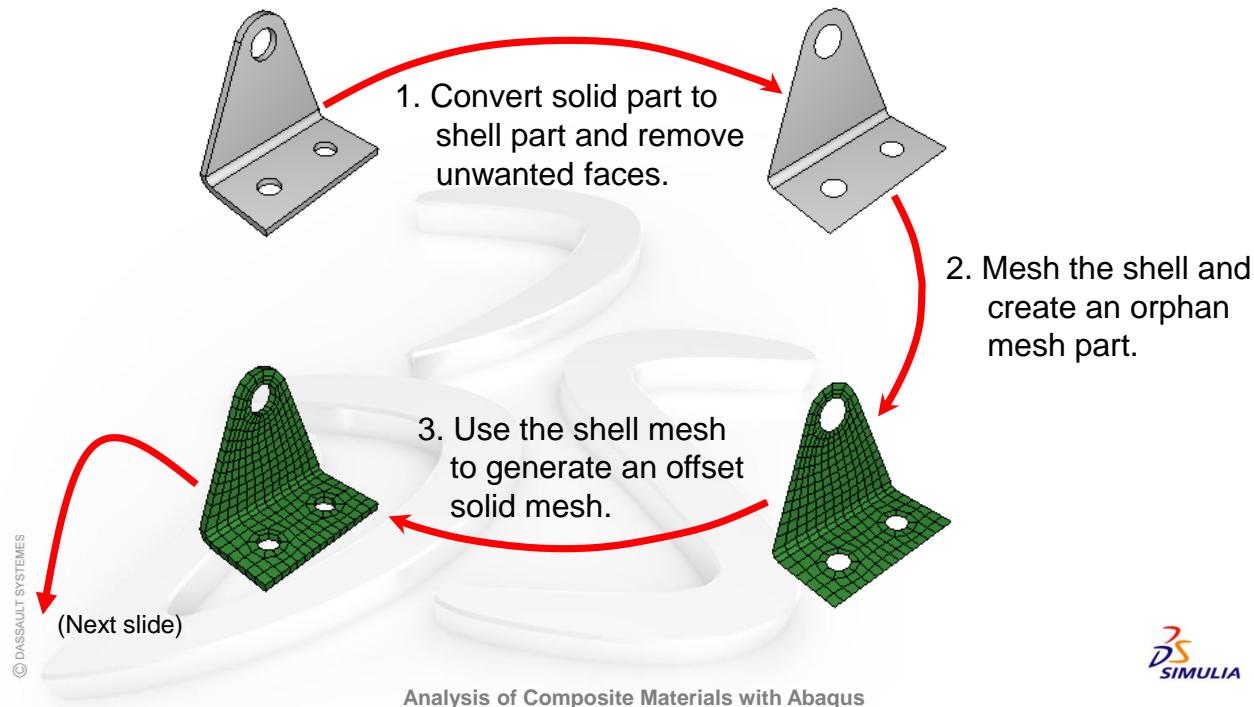
Continuum Shell Meshing

- Offset a shell mesh to generate layers of solid elements.
 - The starting point is a shell orphan mesh.
 - Shell mesh is “thickened” by offsetting nodes normal to the boundary and building elements that propagate out in the normal direction



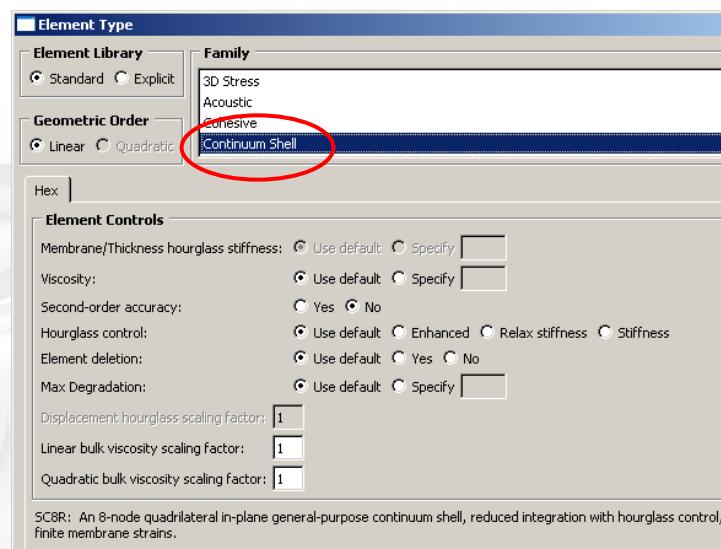
Continuum Shell Meshing

- Generating an oriented mesh via mesh offsetting



Continuum Shell Meshing

- 4. Assign continuum shell element type.



Continuum Solid Elements

© DASSAULT SYSTEMES



L3.44

Continuum Solid Elements

- As mentioned previously, Abaqus/Standard offers a capability to model layered solid elements.
 - The use of composite solids is limited to three-dimensional brick elements that have only displacement degrees of freedom.

© DASSAULT SYSTEMES



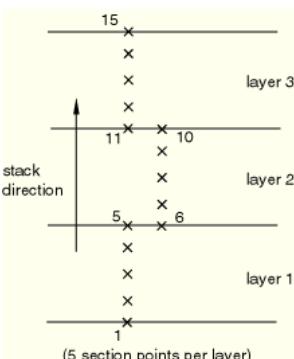
Continuum Solid Elements

- Layered solid elements do not provide a more accurate solution than composite shell elements.
 - They are primarily intended as a modeling convenience.
 - In most cases you should model a composite section with either conventional or continuum shell elements.
 - However, you should use a composite solid section for the following cases:
 - When the transverse shear effects are predominant.
 - When you cannot ignore the normal stress.
 - When you require accurate interlaminar stresses, such as near localized regions of complex loading or geometry.

Continuum Solid Elements

Properties

- Required properties for each layer:
 - Thickness
 - Number of section points
 - Must be an odd number (Simpson's rule is used in the stacking direction)
 - If one section point through the layer is used, it will be located in the middle of the layer thickness
 - Material
 - Orientation



Numbering of section points in a three-layered composite solid element

Symmetry Conditions and Laminated Structures

© DASSAULT SYSTEMES

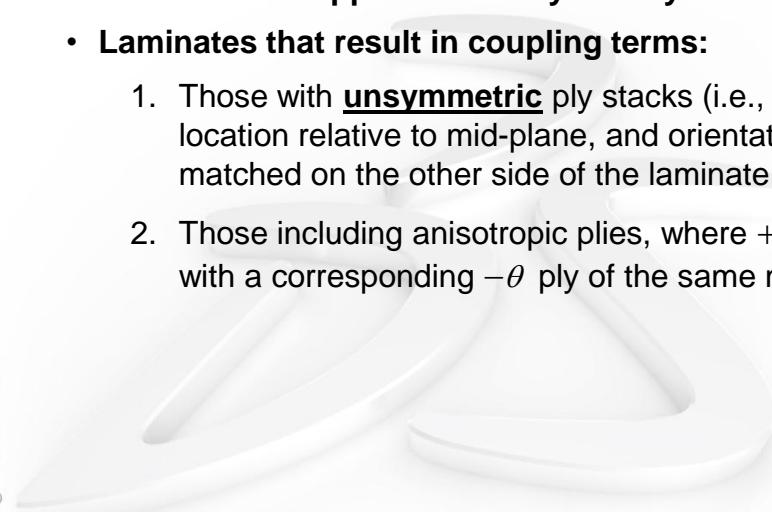


L3.48

Symmetry Conditions and Laminated Structures

- Be careful in the use of symmetry boundary conditions in composite models using a mixed modeling approach (i.e., the laminate stacking sequence is explicitly defined)
- Certain laminates (even those where all layers are composed of *isotropic* materials) lead to coupling terms in the stiffness matrix that invalidate the application of symmetry boundary conditions
- Laminates that result in coupling terms:
 1. Those with unsymmetric ply stacks (i.e., material, thickness, ply location relative to mid-plane, and orientation must be identically matched on the other side of the laminate mid-plane)
 2. Those including anisotropic plies, where $+\theta$ plies are not balanced with a corresponding $-\theta$ ply of the same material and thickness

© DASSAULT SYSTEMES



Symmetry Conditions and Laminated Structures

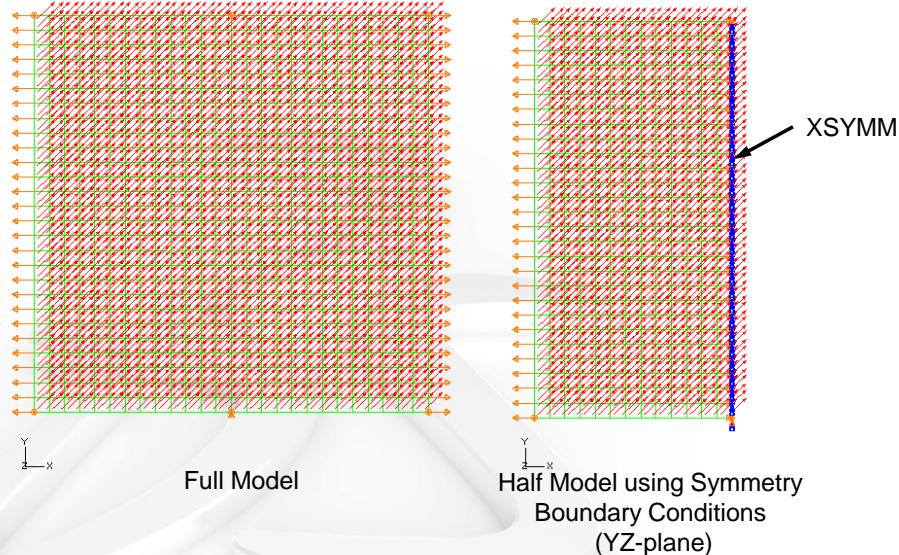
Laminate Conditions		Coupling Type					Examples
Balanced	Symmetric	Bending-Twisting	Twisting-Shearing	Twisting-Stretching	Bending-Shearing	Stretching-Shearing	
No	Yes	X					(60/60)
No	No	X	X	X	X		(60/-30)
Yes	Yes						(60/-60)s
Yes	No			X	X		(60/-60)
-	No		X				(0/90); different materials
-	No		X				(0/90); same materials
-	No		X			X	Isotropic, unsymmetric (e.g., Steel/Aluminum)

Symmetry Conditions and Laminated Structures

- Of all the cases from the prior page, only the (0/90) case with the same material in each ply can legitimately utilize symmetry boundary conditions
- Avoid symmetry boundary conditions for all other cases

Symmetry Conditions and Laminated Structures

- Example: One 45-degree ply laminate in tension
 - Fiber direction and boundary conditions shown below

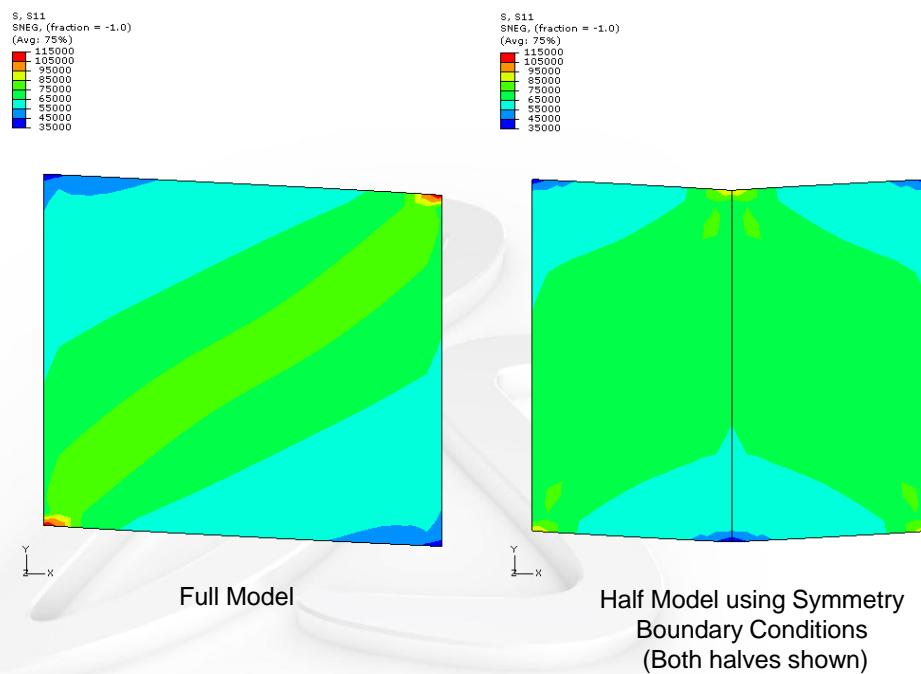


© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Symmetry Conditions and Laminated Structures

- Example (cont'd)



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Notes

Notes

Composite Modeling with Abaqus

Lecture 4

© DASSAULT SYSTEMES



L4.2

Overview

- Introduction
- Understanding Composite Layups
- Understanding Composite Layup Orientations
- Defining Composite Layup Output
- Viewing a Composite Layup
- Abaqus/CAE Demonstration: Three-ply composite
- Composites Modeler for Abaqus/CAE

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES

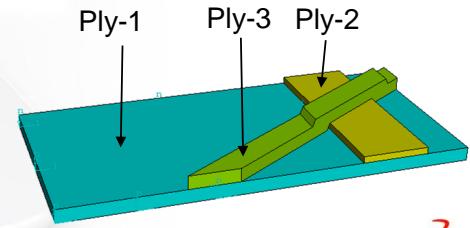


L4.4

Introduction

- A composite layup contains a number of plies.
 - Plies represent the materials as placed in a mold.
 - A ply is composed of an orthotropic material, typically with fibers oriented along a reference orientation, or
 - can be also an isotropic material, e.g., a foam core.
 - Generally each ply has a uniform thickness.
 - Plies are usually the data that the CAD designers/manufacturers know.
 - Plies are inherently easy to conceptualize.

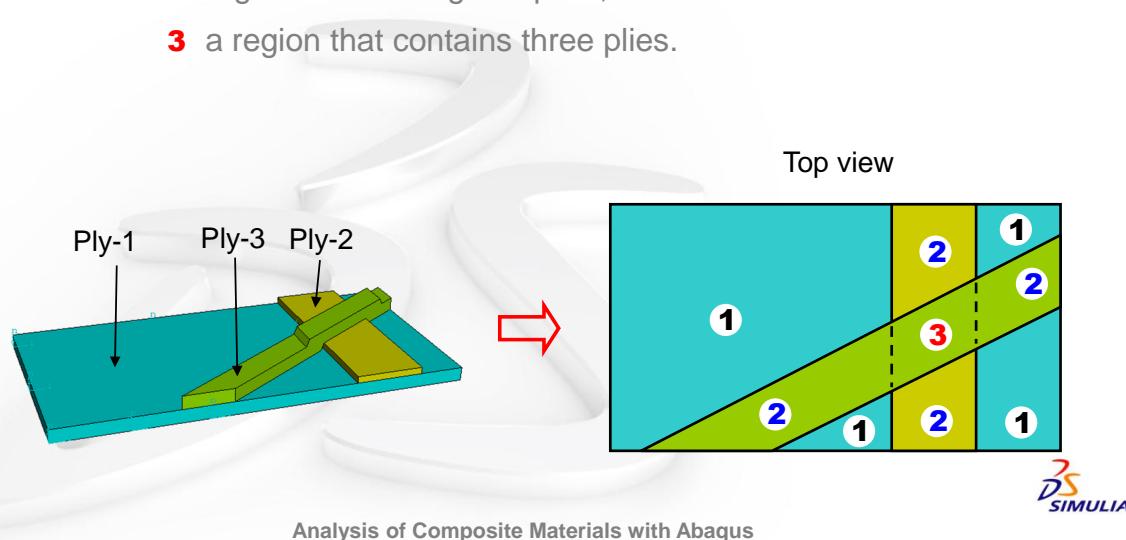
© DASSAULT SYSTEMES



Introduction

- A different number of plies can be contained in different regions of a composite layup.
 - For example, the following composite layup includes
 - regions containing a single ply,
 - regions containing two plies, and
 - a region that contains three plies.

© DASSAULT SYSTEMES



Introduction

- The composite layup interface in Abaqus/CAE is designed to help you manage a large number of plies in a typical composite model.
 - The procedure for creating a composite layup with Abaqus/CAE mirrors the procedure for creating a real composite part:
 - start with a basic shape (partitioned into appropriate regions), then
 - add plies of different materials and thickness to selected regions, and
 - orient the plies in particular directions.
 - The composite modeling and postprocessing are ply-based.
 - Layered conventional shell, continuum shell, and solid elements are supported.
 - conventional shell composite layup ↔ conventional shell elements
 - continuum shell composite layup ↔ continuum shell elements
 - solid composite layup ↔ solid elements

© DASSAULT SYSTEMES

Introduction

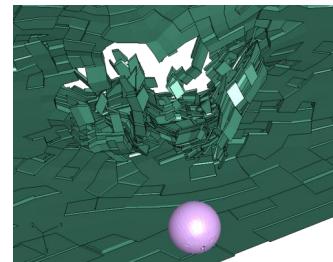
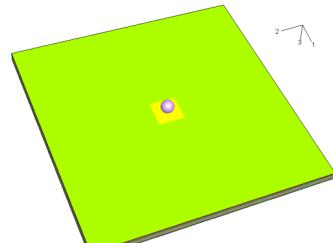
- The GUI is designed for easy manipulation of large numbers (hundreds) of plies and ply data, and large-scale composite structures.
- Ply Management is available.
 - Easily add new plies.
 - Delete, suppress, reposition, or pattern existing plies.
- The ply data can be read from/written to a text file.
- Discrete fields are supported for composite layup reference orientations, shell element offsets, and shell thicknesses.
- Output requests are available for composite layups.
- User-specified ply names are available in the ODB and Abaqus/Viewer for easy tracking in postprocessing operations.
- A ply region can be either Abaqus/CAE geometry, a native mesh, or an orphan mesh.
- Composite layup definitions are suppressible.

Introduction

- In addition to the built-in layup feature, Abaqus also offers fiber simulation capabilities and advanced modeling tools via “Composites Modeler for Abaqus/CAE”.
 - Composites Modeler for Abaqus/CAE is an add-on product developed by Simulayt Ltd.
 - A brief description of Composites Modeler for Abaqus/CAE will be given later in this lecture; more information can be found in the *Composites Modeler for Abaqus/CAE* lecture notes.

Introduction

- This lecture discusses the layup approach to defining composites with Abaqus.
- There exist, however, limitations in this capability when modeling stacked continuum shell/solid elements that require the use of an alternative modeling approach.
 - By *stacked* we mean multiple elements through the thickness
 - If a composite layup is assigned to such a region, each element in the stack will contain the plies defined in the ply table and the analysis results will not be as expected.
 - The alternative modeling methods discussed in Lecture 5 can be used instead to model stacked continuum shell/solid elements.
 - Example: Delamination of composite plates under ballistic impact
 - “Modeling Composite Material Impact with Abaqus/Explicit,” Lecture 10 applies the alternative modeling technique to this problem.



 SIMULIA

Analysis of Composite Materials with Abaqus

Understanding Composite Layups

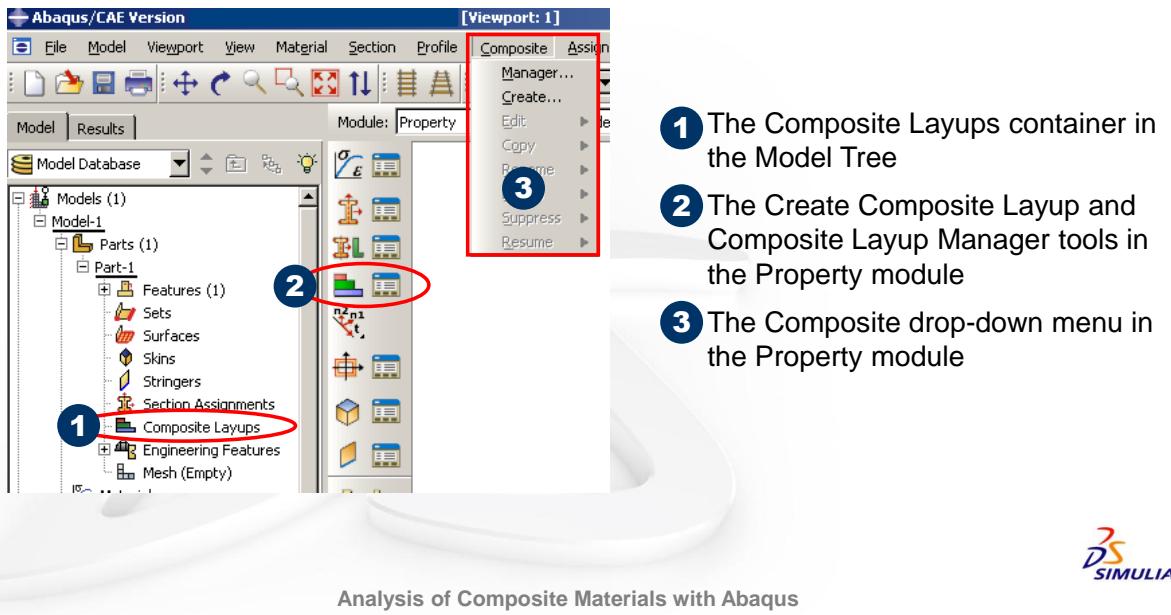


 SIMULIA

Understanding Composite Layups

- The composite layup GUI interface:

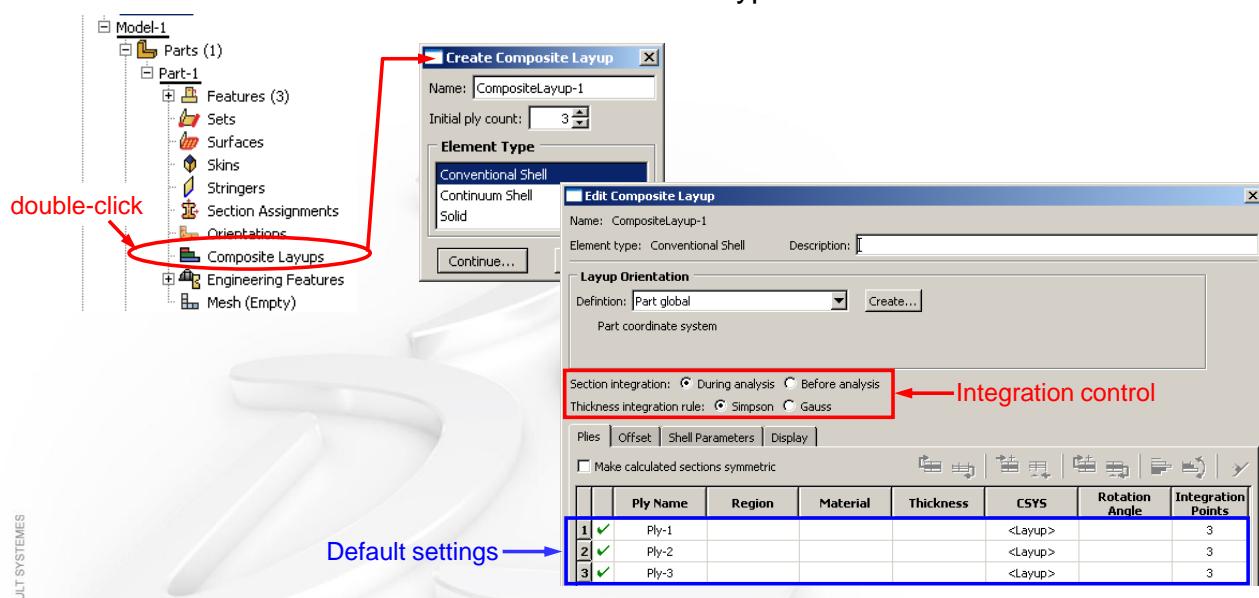
- You can access the composite layup editor in one of the following ways:



Understanding Composite Layups

- Creating a conventional shell composite layup

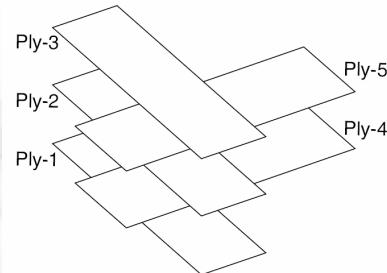
- Select the conventional shell element type



Understanding Composite Layups

- A ply table in the composite layup editor is used to define the name, region, thickness, material, relative orientation, and number of integration points for each ply.
 - Enter the plies that overlap in the composite layup in the order that they appear in the overlapping region.
 - The first ply in the ply table represents the bottom ply in the layup.

© DASSAULT SYSTEMES



Plies	Offset	Shell Param
		Ply Name
1	✓	Ply-1
2	✓	Ply-4
3	✓	Ply-2
4	✓	Ply-5
5	✓	Ply-3

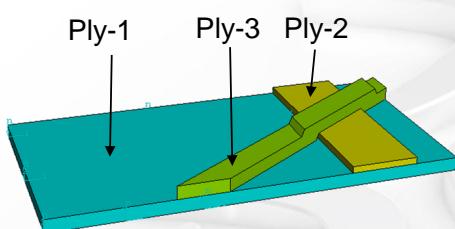
Analysis of Composite Materials with Abaqus



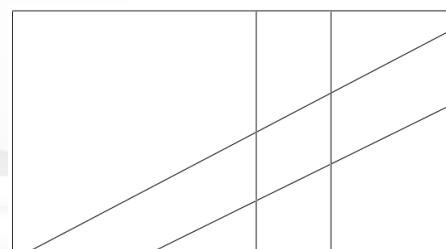
Understanding Composite Layups

- Example: Three-ply composite
 - Conventional shell elements are used.
 - Before creating the layup and defining plies, partition the model to create the regions to which plies will be assigned, if necessary.

© DASSAULT SYSTEMES



geometry of the three-ply composite



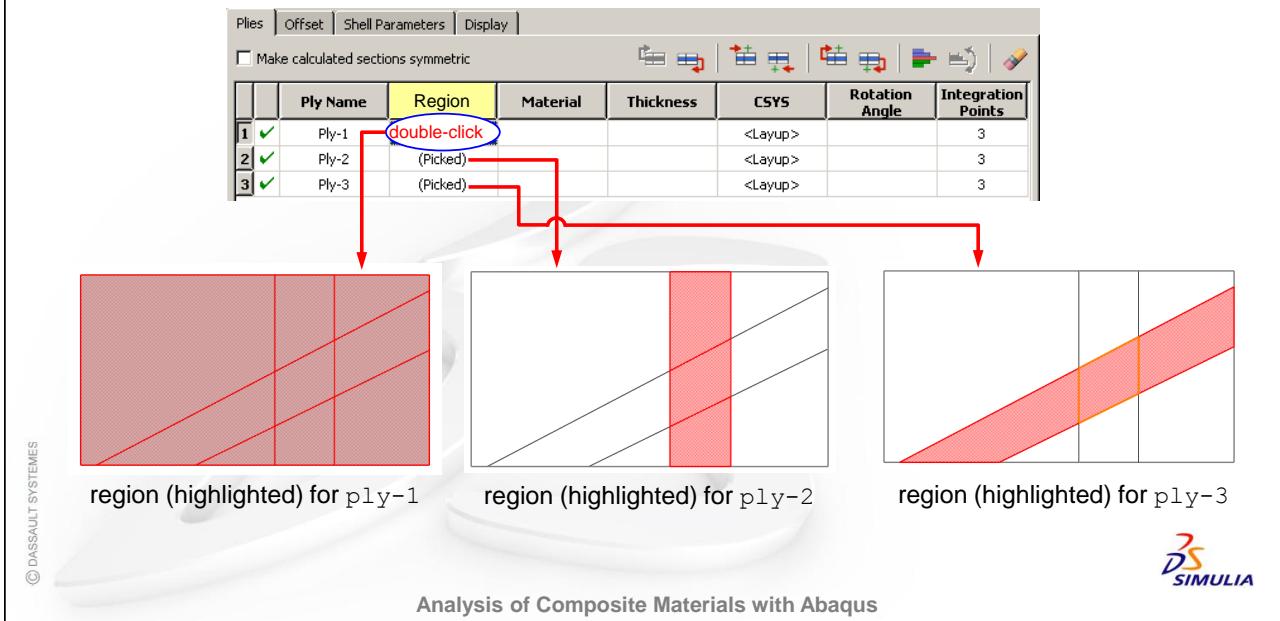
partitioned Abaqus/CAE geometry

Analysis of Composite Materials with Abaqus



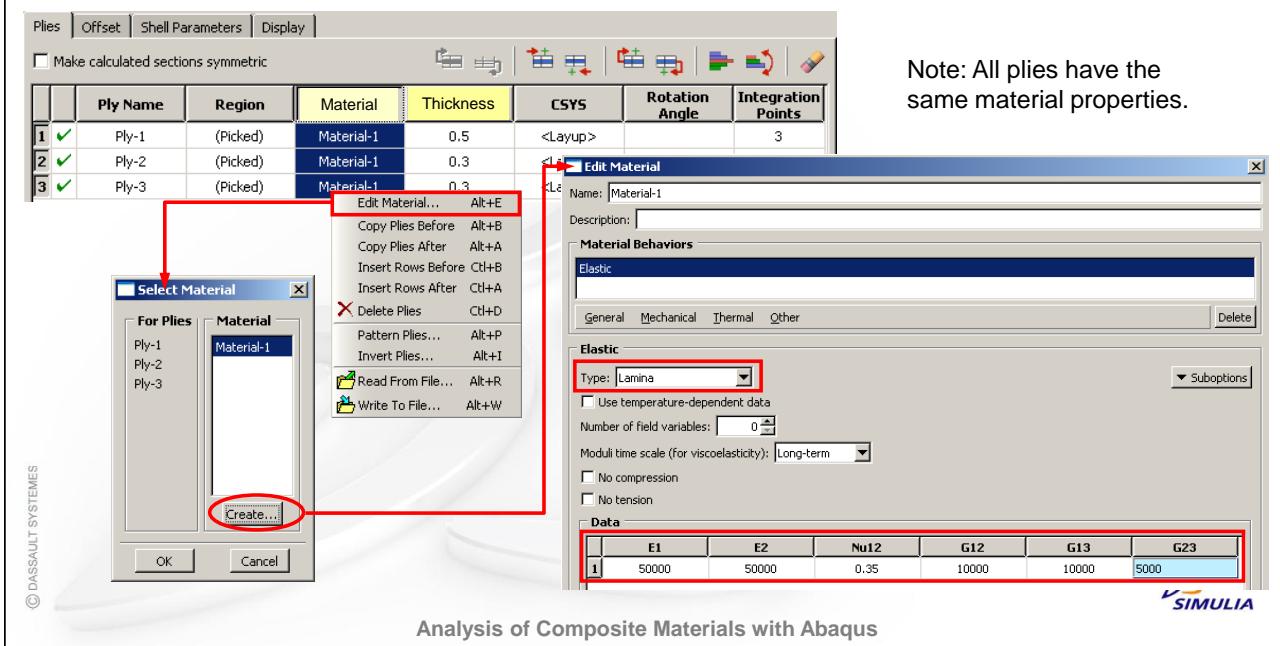
Understanding Composite Layups

- Select the region for each ply
 - The region can be picked directly from the part in the current viewport or a named set that refers to the region.



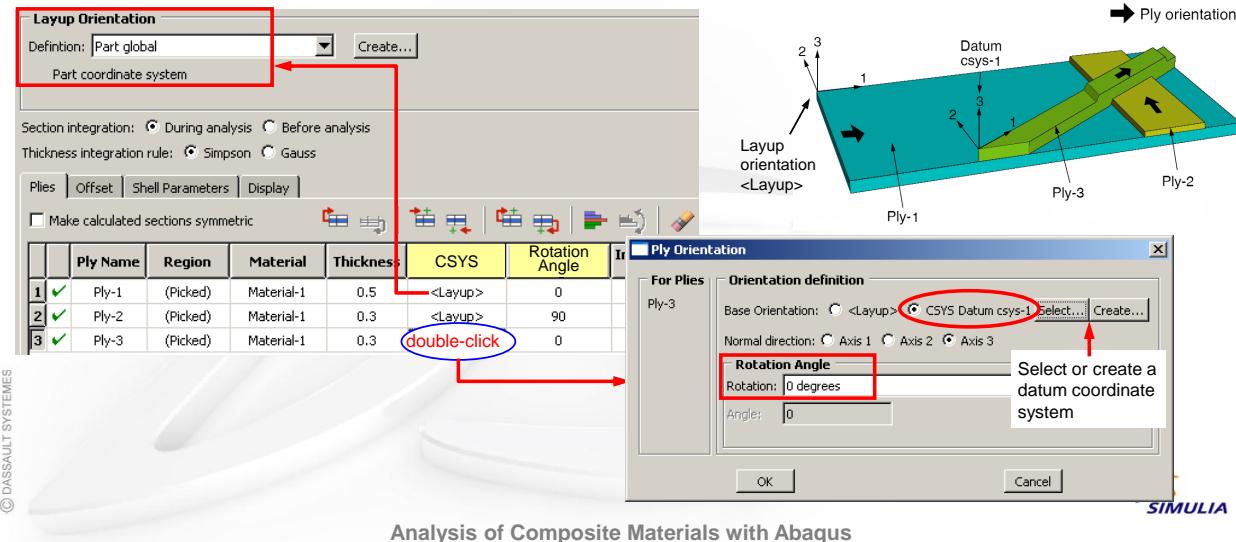
Understanding Composite Layups

- Define the material properties and thicknesses for each ply.
 - Set the material properties for all plies at the same time by clicking mouse button 3 on the header row.



Understanding Composite Layups

- Orient each ply
 - <Layout> is the default orientation.
 - The rotation angle defines the orientation of the fibers within each ply relative to the ply's coordinate system (CSYS column).
 - Details of ply orientations will be discussed in next section.



Understanding Composite Layups

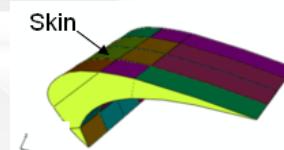
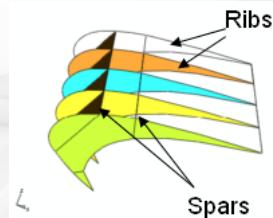
- The composite layup GUI interface is very convenient for analyzing large-scale composite structures.
- Example: Composite wing slat**
 - A wing slat is an aerodynamic device used at the leading edge of an aircraft wing in order to provide smooth air-flow at a higher angle of attack.
 - Each wing typically has a number of slats that are operated during take-off and landing to provide greater lift at slower speeds.
 - In order to reduce the overall structural weight of the aircraft, many structural components are made using high-performance fiber-reinforced composite materials.
 - Each composite layup has a large number of plies, which is typical of aircraft design.



An aircraft wing showing the slats

Understanding Composite Layups

- The wing slat consists of a number of ribs and spars and the skin.
- The geometry is meshed using conventional shell elements, S4R.
- Since a shell offset will be defined for the skin, two composite layups will be defined:
 - Ribs_and_Spars_layup for the ribs and spars
 - skin_layup for the skin



Geometry of a wing slat

© DASSAULT SYSTEMES

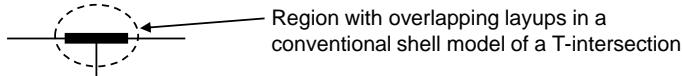


Analysis of Composite Materials with Abaqus

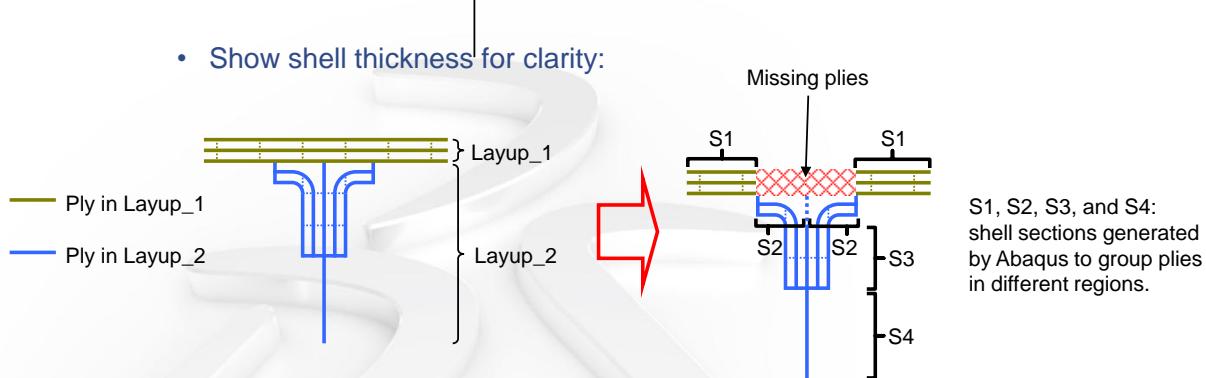
Understanding Composite Layups

- Note:** If you apply two or more composite layups to regions that overlap, Abaqus/CAE uses the properties of the last layup (based on the names of the composite layups in alphabetical order).

- For example:



- Show shell thickness for clarity:



- Resolution:** A single layup should be defined for the entire part to avoid missing plies when meshing connections between structural components.

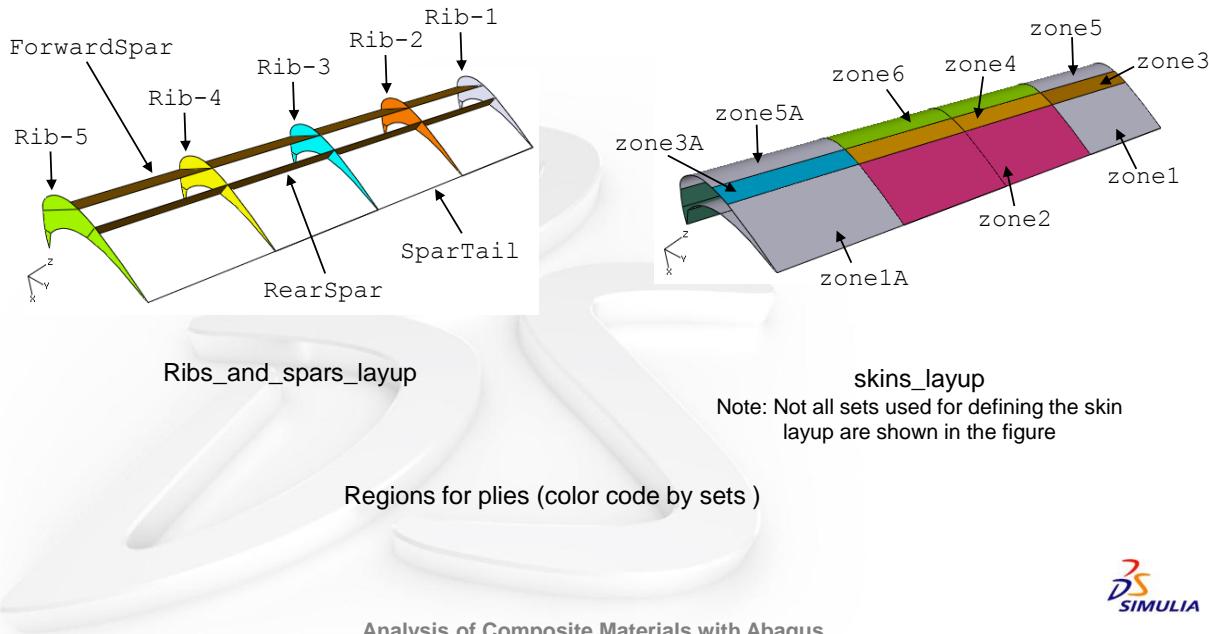
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Understanding Composite Layups

- Define named sets for the regions to which plies will be assigned.



Analysis of Composite Materials with Abaqus

Understanding Composite Layups

- Define the ribs and spars layup which contains 82 plies.
 - Read the ply data from a text file.

Edit Composite Layup

Name: ribs_and_spars_layup
Element type: Conventional Shell Description:

Layup Orientation
Definition: Part global Create...
Part coordinate system

Section integration: During analysis Before analysis
Thickness integration rule: Simpson Gauss

Plies | Offset | Shell Parameters | Display |
Make calculated sections symmetric

Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
1	<Layup>		3			
2	<Layup>		3			
3	<Layup>		3			
<input type="button" value="Move Plies Down Alt+D"/> <input type="button" value="Copy Plies Before Alt+B"/> <input type="button" value="Copy Plies After Alt+A"/> <input type="button" value="Insert Rows Before Ctrl+B"/> <input type="button" value="Insert Rows After Ctrl+A"/> <input type="button" value="Delete Plies Ctrl+D"/> <input type="button" value="Pattern Plies... Alt+P"/> <input type="button" value="Read From File... Alt+R"/> <input type="button" value="Write To File... Alt+W"/>						

Read Data from ASCII File

File: D:/users/slats/ribs_and_spars_layup Select...
Field delimiter: spaces, tabs, or commas
Start reading values into table row: 1
Start reading values into table column: 2
OK Cancel

ForwardSpar

Plies

Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
Ply_76	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	45	3
Ply_77	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	45	3
Ply_78	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	45	3
Ply_79	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	-45	3
Ply_80	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	-45	3
Ply_81	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	-45	3
Ply_82	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	-45	3
Ply_83	Rib-1	Glass-Epoxy	0.2	Rib1-CSYS.3	45	3
Ply_84	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS.3	90	3
Ply_85	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS.3	90	3
Ply_86	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS.3	45	3
Ply_87	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS.3	45	3
Ply_88	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS.3	45	3
Ply_89	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS.3	90	3
Ply_90	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS.3	45	3
Ply_91	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS.3	45	3
Ply_92	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS.3	-45	3
Ply_93	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS.3	45	3

SIMULIA

Analysis of Composite Materials with Abaqus

Understanding Composite Layups

- Define the skin layup which contains 59 plies

© DASSAULT SYSTEMES

Edit Composite Layup

Name: skin_layup
Element type: Conventional Shell
Description: []

Layup Orientation
Definition: Part global [] Create...
Part coordinate system

Section integration: During analysis Before analysis
Thickness integration rule: Simpson Gauss

Plies | Offset | Shell Parameters | Display |

Plies

	Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
1	Ply_1	Zone6	Graphite-Epoxy	0.2	<Layup>	90	3
2	Ply_2	Zone6	Graphite-Epoxy	0.2	<Layup>	90	3
3	Ply_3	Zone6	Graphite-Epoxy	0.2	<Layup>	90	3
4	Ply_4	Zone6	Graphite-Epoxy	0.2	<Layup>	90	3
5	Ply_5	Zone6	Graphite-Epoxy	0.2	<Layup>	90	3
6	Ply_6	Zone5A	Graphite-Epoxy	0.21	<Layup>	-45	3
7	Ply_7	Zone5A	Graphite-Epoxy	0.21	<Layup>	-45	3
8	Ply_8	Zone5A	Graphite-Epoxy	0.21	<Layup>	-45	3
9	Ply_9	Zone5A	Graphite-Epoxy	0.2	<Layup>	-45	3
10	Ply_10	Zone5A	Graphite-Epoxy	0.2	<Layup>	-45	3
11	Ply_11	Zone8	Graphite-Epoxy	0.2	<Layup>	-45	3
12	Ply_12	Zone8	Graphite-Epoxy	0.21	<Layup>	-45	3
13	Ply_13	Zone8	Graphite-Epoxy	0.2	<Layup>	-45	3
14	Ply_14	Zone8	Graphite-Epoxy	0.2	<Layup>	-45	3
15	Ply_15	Zone8	Graphite-Epoxy	0.2	<Layup>	-45	3
16	Ply_16	Zone8	Graphite-Epoxy	0.2	<Layup>	-45	3
17	Ply_17	Zone3	Graphite-Epoxy	0.2	<Layup>	45	3
18	Ply_18	Zone3	Graphite-Epoxy	0.2	<Layup>	45	3

Read Data from ASCII File

File: D:\users\slats\skin_layup.csv Select...
Field delimiter: spaces, tabs, or commas
Start reading values into table row: 1
Start reading values into table column: 2
OK Cancel

Analysis of Composite Materials with Abaqus

SIMULIA

Understanding Composite Layups

- Offset the reference surface to the top surface (outer surface) of the skin
 - By default, the middle surface is the reference surface.
 - In the composite layup editor, choose **Top surface** to offset the reference surface to the top surface of the skins.
 - You can also specify an offset ratio, or
 - choose an existing element-based discrete field.
 - Note: Abaqus/CAE allows you to select only valid distributions, which, for an offset, are scalar distributions applied to elements.

Edit Composite Layup

Name: skin_layup [] Element type: Conventional Shell Description: []

Layup Orientation
Definition: Part global [] Create...
Part coordinate system

Section integration: During analysis Before analysis
Thickness integration rule: Simpson Gauss

Plies | Offset | Shell Parameters | Display |

Shell Reference Surface and Offsets

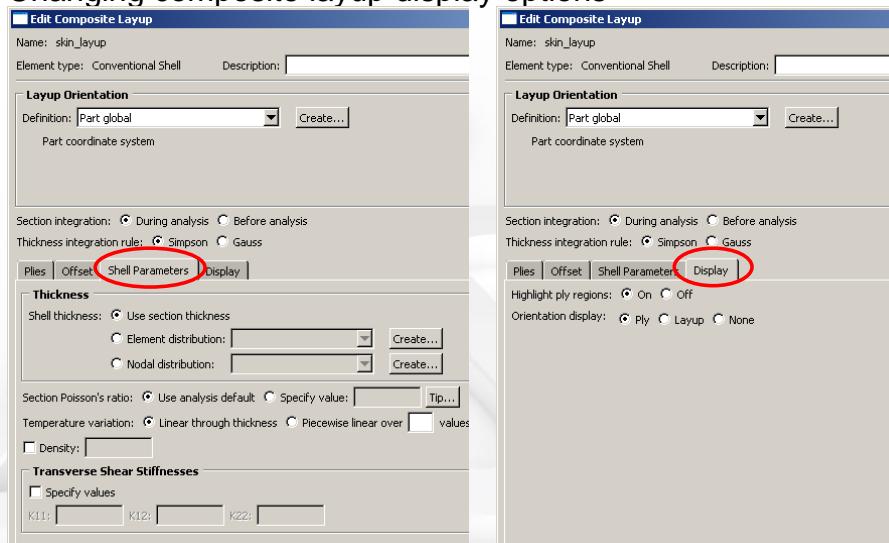
Middle surface
 Top surface
 Bottom surface
 Specify offset ratio: []
 Distribution: [] Create...

D SIMULIA

Understanding Composite Layups

- The composite layup editor also supports
 - Defining shell parameters, including
 - Section Poisson's ratio and transverse shear stiffness.
 - Changing composite layup display options

© DASSAULT SYSTEMES

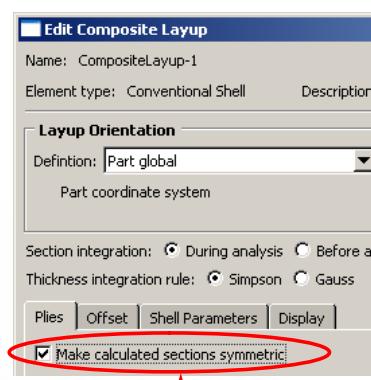


Analysis of Composite Materials with Abaqus

Understanding Composite Layups

- Symmetry option
 - This option simplifies the process of defining composite layups whose plies are symmetric about a central layer.
 - You only need to specify half of the plies in the composite layup, starting with the bottom ply in the first row and ending with the central ply.
 - Abaqus automatically appends plies to the layup definition by repeating all of the specified plies (including the central ply) in the reverse order.

© DASSAULT SYSTEMES



Toggle on this option to activate simplified modeling of symmetric composites



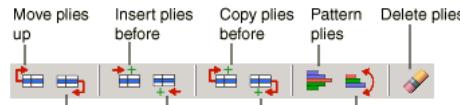
Analysis of Composite Materials with Abaqus

Understanding Composite Layups

• Ply Management

- is very convenient to ...
 - Move plies
 - Copy plies
 - Delete plies
 - Invert plies
 - Pattern a group of selected plies
 - Create symmetric layup
 - Copy plies multiple times
 - Read from a file
 - Write to a file

- is activated by clicking mouse button 3 on the ply table.



These actions can be performed using the icons in the composite layup editor above the ply table.

Move Plies Down	Alt+D
Copy Plies Before	Alt+B
Copy Plies After	Alt+A
Insert Rows Before	Ctl+B
Insert Rows After	Ctl+A
Delete Plies	Ctl+D
Pattern Plies...	Alt+P
Read From File...	Alt+R
Write To File...	Alt+W



Analysis of Composite Materials with Abaqus

Understanding Composite Layups

• Symmetry option vs. symmetry pattern

- Symmetry option
 - Plies generated using the symmetry option cannot be viewed in the layup editor.
 - However, they can be viewed in ply stack plots (will be discussed later in section “Viewing a Composite Layup”) and Abaqus/Viewer, and are labeled using `Sym_` as the prefix to the repeated ply’s original name.
 - Any manipulation of the original plies will automatically propagate to the symmetric plies.
- Symmetry pattern
 - Plies generated using the symmetry pattern can be viewed in the layup editor.
 - Manipulations of the original plies will NOT propagate to the symmetric plies.



Analysis of Composite Materials with Abaqus

Understanding Composite Layups

- The combination of the symmetry option and symmetry pattern can significantly simplify the process of defining composite layups with certain symmetric stacking sequences.
- For example, defining an 8-ply layup with the following stacking sequence: $[(-45, 45)_S]_S$:

1 Apply the symmetry pattern: Symmetric about last ply in the layup

2 Toggle on the Make calculated sections symmetric option

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

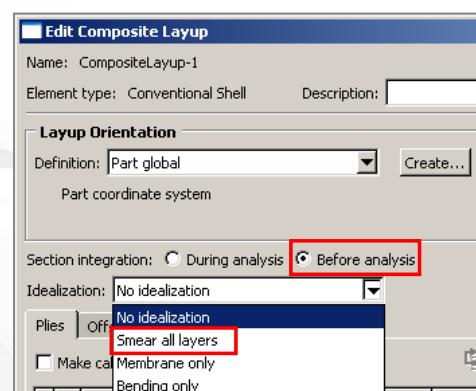
D SIMULIA

Understanding Composite Layups

- Idealizations for a shell composite layup integrated before the analysis**
 - Idealizations allow you to modify the stiffness coefficients in a shell section based on assumptions about the shell's makeup or expected behavior.
 - The following idealizations are available:

1 Smeared layers

- Ignores the effects of the stacking sequence for the plies in the composite layup.
- Contributions from each specified ply are smeared across the entire thickness of the layup, resulting in a general response independent of the stacking sequence.



Understanding Composite Layups

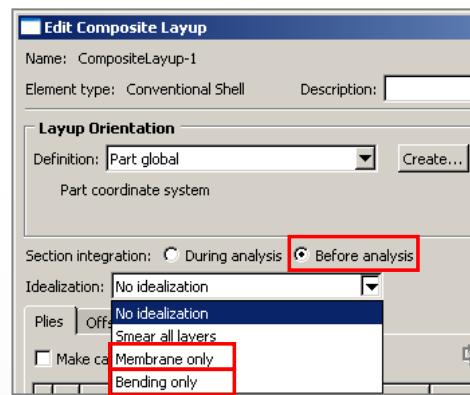
2 Membrane only

- Retains only the membrane stiffness for shells whose predominant response will be in-plane stretching.

3 Bending only

- Retains only the bending stiffness for shells whose predominant response will be pure bending.

- Idealizations modify the shell general stiffness coefficients after they have been computed normally, including the effects of any offsets.
- **Note:** Select **No idealization** (default) to account for the complete stiffness of the shell as determined by the material assignments and ply composition.



© DASSAULT SYSTEMES

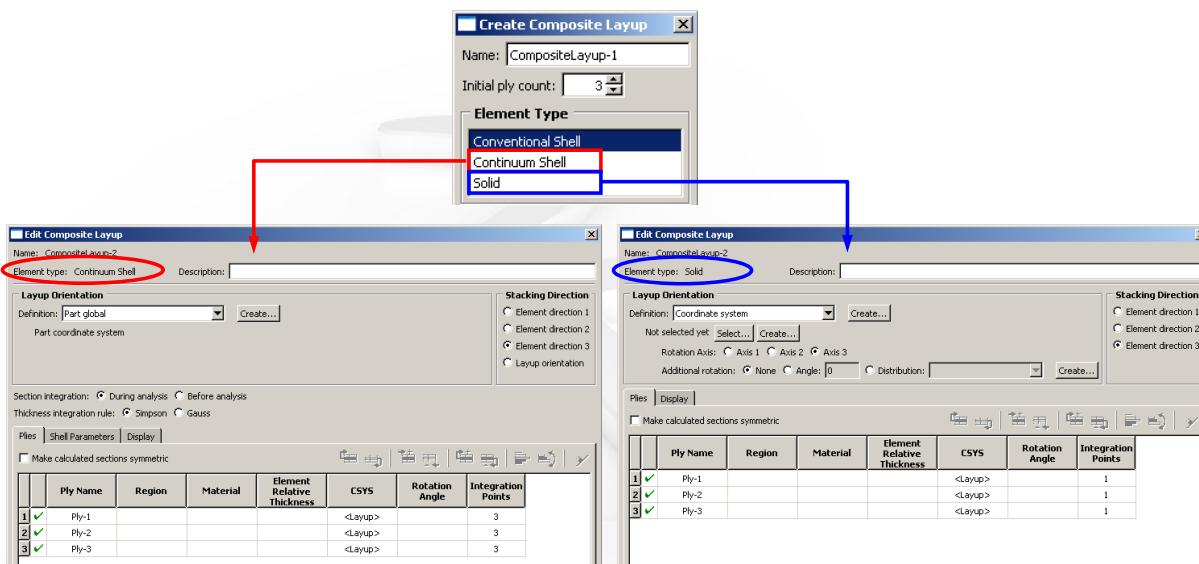


Analysis of Composite Materials with Abaqus

Understanding Composite Layups

• Defining a continuum shell/solid composite layup

- Use a similar procedure to that described for a conventional shell composite layup; therefore, the details will not be discussed.



© DASSAULT SYSTEMES

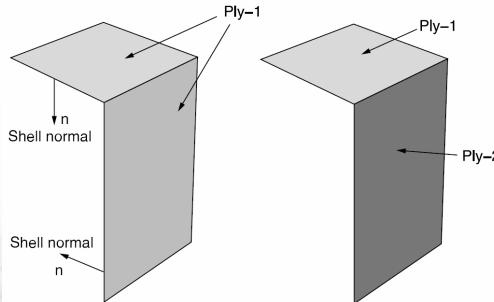


Analysis of Composite Materials with Abaqus

Understanding Composite Layups

- **Limitations**

- Abaqus cannot analyze a composite layup if the shell normal of a single ply makes a sharp transition through an angle of 90° or greater.
 - Separate plies should be used to model such a region.



- The composite layup editor does not support the rebar option.
 - However, alternative methods are available to define rebar layers (discussed in Lecture 6).

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Understanding Composite Layup Orientations

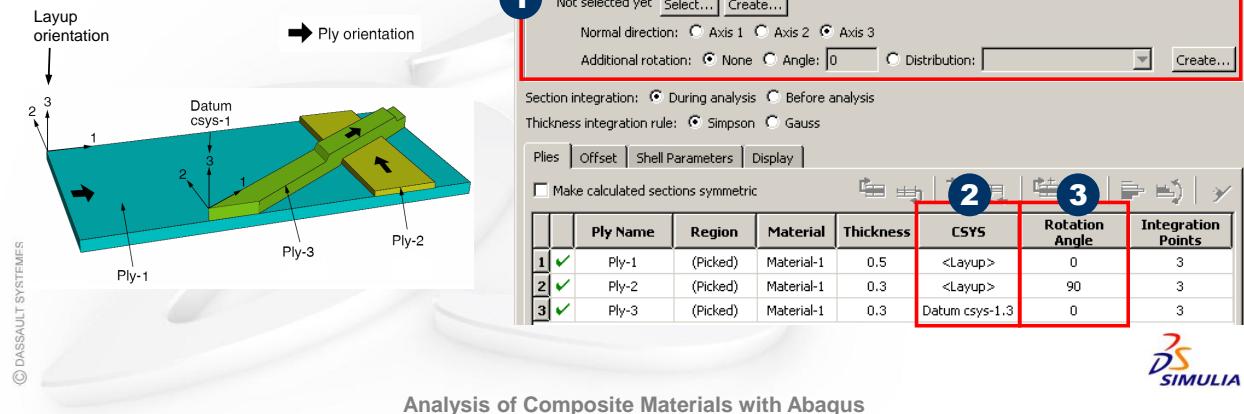
© DASSAULT SYSTEMES



Understanding Composite Layup Orientations

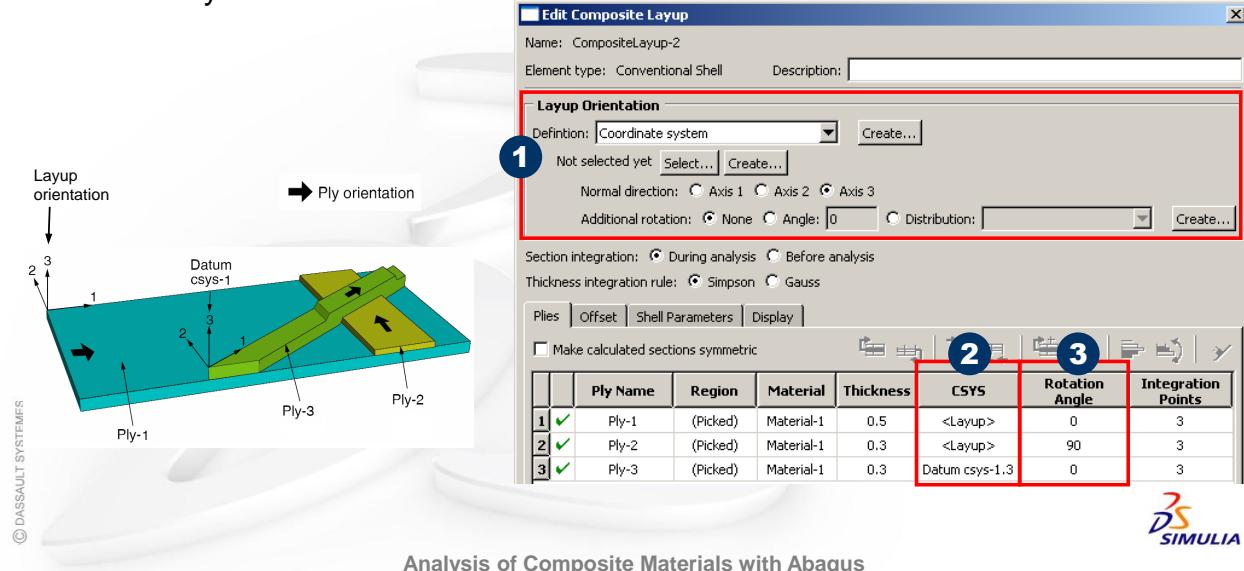
- The orientation of the fibers within each ply of a composite layup plays an important role in determining the physical description of the model.
- The composite layup editor derives the orientation of the fibers from three parameters that are relative to each other:

- 1 Layup orientation
- 2 Ply orientation
- 3 Additional rotation



Understanding Composite Layup Orientations

- By default, ply angles are given with respect to a reference coordinate system listed for each ply.
- If no system is specified, then the default is to use the layup orientation system which, by default, is the same as the part's Global Cartesian system.

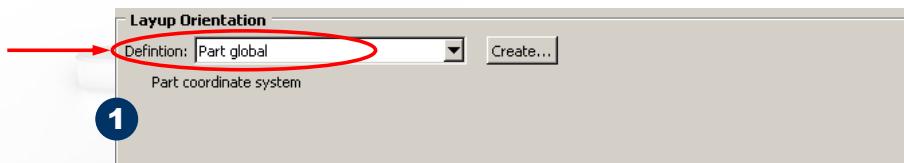


Understanding Composite Layup Orientations

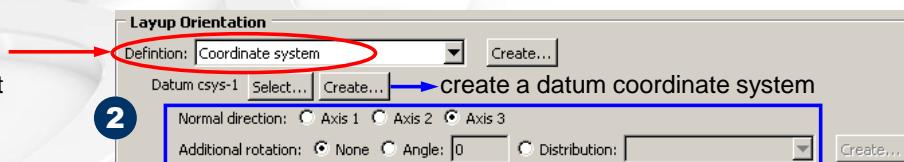
- **Layup orientation**

- Defines the base or reference orientation <Layup> for the layup.
- Determined by one of the following options available in the composite layup editor:

Use the default layup orientation (same as that of the part)



Select a datum coordinate system that defines the orientation



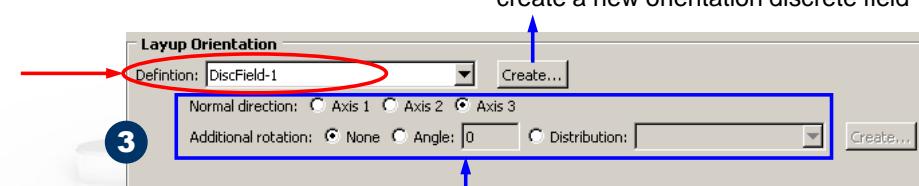
Define normal direction of the layup and additional rotation about the layup normal direction



Analysis of Composite Materials with Abaqus

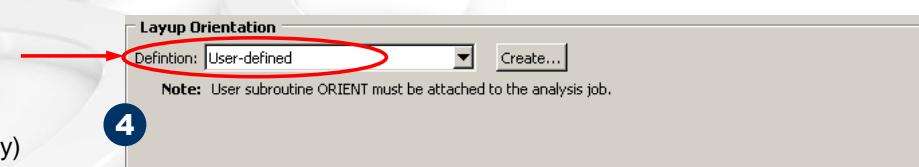
Understanding Composite Layup Orientations

Select an orientation discrete field that defines a spatially varying orientation



define normal direction of the layup and additional rotation about the layup normal direction

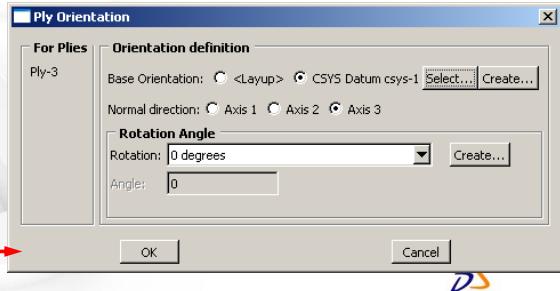
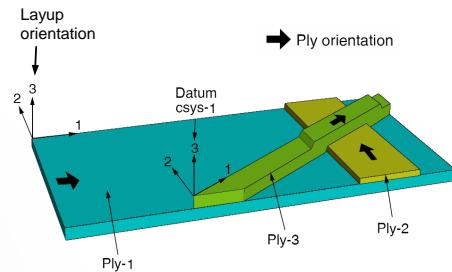
Select an orientation defined in user subroutine ORIENT (Abaqus/Standard only)



Understanding Composite Layup Orientations

Ply orientation

- defines the relative orientation of each ply combined with a CSYS and a rotation angle.
- is determined by
 - selecting 0° , $\pm 45^\circ$ or 90° from the base orientation <layup>,
 - specifying a floating point value between -90° and $+90^\circ$ from the base orientation <layup>, or
 - selecting a coordinate system and additional rotation angle.

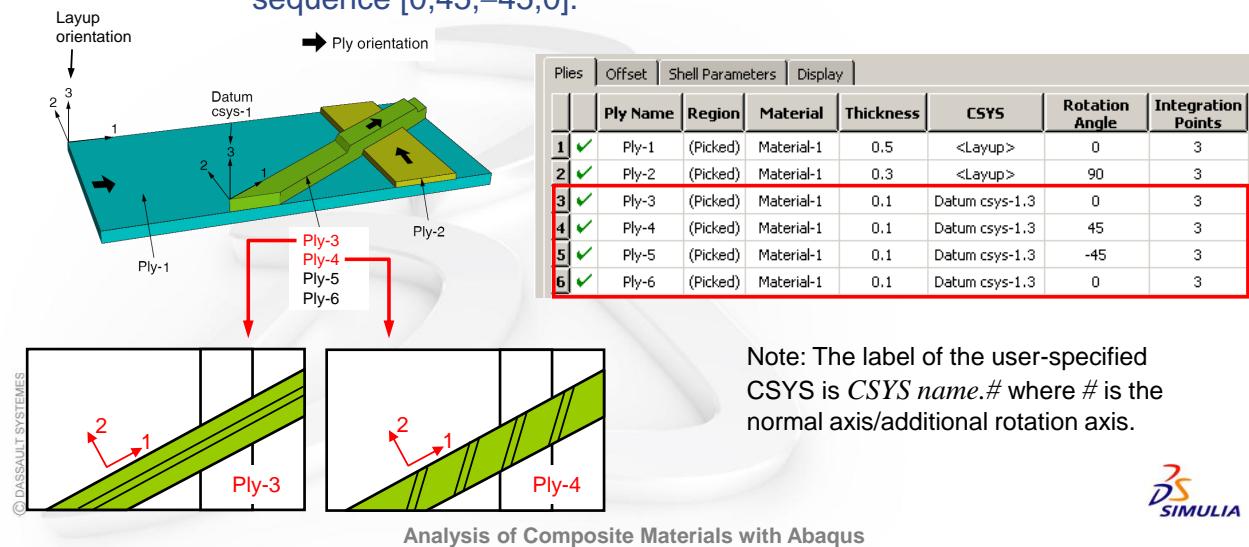


	Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
1	Ply-1	(Picked)	Material-1	0.5	<Layup>	0	3
2	Ply-2	(Picked)	Material-1	0.3	<Layup>	90	3
3	Ply-3	(Picked)	Material-1	0.3	Datum csys-1.3	0	3

Analysis of Composite Materials with Abaqus

Understanding Composite Layup Orientations

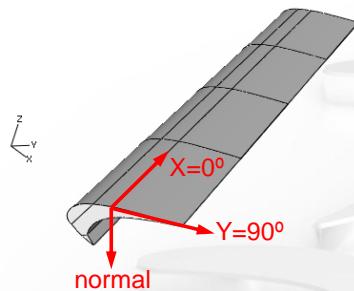
- For plies that are not aligned along the layup orientation, the user-specified reference coordinate system makes it easy to define their orientations.
 - For example, consider an extension to the previous model where the top strip now consists of four plies arranged in the stacking sequence [0,45,-45,0].



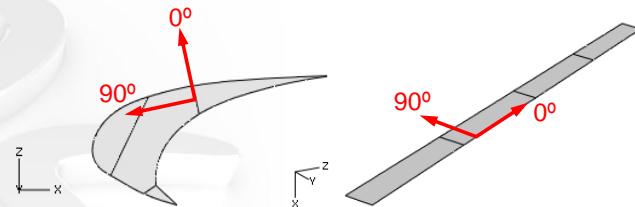
Understanding Composite Layup Orientations

- Defining the layup and ply orientations

- Example: Composite wing slat



Layup orientations for skin of the slat



Layup orientations for ribs and spars of the slat

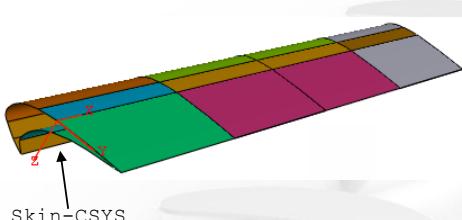
© DASSAULT SYSTEMES



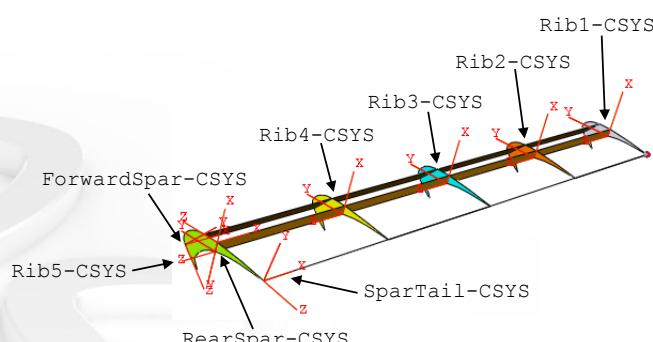
Analysis of Composite Materials with Abaqus

Understanding Composite Layup Orientations

- Create the datum coordinate systems as indicated in the following figures.



Layup orientations for skin of the slat



Layup orientations for ribs and spars of the slat

© DASSAULT SYSTEMES

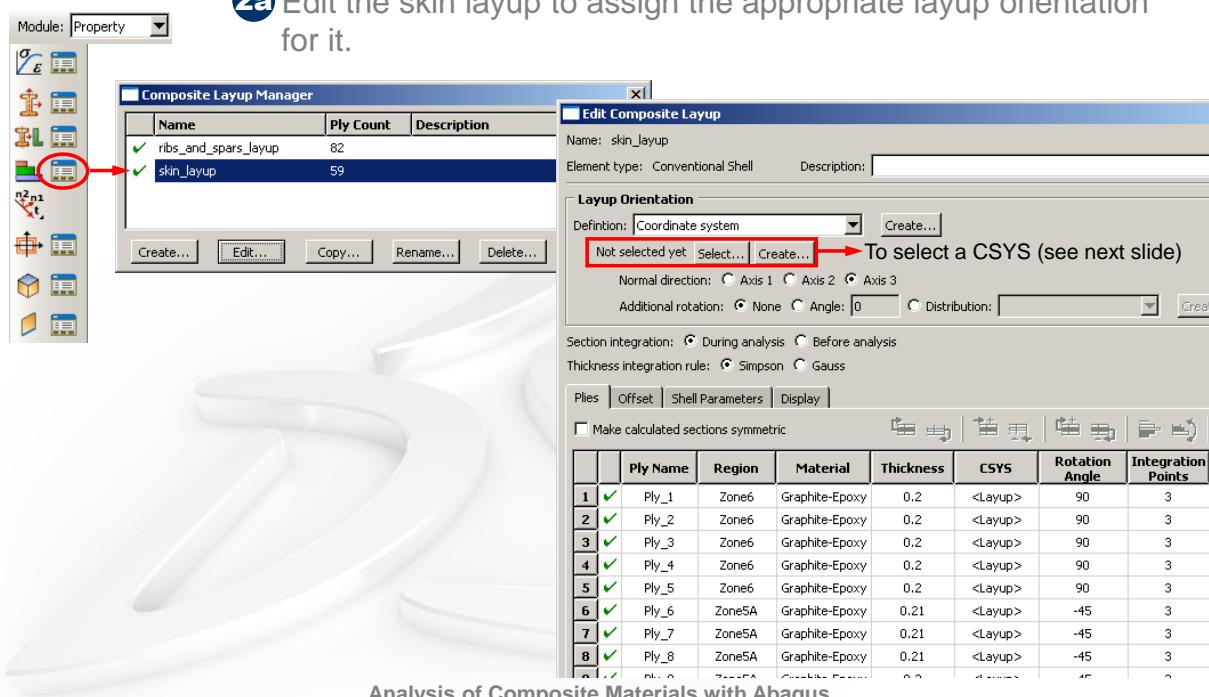


Analysis of Composite Materials with Abaqus

Understanding Composite Layup Orientations

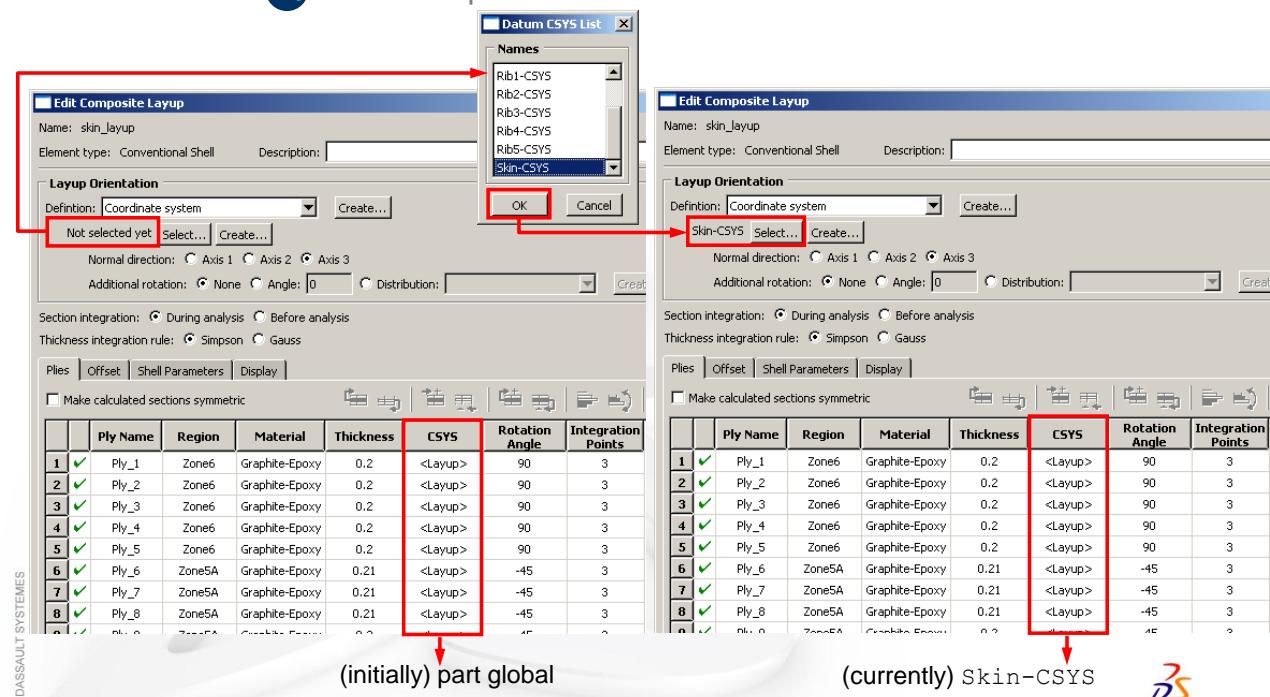
2 Define the layup orientation for the skin layup.

2a Edit the skin layup to assign the appropriate layup orientation for it.



Understanding Composite Layup Orientations

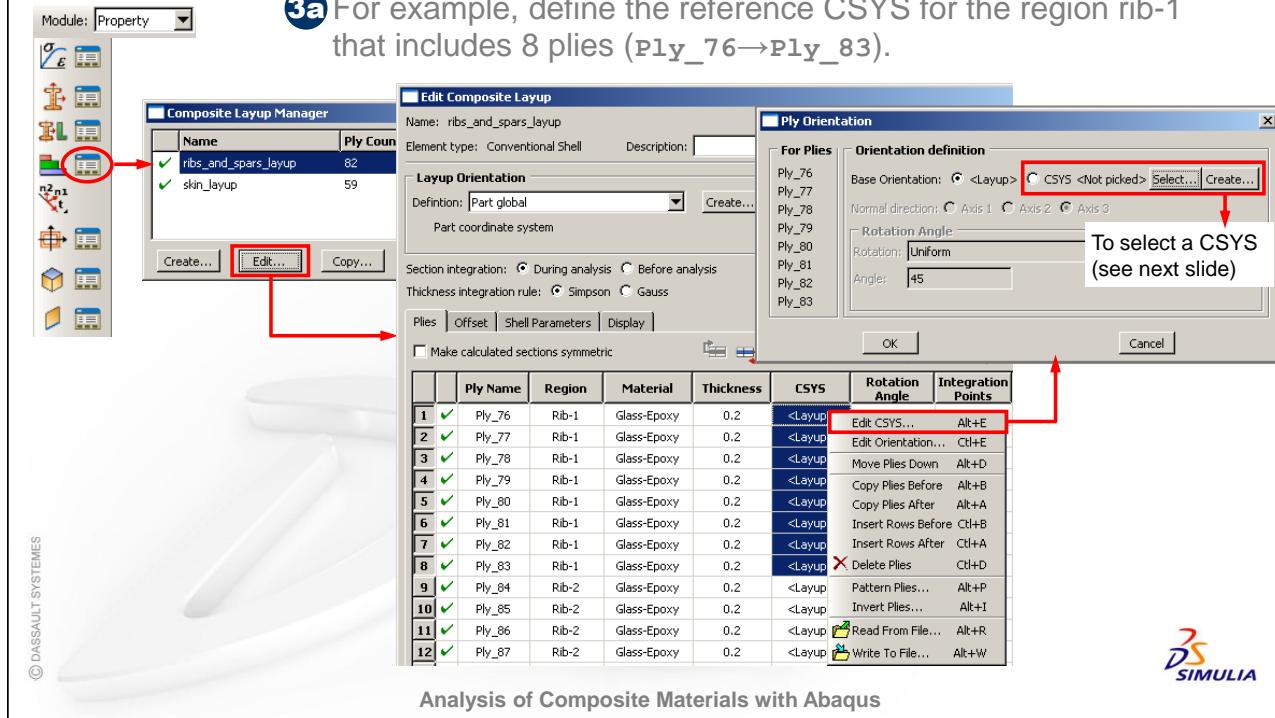
2b Select the predefined datum CSYS named skin-CSYS.



Understanding Composite Layup Orientations

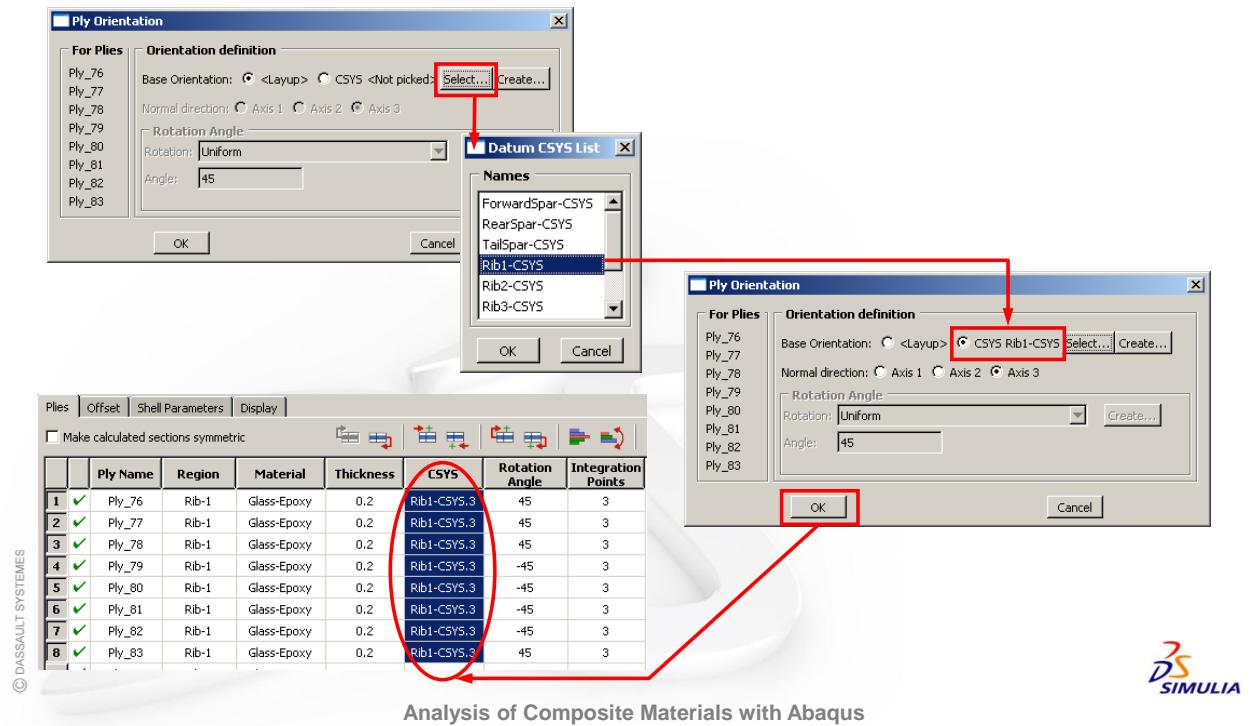
3 Define the reference coordinate systems for ribs and spars.

3a For example, define the reference CSYS for the region rib-1 that includes 8 plies (Ply_76→Ply_83).



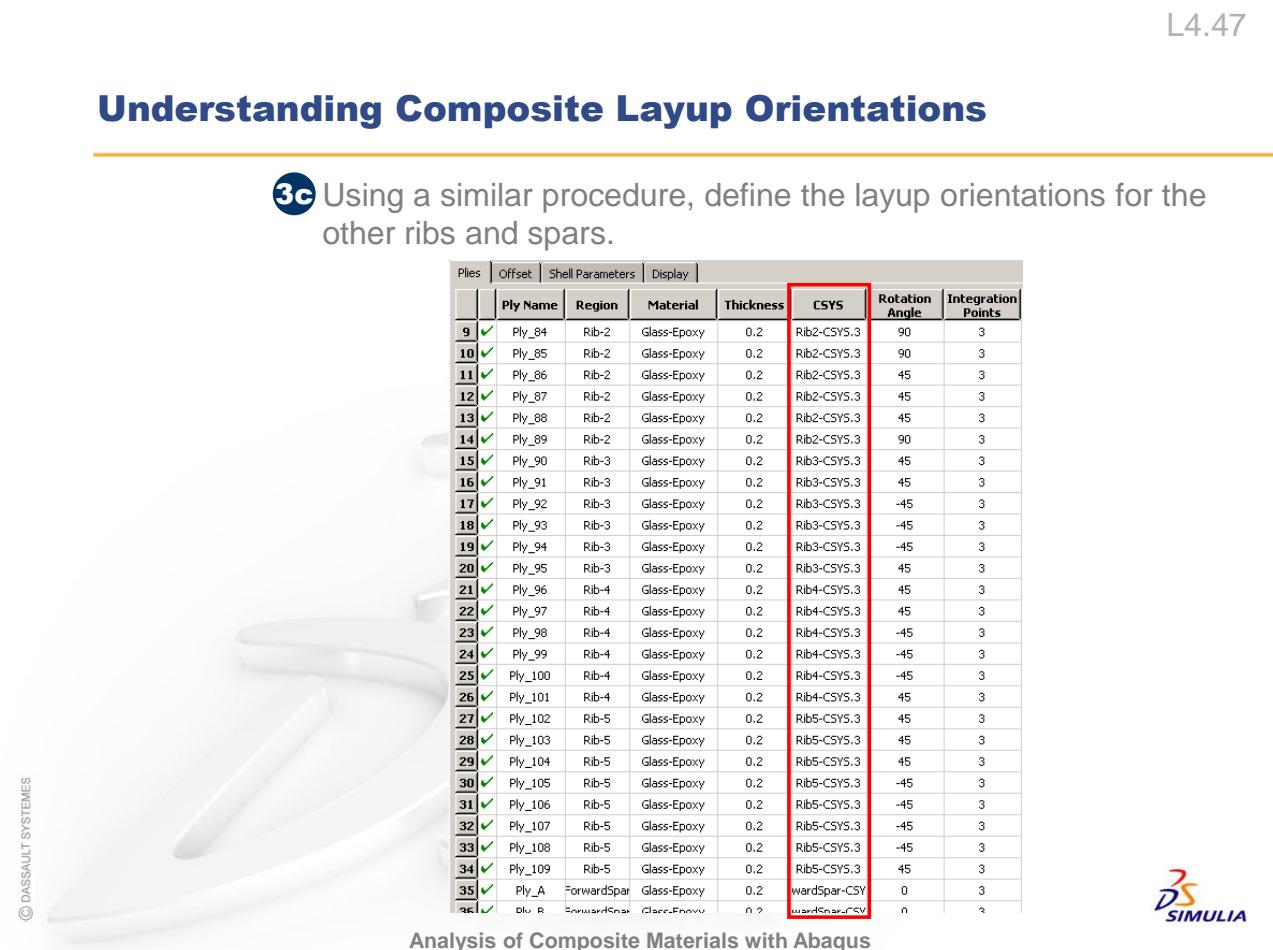
Understanding Composite Layup Orientations

3b Select the predefined datum coordinate system Rib1-CSYS.



Understanding Composite Layup Orientations

- 3c** Using a similar procedure, define the layup orientations for the other ribs and spars.



The table lists 36 plies (Ply_84 to Ply_A) across various regions (Rib-2, Rib-3, Rib-4, Rib-5, ForwardSpars). Each row includes Ply Name, Region, Material (Glass-Epoxy), Thickness (0.2), CSYS (e.g., Rib2-CSYS, Rib3-CSYS, Rib4-CSYS, Rib5-CSYS, wardSpars-CSY), Rotation Angle, and Integration Points. The last two rows are for the ForwardSpars.

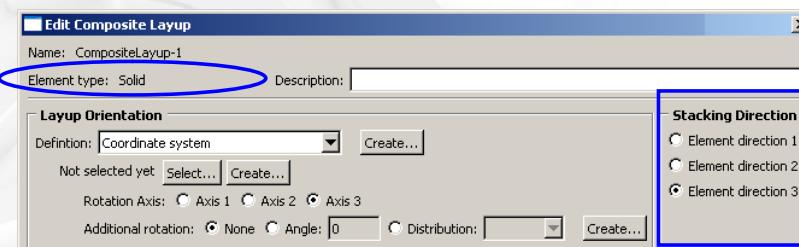
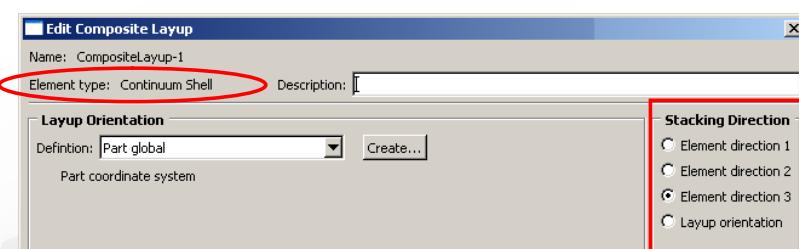
	Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
9	Ply_84	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS,3	90	3
10	Ply_85	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS,3	90	3
11	Ply_86	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS,3	45	3
12	Ply_87	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS,3	45	3
13	Ply_88	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS,3	45	3
14	Ply_89	Rib-2	Glass-Epoxy	0.2	Rib2-CSYS,3	90	3
15	Ply_90	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS,3	45	3
16	Ply_91	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS,3	45	3
17	Ply_92	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS,3	-45	3
18	Ply_93	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS,3	-45	3
19	Ply_94	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS,3	-45	3
20	Ply_95	Rib-3	Glass-Epoxy	0.2	Rib3-CSYS,3	45	3
21	Ply_96	Rib-4	Glass-Epoxy	0.2	Rib4-CSYS,3	45	3
22	Ply_97	Rib-4	Glass-Epoxy	0.2	Rib4-CSYS,3	45	3
23	Ply_98	Rib-4	Glass-Epoxy	0.2	Rib4-CSYS,3	-45	3
24	Ply_99	Rib-4	Glass-Epoxy	0.2	Rib4-CSYS,3	-45	3
25	Ply_100	Rib-4	Glass-Epoxy	0.2	Rib4-CSYS,3	-45	3
26	Ply_101	Rib-4	Glass-Epoxy	0.2	Rib4-CSYS,3	45	3
27	Ply_102	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	45	3
28	Ply_103	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	45	3
29	Ply_104	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	45	3
30	Ply_105	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	-45	3
31	Ply_106	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	-45	3
32	Ply_107	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	-45	3
33	Ply_108	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	-45	3
34	Ply_109	Rib-5	Glass-Epoxy	0.2	Rib5-CSYS,3	45	3
35	Ply_A	ForwardSpars	Glass-Epoxy	0.2	wardSpars-CSY	0	3
36	Dls_R	ForwardSpars	Glass-Epoxy	n ?	wardSpars-CSY	n	?



Analysis of Composite Materials with Abaqus

Understanding Composite Layup Orientations

- For a continuum shell/solid composite layup, you can directly specify a stacking direction.
 - Choosing a stacking direction of the continuum shell elements is also discussed in Appendix 3, “Modeling Issues for Continuum Shell Elements.”



Analysis of Composite Materials with Abaqus

Defining Composite Layup Output

© DASSAULT SYSTEMES

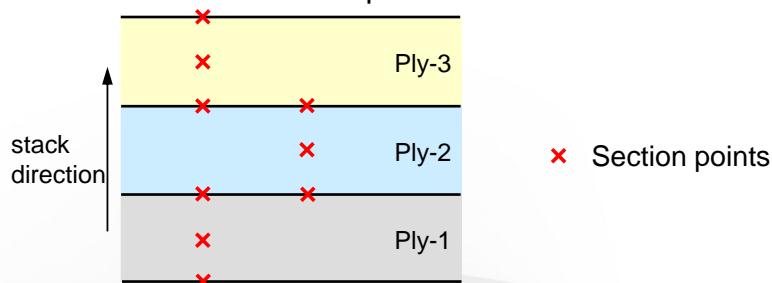


L4.50

Defining Composite Layup Output

• Overview

- Section points are integration points through the element thickness and are used as the locations to output the results.



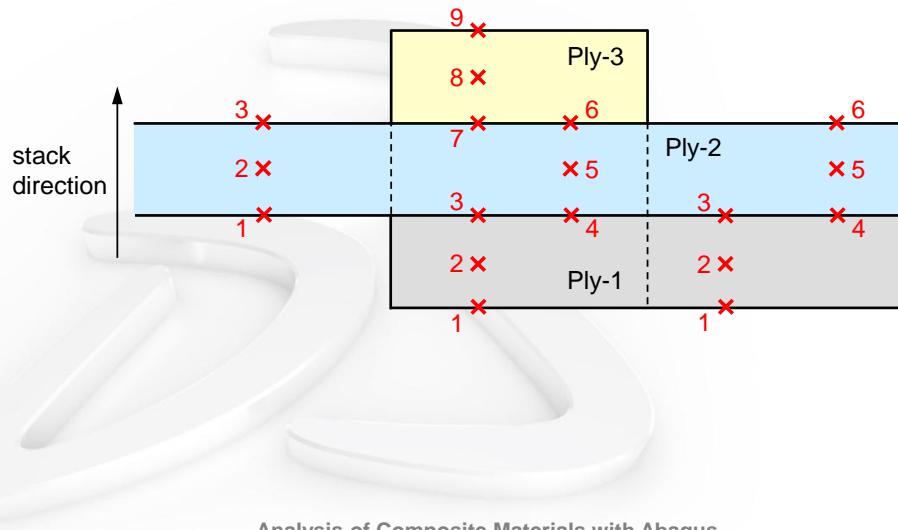
- By default, a shell composite layup has three integration points for each ply; a solid composite layup has one integration point for each ply.
 - For the shell composite layup integrated during the analysis, you can specify the number of integration points in each ply.
 - For a pre-integrated shell layup, three integration points are used for each ply in the layup.

© DASSAULT SYSTEMES



Defining Composite Layup Output

- Section points are numbered sequentially from the bottom of the bottom ply to the top of the top ply.
 - Note that the bottom ply is the first ply defined in a composite layup.
 - Example: Section point numbering for a three-ply composite layup.



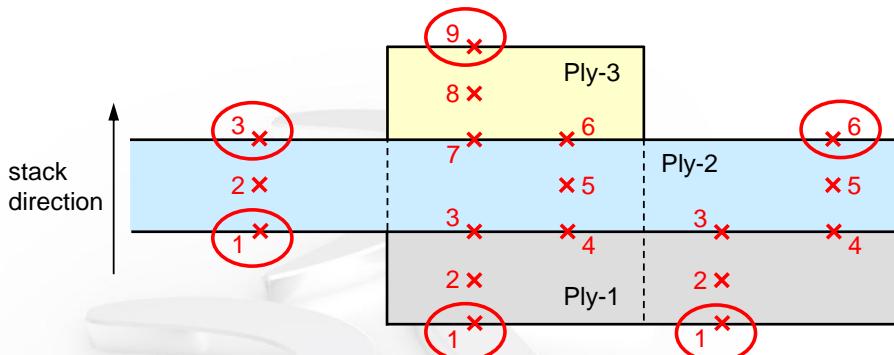
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Defining Composite Layup Output

- By default, Abaqus writes field output data from only the top and bottom of a conventional/continuum shell composite layup (circled section points).



- To output data from individual plies, create a field/history output request for a composite layup.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Defining Composite Layup Output

- Define composite layup output
 - Example: Three-ply composite layup

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

1. Select a particular layup

2. Choose output variables

3. Choose position(s) for results

SIMULIA

Defining Composite Layup Output

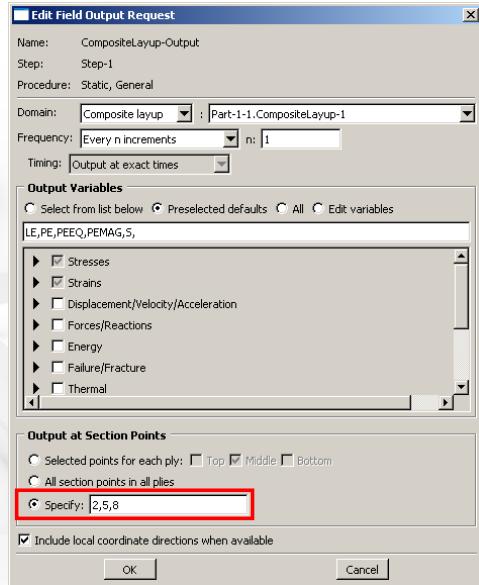
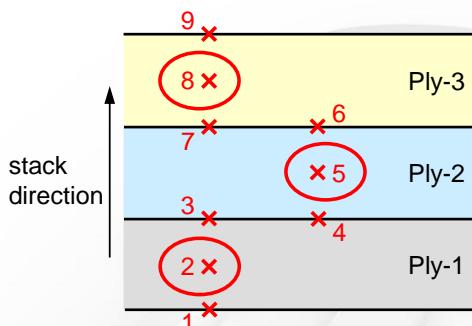
- Example: Composite Wing Slat
 - For the skin layup, request output at all section points in all plies; for the spars and ribs layup, at the middle section point for each ply.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Defining Composite Layup Output

- Alternatively use the **Specify** option to specify the section point numbers.
 - For example, request output at the middle section points of each ply (circled section points).



Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

Viewing a Composite Layup

- Abaqus/CAE provides several tools to view a composite layup in pre- and postprocessing:
 - Display group by composite layup or ply
 - Color code by composite layup or ply
 - Color code by section shows sections computed from the layup.
 - Ply stack plot
 - Display a “core sample” of the layup for a probed region.
 - Ply-based postprocessing
 - View contour, symbol, or material orientation plots by ply.
 - Envelope plot
 - Display the critical value (absolute maximum, maximum, or minimum) across all of the plies in the layup in a contour plot.
 - Through thickness X-Y plot

© DASSAULT SYSTEMES

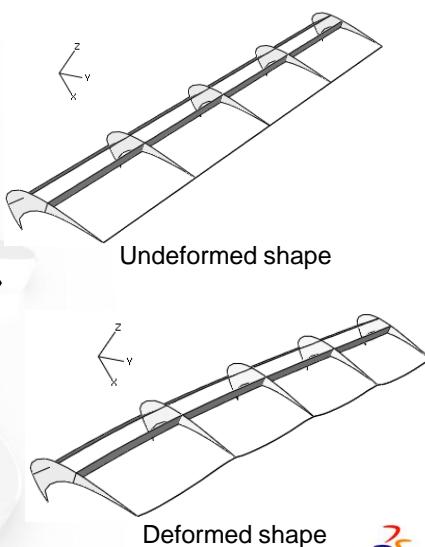
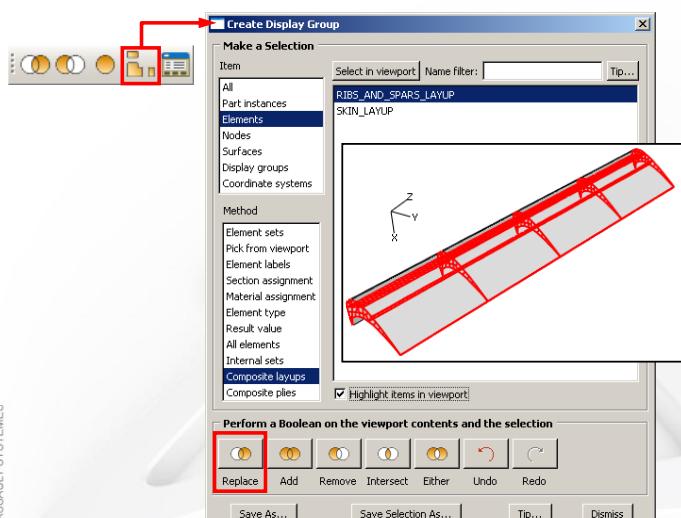


Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- Display by composite layup or ply
 - Example: Composite wing slat

1 Create a Display Group based on Composite layups to display the ribs and spars layup.



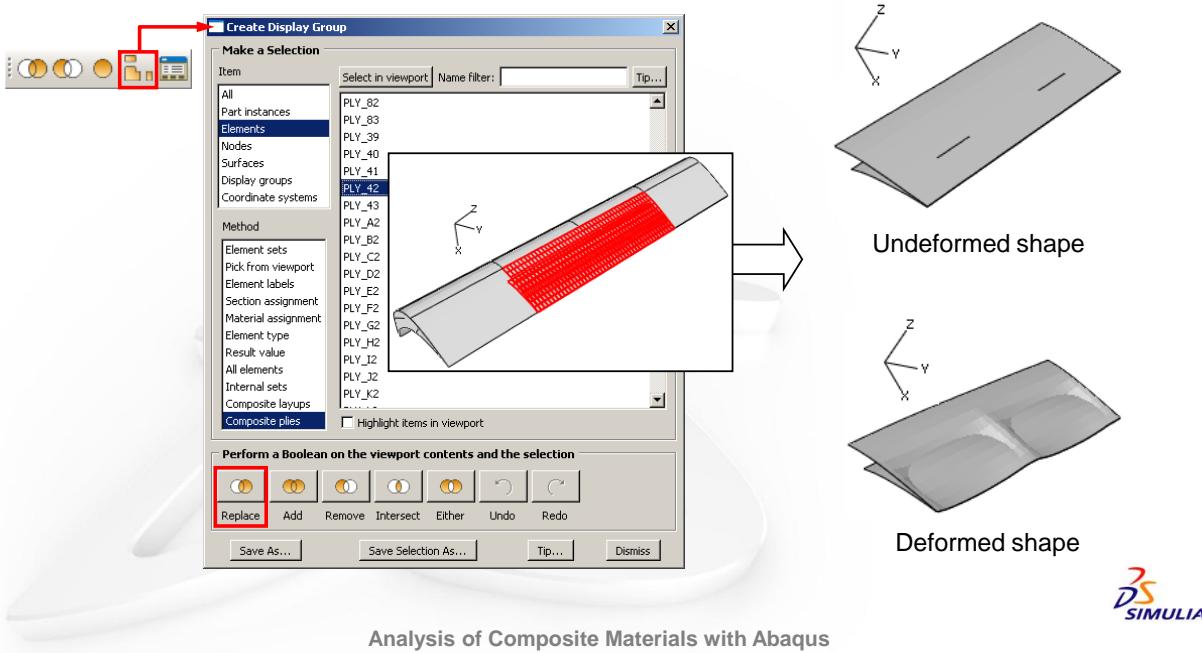
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- 2** Create a Display Group based on Composite plies to display a ply (e.g., PLY_42) in the skin composite layup

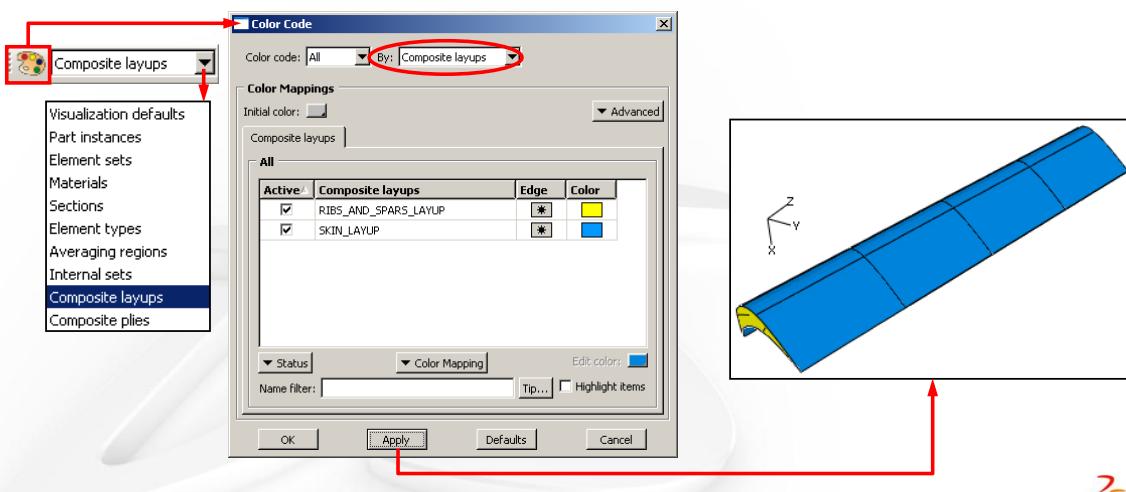


Viewing a Composite Layup

- Color code by composite layup or ply

1 Color code by composite layups

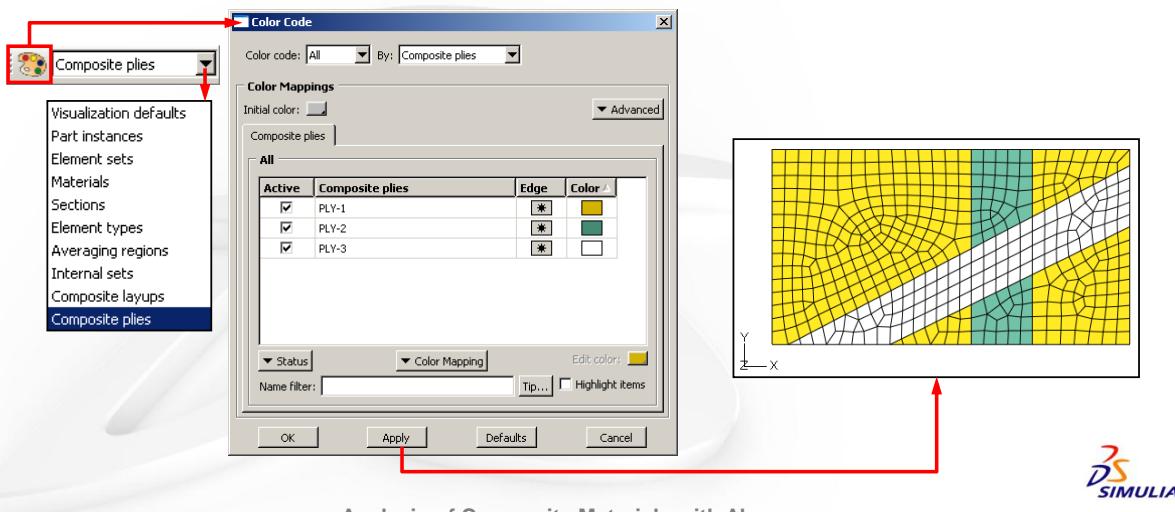
- Example: Composite wing slat



Viewing a Composite Layup

2 Color code by composite ply

- Abaqus/CAE applies color coding on only one ply in a region, which by default is the last ply (in alphabetical order).
- To view a different ply, deactivate selected plies.
- Example: Three-ply composite layup

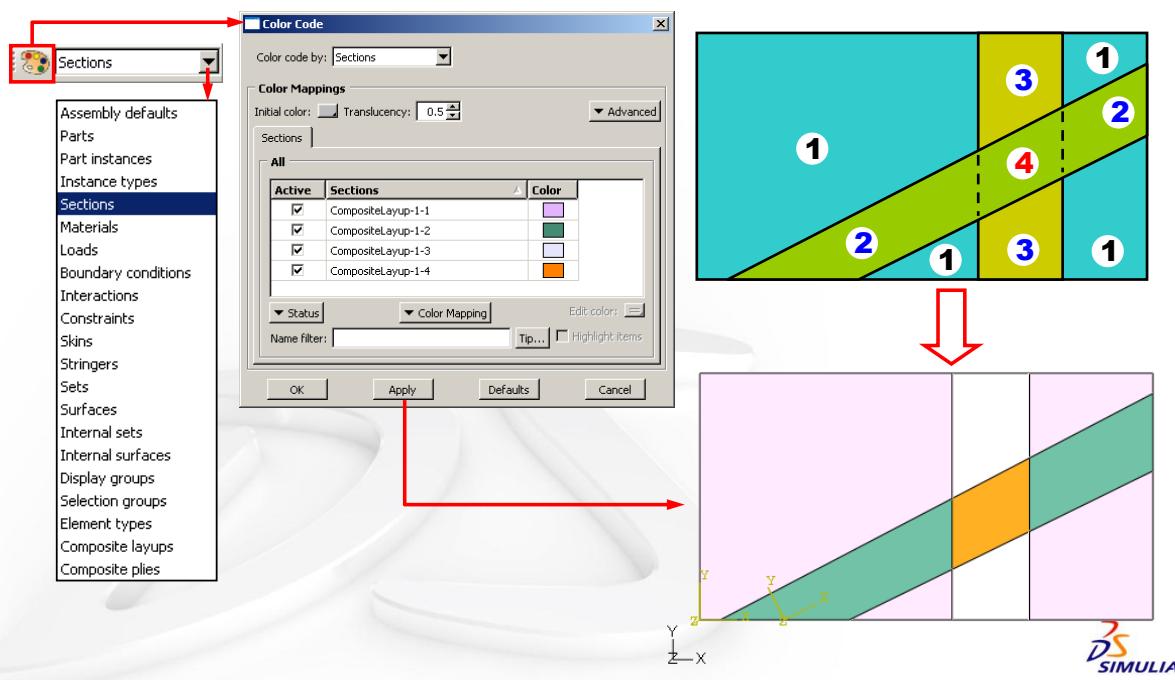


© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- Color code by section shows sections computed from the layup.



© DASSAULT SYSTEMES

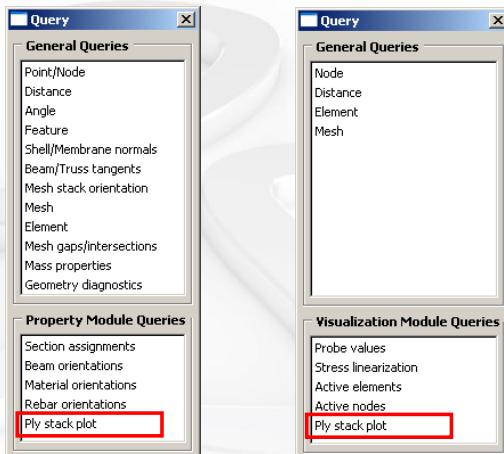
Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- **Ply stack plot**

- is a graphical representation from a selected region of a composite model.
- can be accessed using the **Query** tool in either the Property module or the Visualization module.

© DASSAULT SYSTEMES

**SIMULIA**

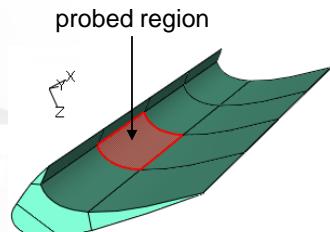
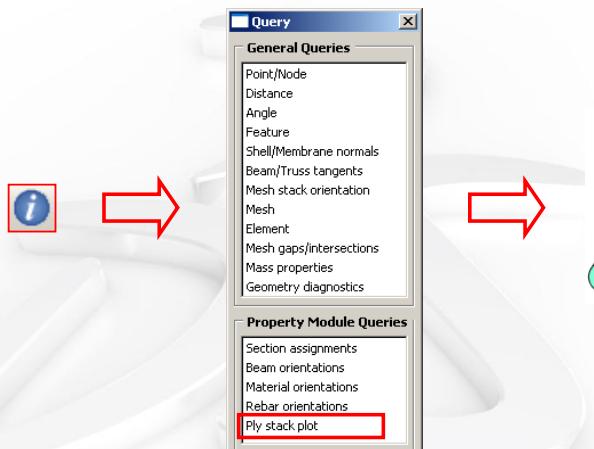
Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- **Display the ply stack plot**

- Example: Composite slat wing
 - In the Property module, click the **Query** tool and select a region to be queried for ply stack plot.

© DASSAULT SYSTEMES

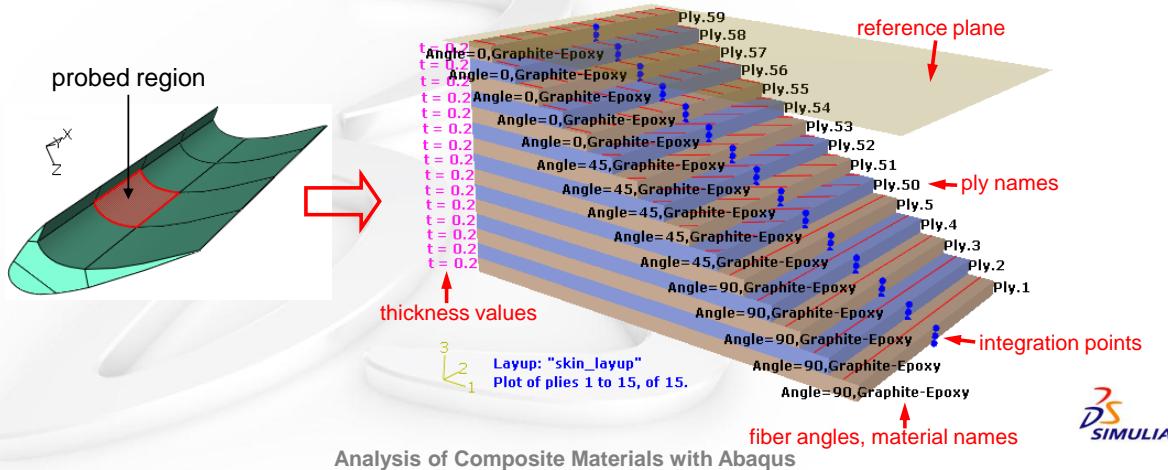
**SIMULIA**

Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- The ply stack plot will reside in its own viewport.
- The user can do the usual view manipulations and printing.
- The staircase appearance has no physical meaning.
- Lines drawn on the plies show the orientation angles with respect to the local 1-direction, if the layup orientation is used to define the ply's orientation.

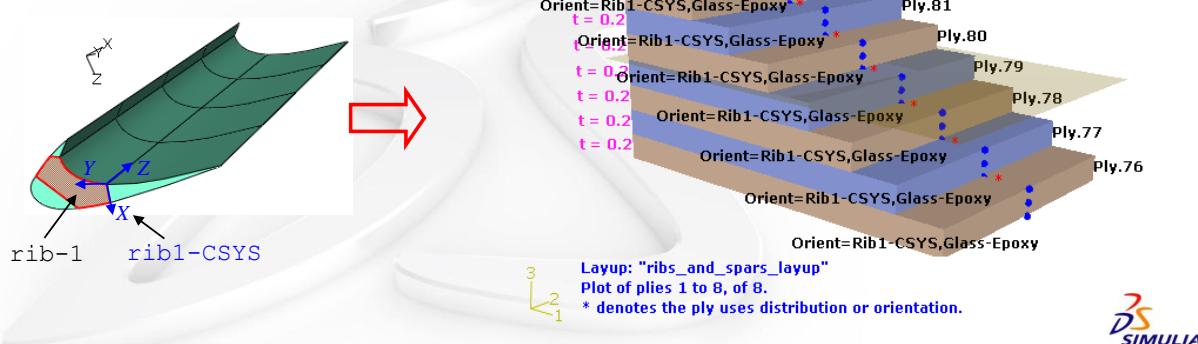
© DASSAULT SYSTEMES



Viewing a Composite Layup

- Note: If a user-defined reference CSYS is used to define a ply's orientation, Abaqus/CAE cannot project the CSYS onto the shell element without knowing the spatial orientation of the element; therefore, no fiber direction will be drawn on the plies.
 - For example, the ply stack plot of the region `rib-1`.
 - For the same reason, no fiber direction is drawn on the plies if you use a distribution to define the ply's orientation.

© DASSAULT SYSTEMES



Viewing a Composite Layup

- Customize the appearance of a ply stack plot

- The **Ply Stack Plot Options** dialog box is used to customize the appearance of a ply stack plot and can be accessed in either the Property module or the Visualization module.

© DASSAULT SYSTEMES

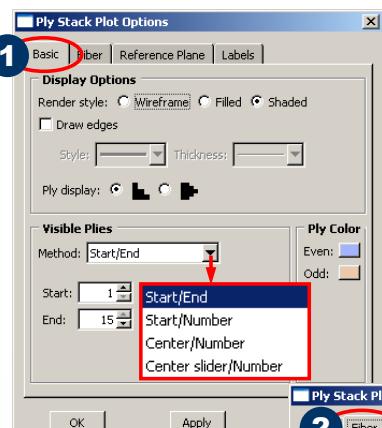


Analysis of Composite Materials with Abaqus

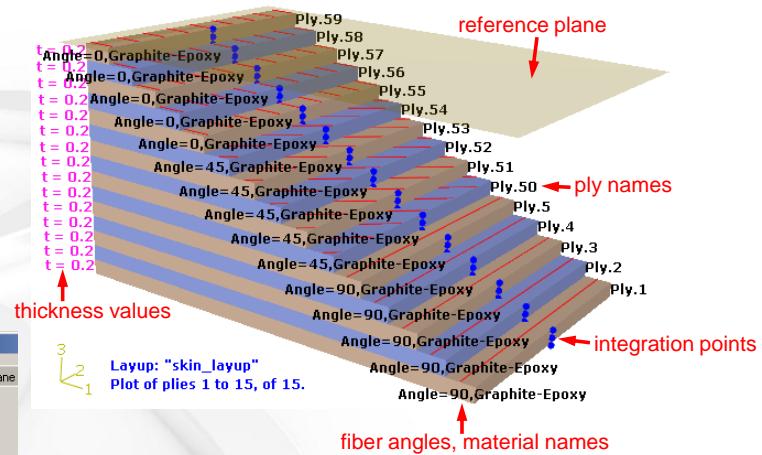
Viewing a Composite Layup

- Control the display options to view a certain number of plies.
- Customize the appearance of the plot.

1



© DASSAULT SYSTEMES

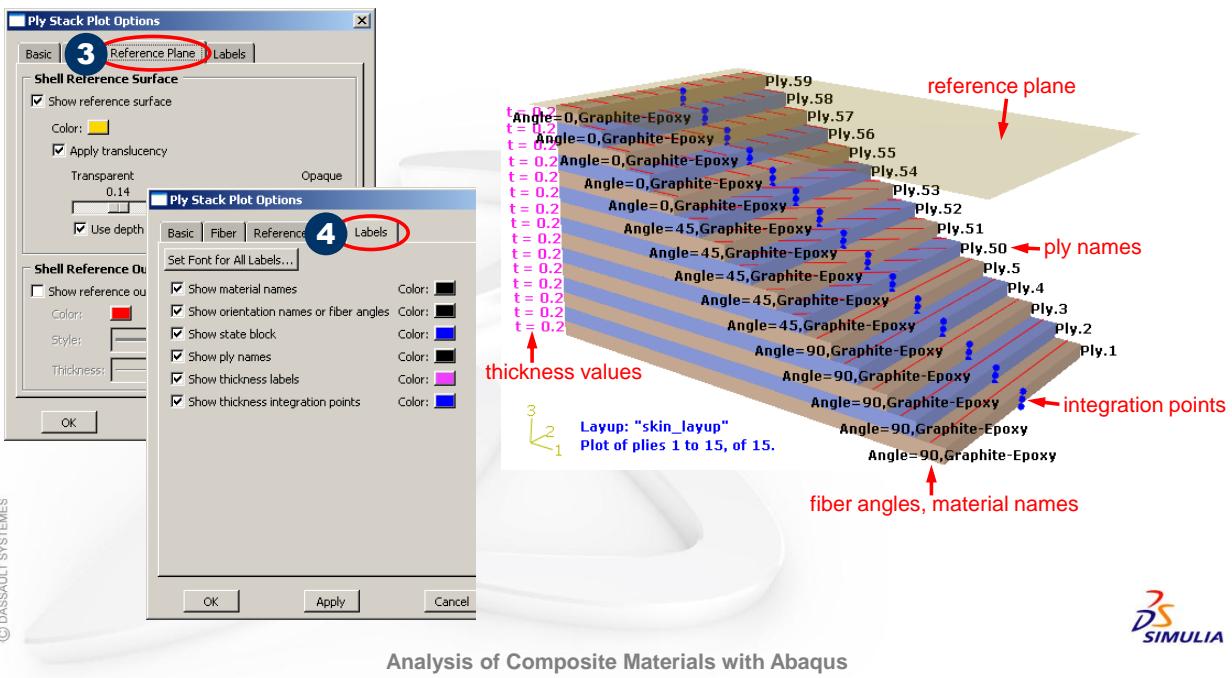


Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

3 Control the appearance of the reference surface.

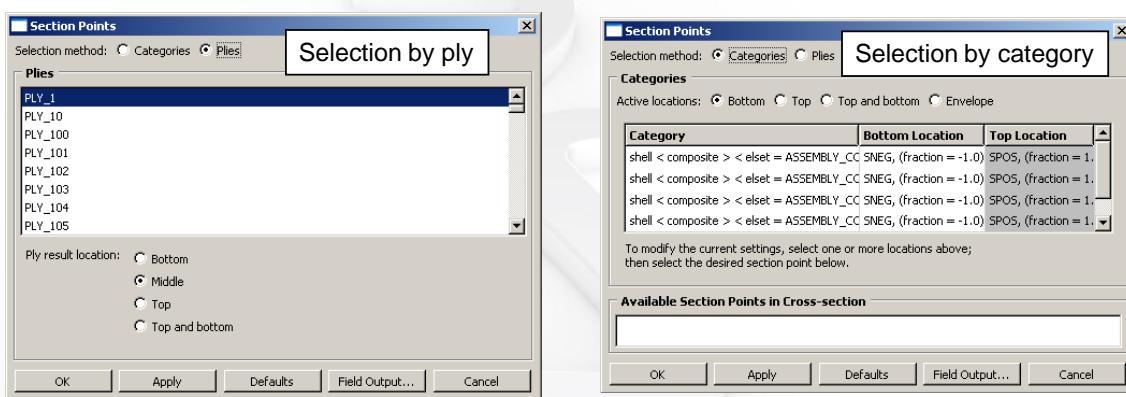
4 Customize the labels.



Viewing a Composite Layup

Ply-based postprocessing

- displays contour, symbol, or material orientation plots by selecting a particular ply and the location in the ply.
- works with section point based postprocessing (selection by category).
 - Section point based postprocessing will be discussed in next lecture, "Alternative Modeling Techniques for Composites."



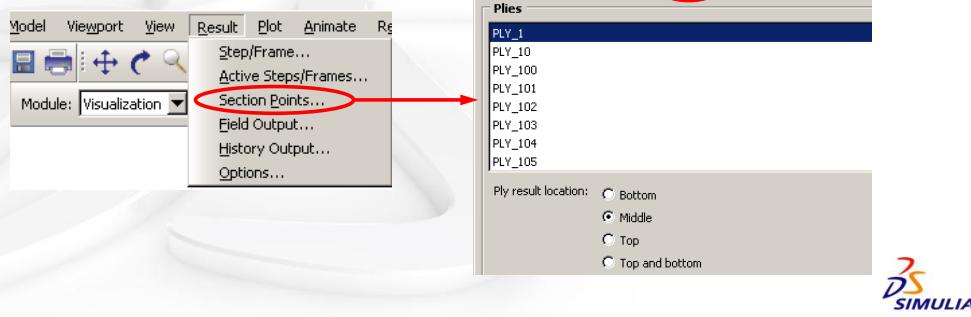
Viewing a Composite Layup

- Example: Composite wing slat

1 Open the **Section Points** dialog box and choose the selection method by ply.

2 Select a ply and ply result location.

- The ply result location can be bottom, middle, top, or both top and bottom.
 - The middle location is either precise (e.g., 3rd out of 5) or imprecise (e.g., 3rd out of 6).
- Only one ply is allowed at a time.

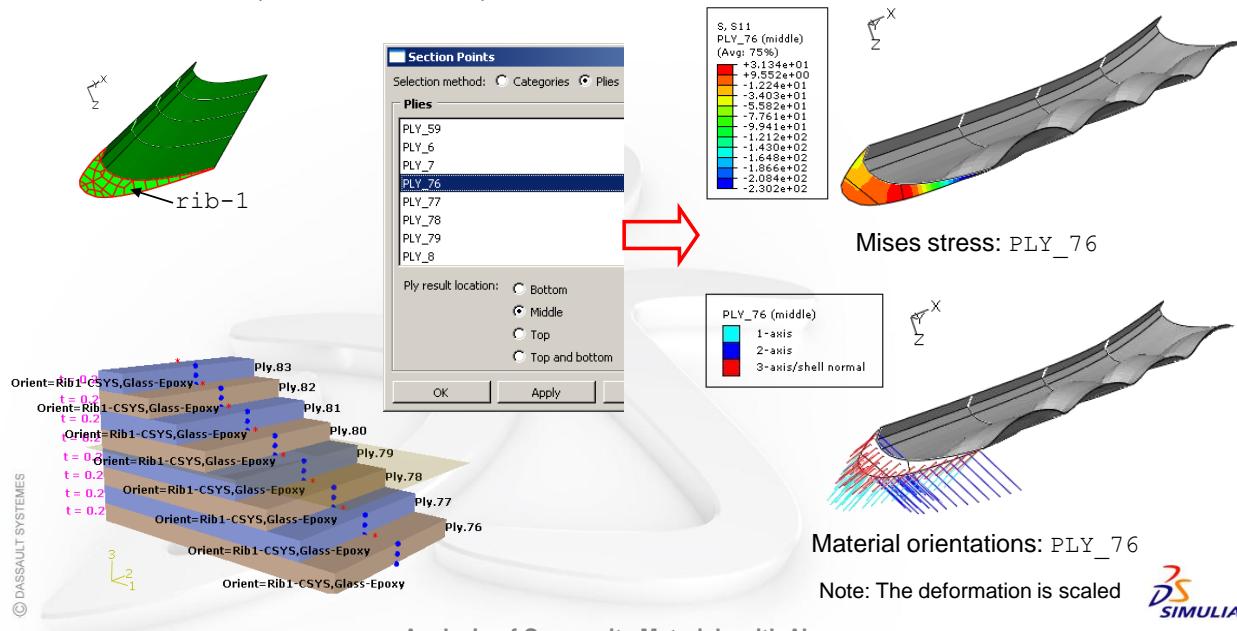


Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

3a Display the results from a selected ply of the rib composite layup:

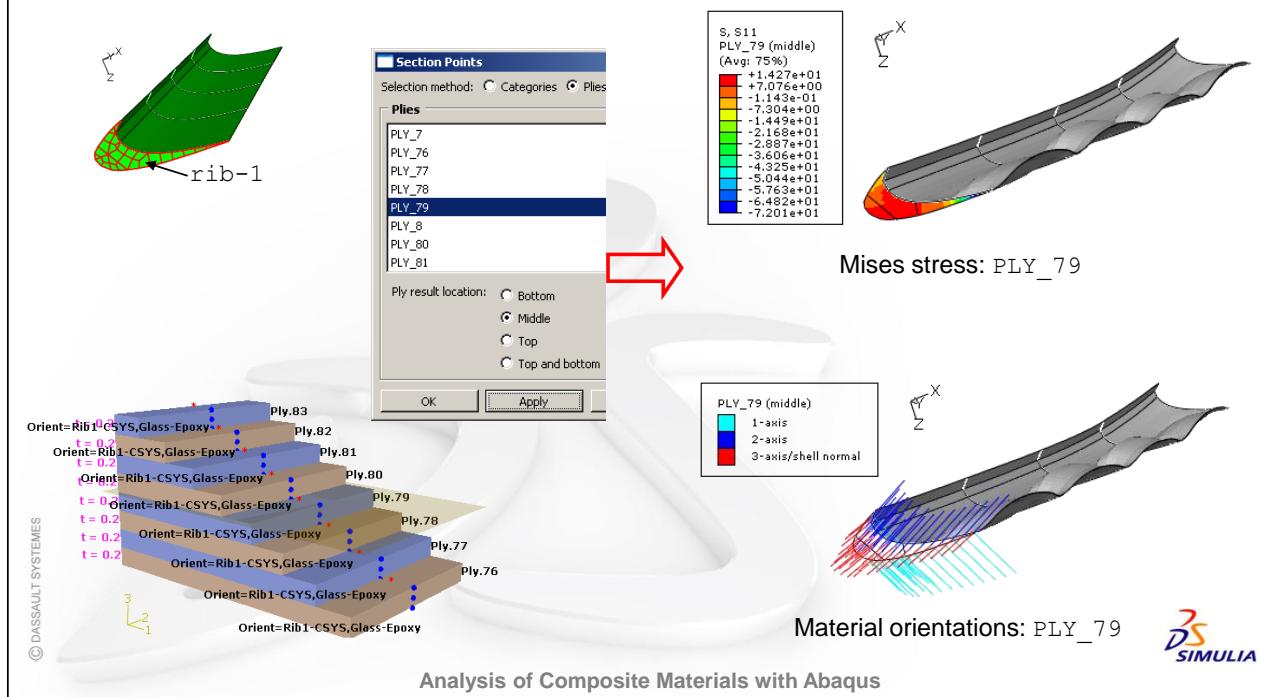
- Any area of the model beyond the specified ply will be colored grey (no results color).



Analysis of Composite Materials with Abaqus

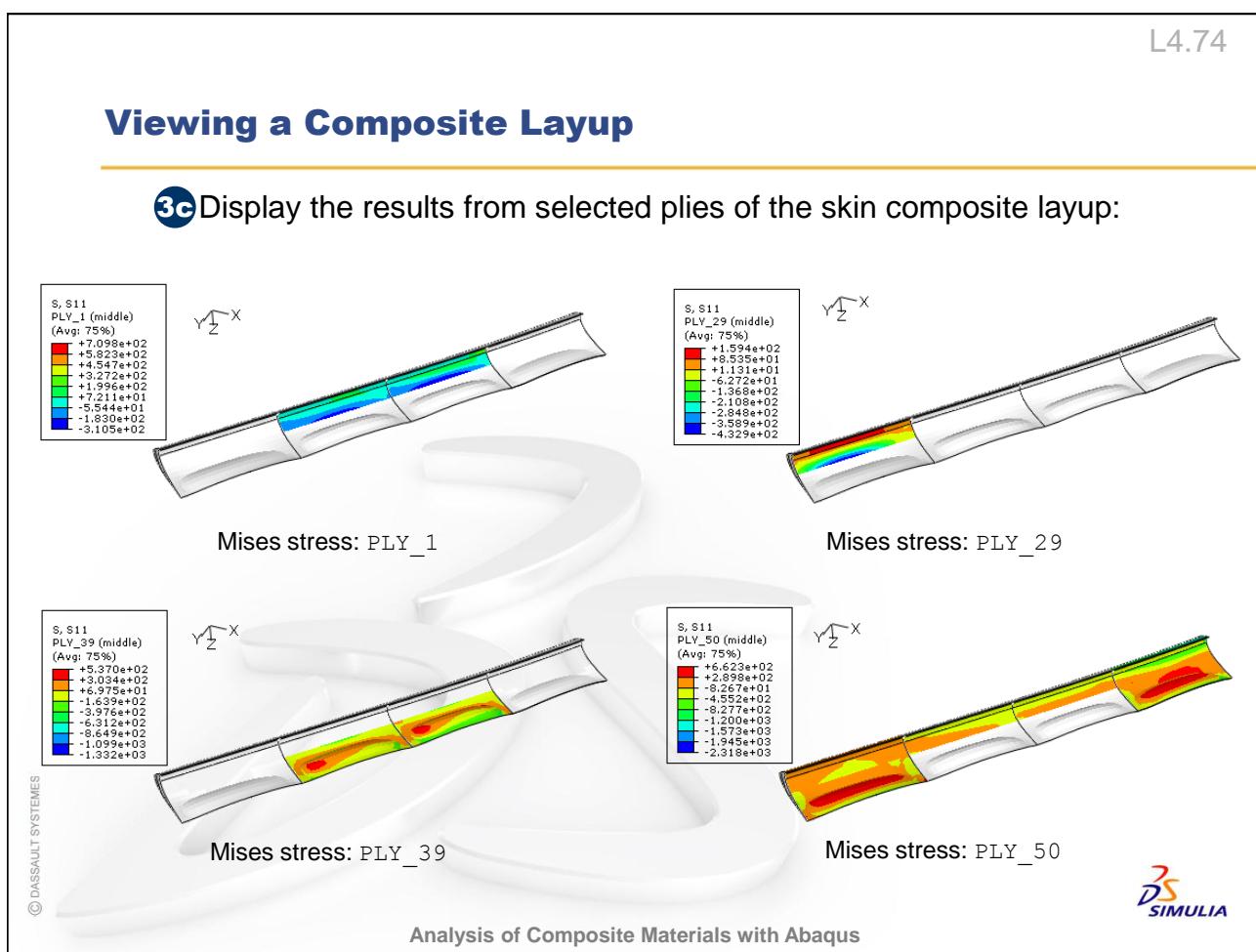
Viewing a Composite Layup

3b Display the results from a selected ply of the rib composite layup:



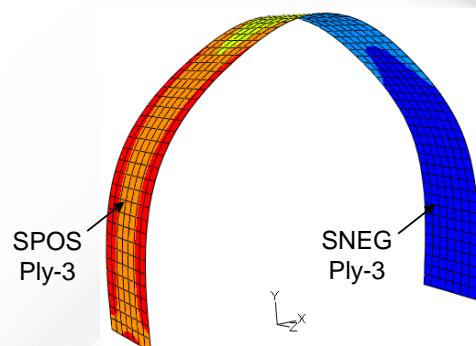
Viewing a Composite Layup

3c Display the results from selected plies of the skin composite layup:

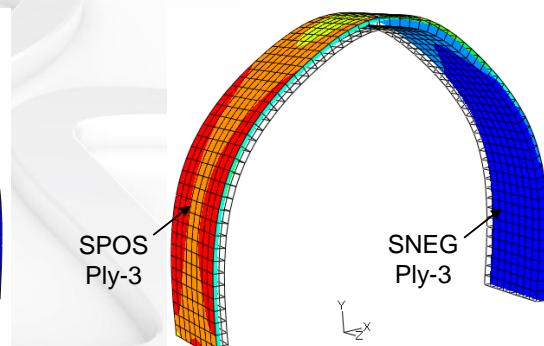


Viewing a Composite Layup

- Contour plots displaying output at both the top and bottom of the selected ply vary in appearance depending on the type of composite layup.
 - In a conventional shell composite layup the two contours appear as a double-sided shell with different contours on each side.
 - In a continuum shell composite layup the two contours appear as distinct single-sided contours at each section point location.



Conventional shell composite layup
(three plies)



Continuum shell composite layup
(three plies)

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Viewing a Composite Layup

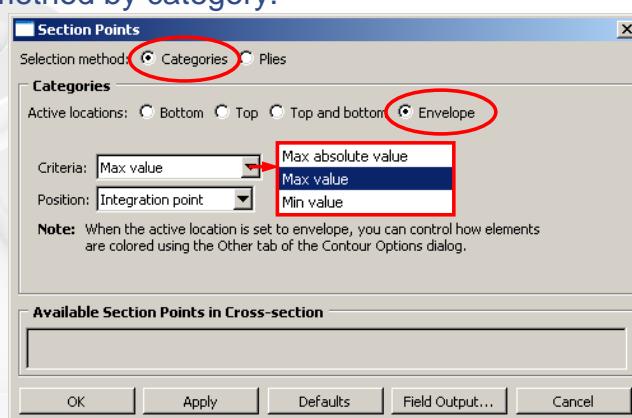
• Envelope plot

- Envelope plot displays the critical value (maximum absolute value, maximum, or minimum) across all of the plies at each material point in the model in a contour plot.
- Example: Composite wing slat

1 Choose the selection method by category.

2 Select

- Envelope as the active location,
- Max value as the criterion, and
- Integration point as the position.



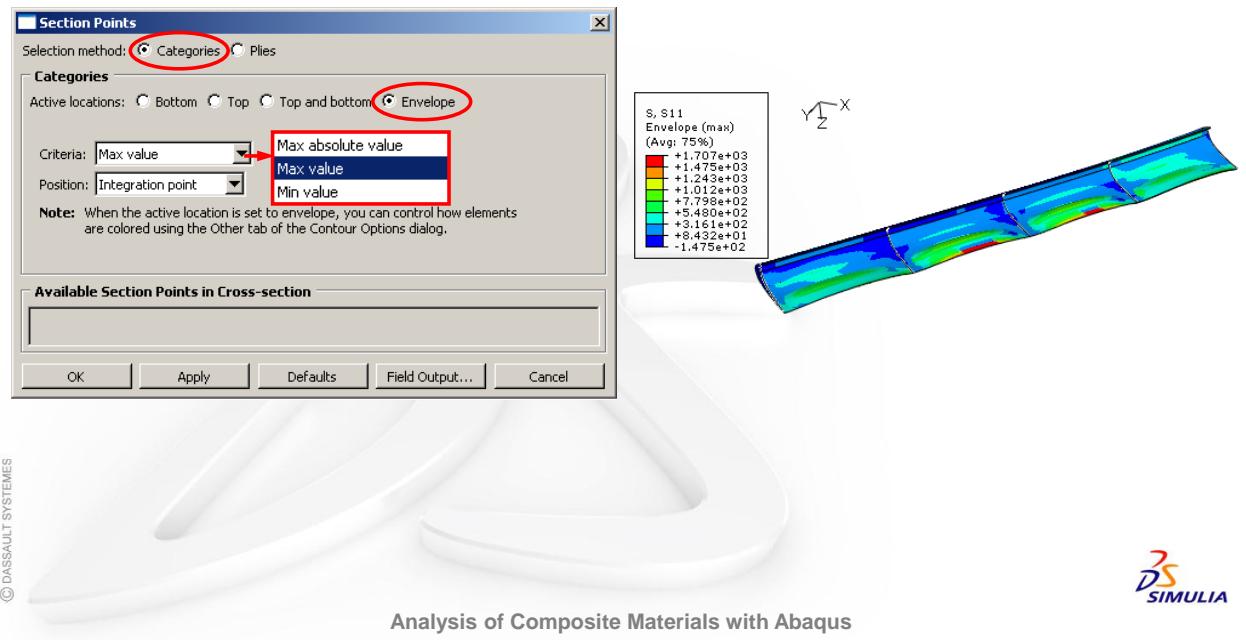
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Viewing a Composite Layup

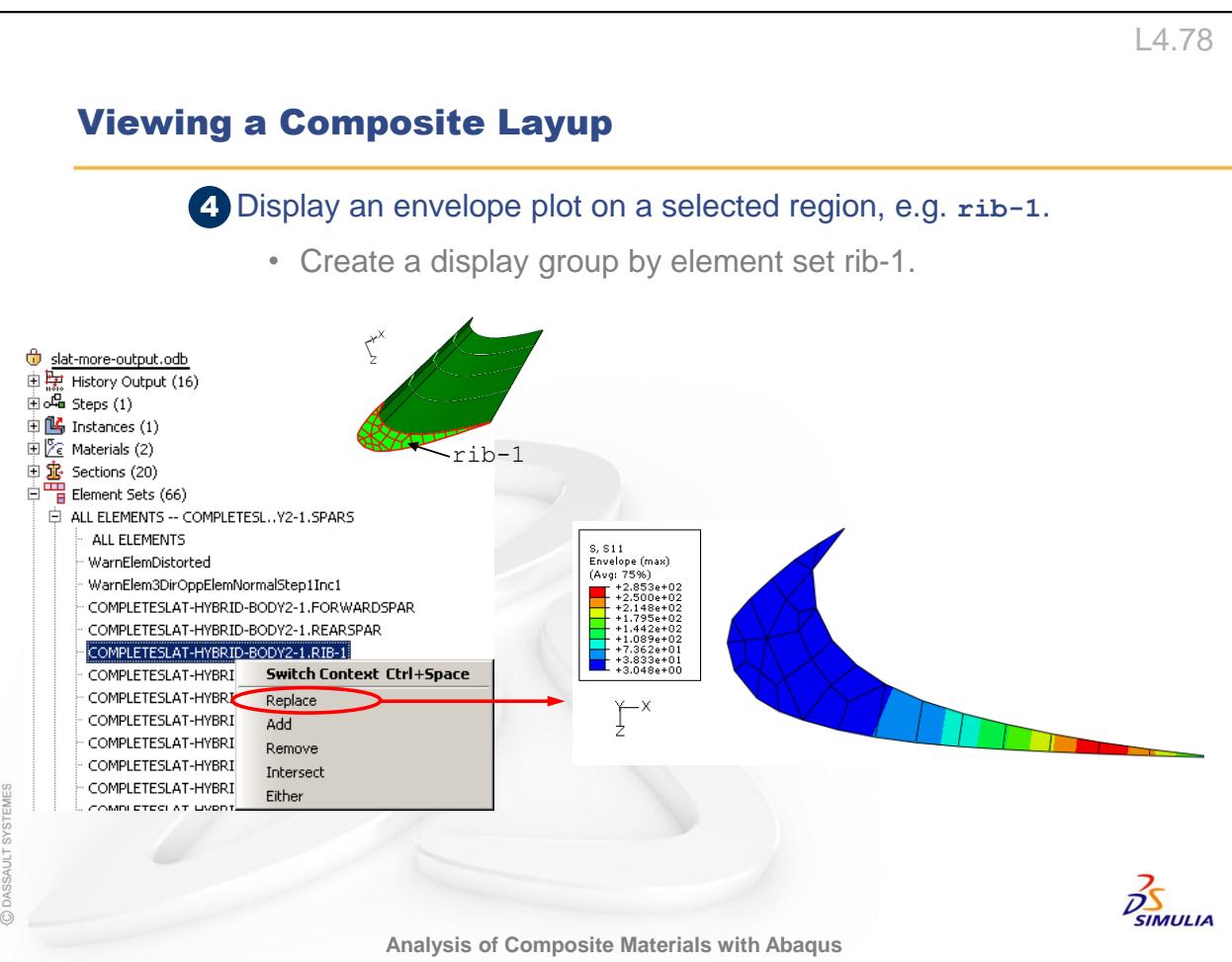
- 3 Plot the envelope plot on the wing slat model.



Viewing a Composite Layup

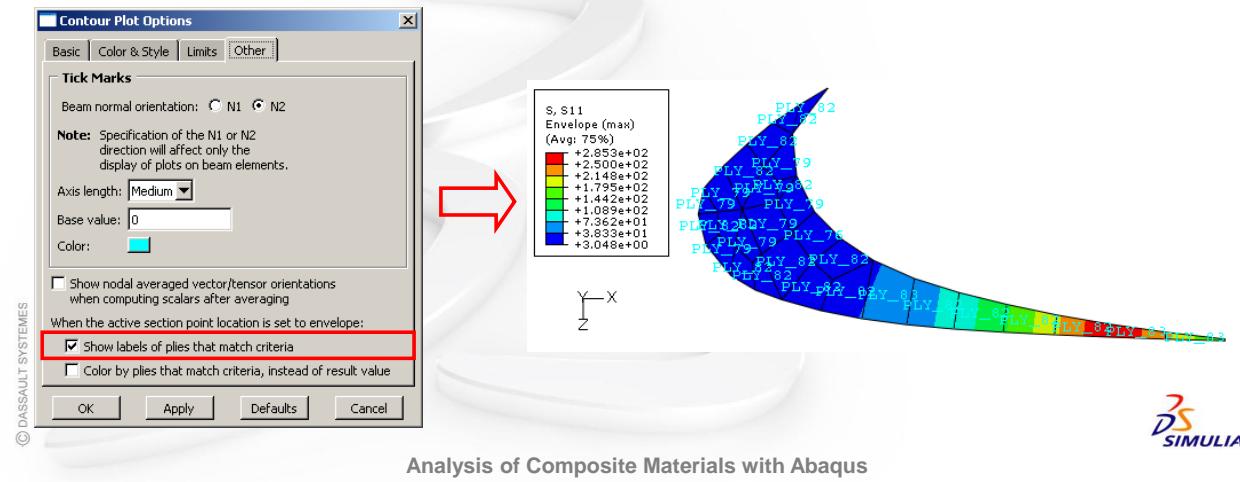
- 4 Display an envelope plot on a selected region, e.g. rib-1.

- Create a display group by element set rib-1.



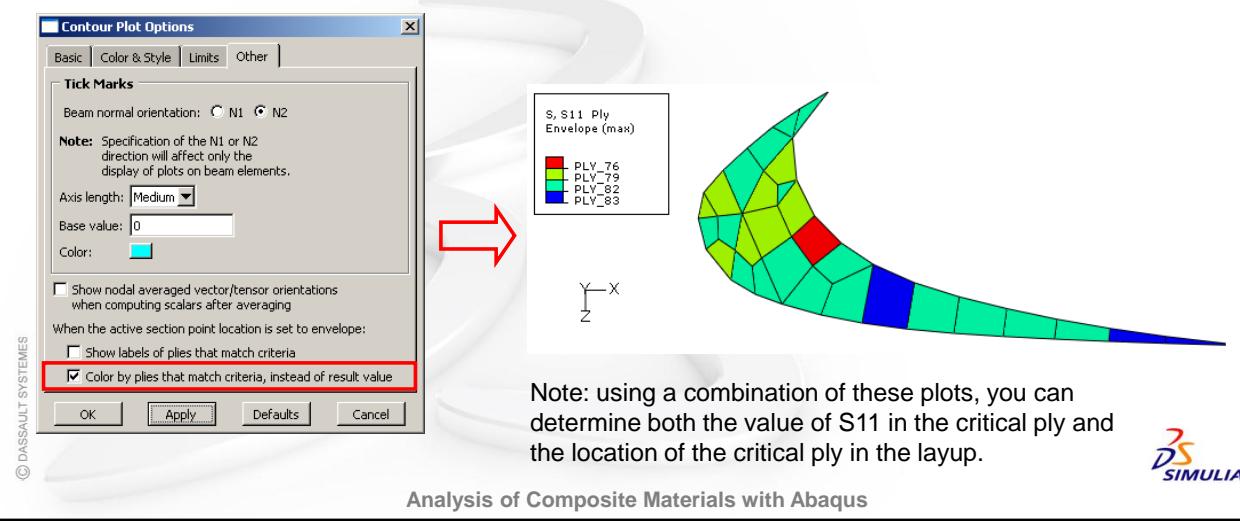
Viewing a Composite Layup

- Determine the critical plies from an envelope plot
 - Show names of critical plies
 - Example: Composite wing slat
 - Create the envelope plot on the selected region, e.g. rib-1.
 - In the Contour Plot Options dialog box, select Show labels of plies that match criteria.



Viewing a Composite Layup

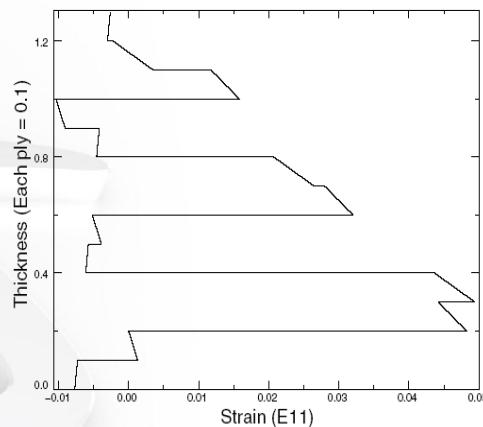
- Show quilt plot of critical plies
 - Example: Composite wing slat
 - Create the envelope plot on the selected region, e.g. rib-1.
 - In the Contour Plot Options dialog box, select Color by plies that match criteria, instead of result value.



Viewing a Composite Layup

- **Through-thickness X–Y plots**

- Display the behavior of the plies across the entire thickness of the layup.
- Read X–Y data from field output results at the section points in a shell element.
- For example, the figure illustrates a through-thickness plot of the strain in the fiber direction through 13 plies of a composite layup.
 - The strain is discontinuous because the orientation of the fiber changes between plies.



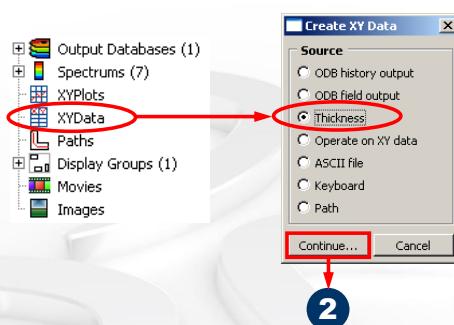
Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- **Creating a through-thickness X–Y plot**

- Example: Composite wing slat

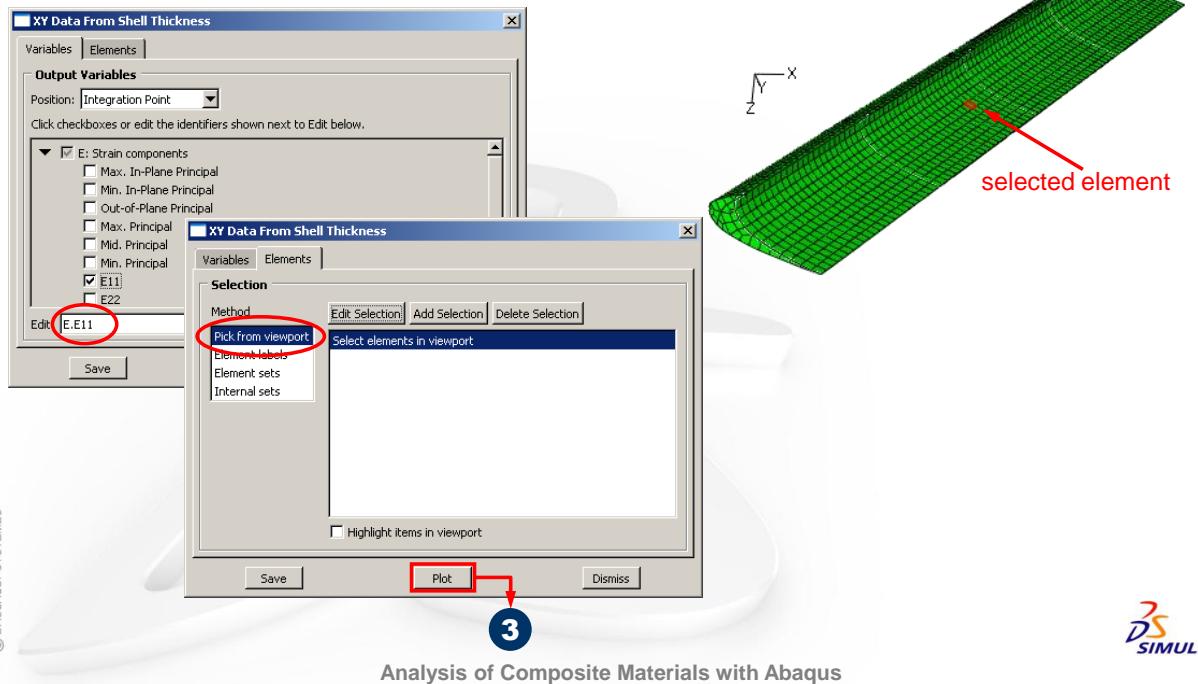
1 Locate the Thickness option.



Analysis of Composite Materials with Abaqus

Viewing a Composite Layup

- 2 Select an output variable for an element (or a set of elements).

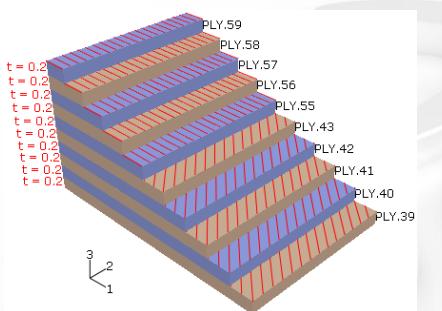


© DASSAULT SYSTEMES

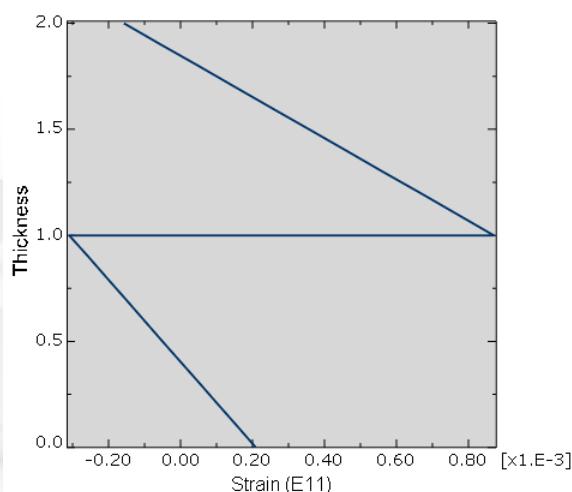


Viewing a Composite Layup

- 3 Plot the through-thickness variation.



The stack plot of the selected element



The through-thickness X-Y plot

© DASSAULT SYSTEMES



Abaqus/CAE Demonstration: Three-ply composite

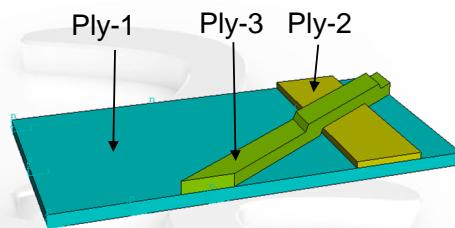
© DASSAULT SYSTEMES



L4.86

Abaqus/CAE Demonstration: Three-ply composite

- In this demonstration Abaqus/CAE is used to model the three-ply composite plate and postprocess the results.
 - Note: The narrated version of this demonstration can be accessed via the link to the Web based training provided in SIMULIA Answer 3417.



[Play Demo](#)

© DASSAULT SYSTEMES



Composites Modeler for Abaqus/CAE

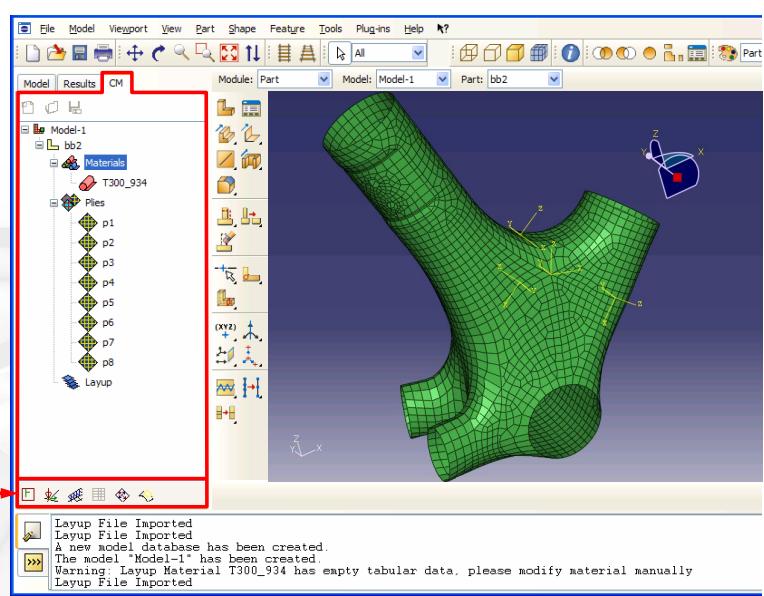
© DASSAULT SYSTEMES



L4.88

Composites Modeler for Abaqus/CAE

- Composites Modeler for Abaqus/CAE (CMA) is an add-on product that extends the Abaqus built-in ply modeling features by providing
 - Advanced fiber modeling (draping)
 - Import/export of .Layup files
 - Integration (bidirectional) between Abaqus and CATIA Composite Design (CPD)
 - Ply visualization tools



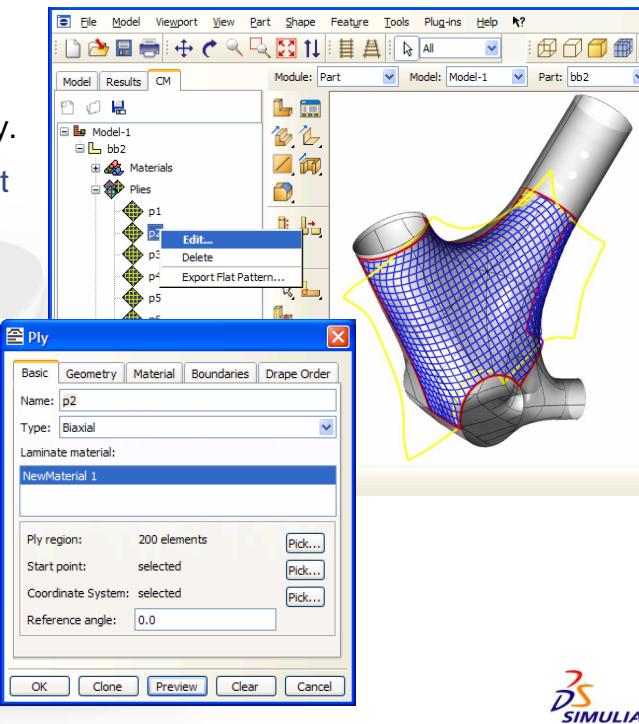
GUI interface: Composites Modeler of Abaqus/CAE

© DASSAULT SYSTEMES

Composites Modeler for Abaqus/CAE

- Draping calculation

- Accounts for local fiber directions when tape/cloth is draped over curved geometry.
- Projected CSYS may not account for the curved geometry correctly.
- Producibility (flat pattern prediction) to ensure that manufacturable plies are proposed.



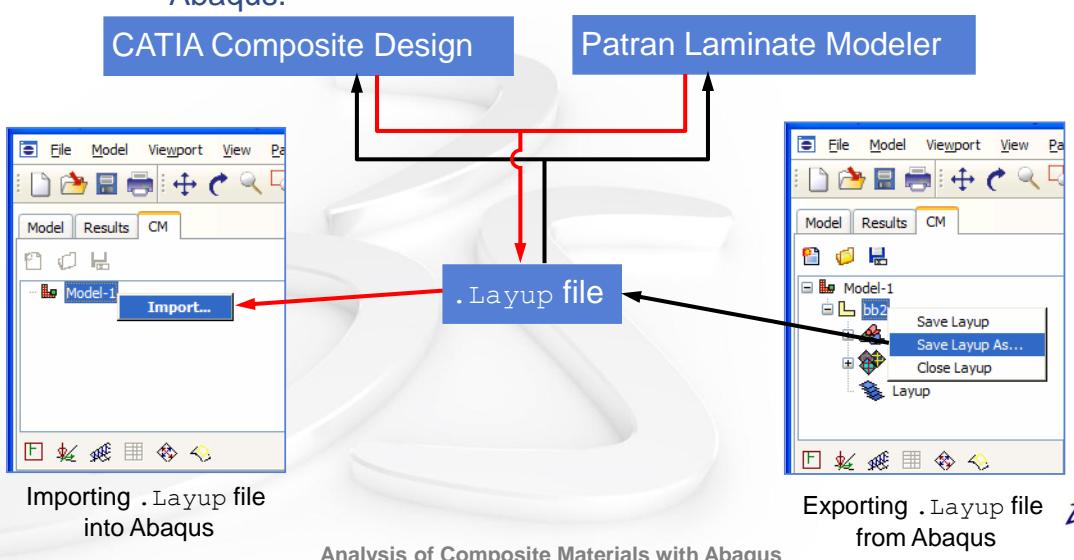
Analysis of Composite Materials with Abaqus

© DASSAULT SYSTEMES

Composites Modeler for Abaqus/CAE

- Import/export of .Layup files

- The .Layup file is supported by Patran Laminate Modeler, CATIA Composite Design (CPD),
- It is the neutral interchange format between these products and Abaqus.

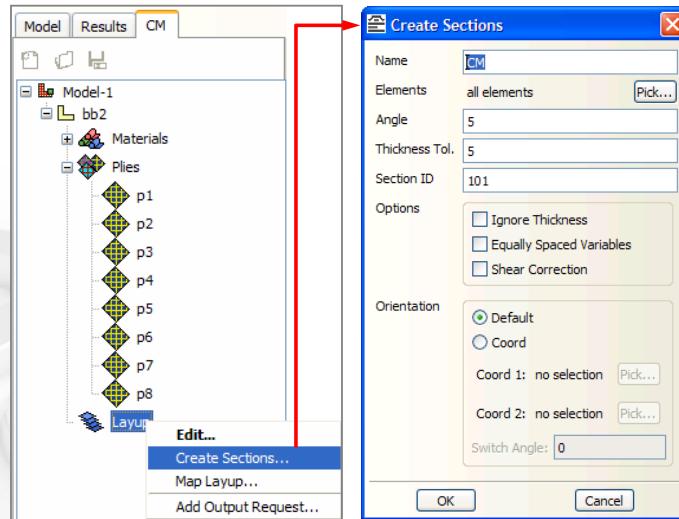


Analysis of Composite Materials with Abaqus

© DASSAULT SYSTEMES

Composites Modeler for Abaqus/CAE

- The layup created by CMA is incorporated into shell section definitions.
 - After creating sections from the layup, you can use the built-in ply stack plot tool to display the layup from a probed region/element.
- Ply-based postprocessing is also supported.
- To learn more about CMA, please consult the *Composites Modeler for Abaqus/CAE* lecture notes.



Creation of shell sections

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Notes

Notes

Alternative Modeling Techniques for Composites

Lecture 5

© DASSAULT SYSTEMES



L5.2

Overview

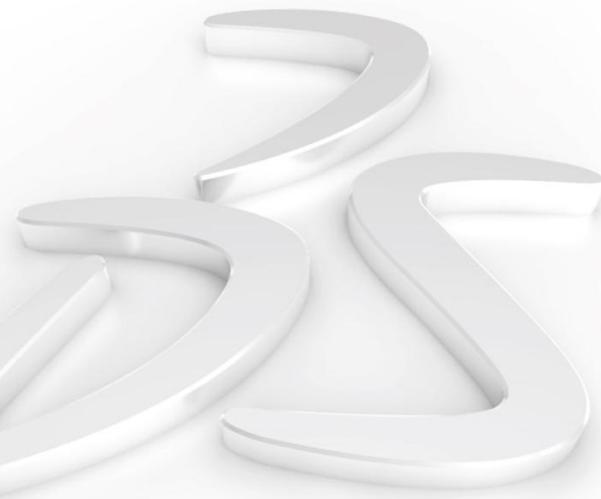
- **Introduction**
- **Laminated Shell Section Definition**
- **Laminated Solid Section Definition**
- **Section Point Based Postprocessing Technique**

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES



L5.4

Introduction

- **As alternative modeling techniques for composites,**
 - Shell sections can be used to define a laminated conventional shell made of one or more materials.
 - Shell sections can also be used to model **stacked** continuum shell elements.
 - Similarly, solid sections can be used to define a laminated solid and to model stacked solid elements.
 - Section point based postprocessing can be applied to view contour plots of integration point values, material orientations, and output from reinforcement (rebar) layers in the shell.
- **Consistent Abaqus/CAE and keywords interfaces are provided.**

© DASSAULT SYSTEMES



Laminated Shell Section Definition

© DASSAULT SYSTEMES



L5.6

Laminated Shell Section Definition

• Shell section definition

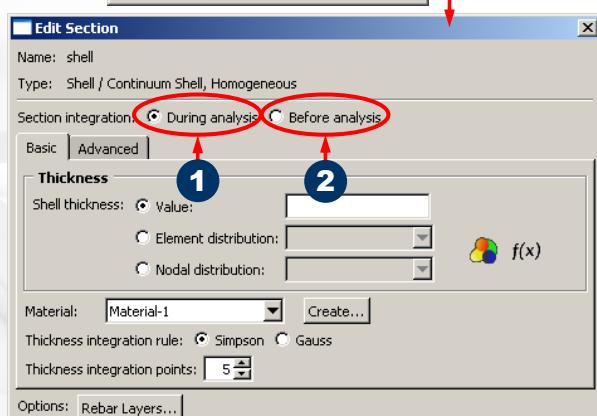
- A shell section can be defined with:

*SHELL SECTION

- ① employs numerical integration during the analysis for use with nonlinear materials.

*SHELL GENERAL SECTION

- ② pre-integrates the section for use with linear materials.



© DASSAULT SYSTEMES

Laminated Shell Section Definition

- Both section options support multilayer composite shells.

***SHELL SECTION, COMPOSITE**

***SHELL GENERAL SECTION, COMPOSITE**

- The section can have any number of layers with independent orientation and independent material in each layer.



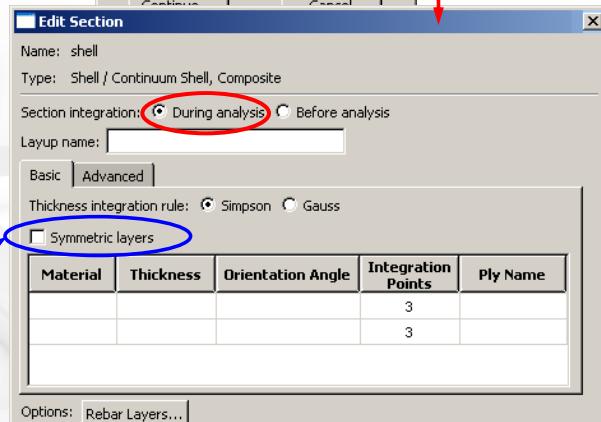
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Shell Section Definition

- The ***SHELL SECTION, COMPOSITE** option defines for each ply:
 - Section points
 - Material definition
 - Material orientation
 - Thickness
 - Ply name
- Simpson's rule is used for section stiffness integration.
- In general this option is used when nonlinear material response must be modeled.
- Include the SYMMETRIC parameter to activate simplified modeling of symmetric composites.



© DASSAULT SYSTEMES



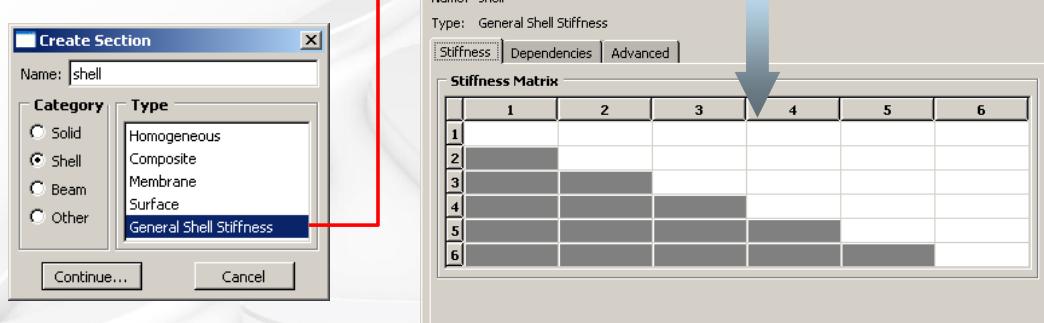
Analysis of Composite Materials with Abaqus

Laminated Shell Section Definition

- The *SHELL GENERAL SECTION option provides significant computational savings if the material behavior is linear.
 - In this case the section stiffness matrix can be given directly:

$$\begin{bmatrix} N_1 \\ N_2 \\ N_{12} \\ M_1 \\ M_2 \\ M_{12} \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} & B_{11} & B_{12} & B_{13} \\ & A_{22} & A_{23} & B_{12} & B_{22} & B_{23} \\ & & A_{33} & B_{13} & B_{23} & B_{33} \\ & & & D_{11} & D_{12} & D_{13} \\ & & & & D_{22} & D_{23} \\ & & & & & D_{33} \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \\ \kappa_1 \\ \kappa_2 \\ \kappa_{12} \end{bmatrix}$$

sym



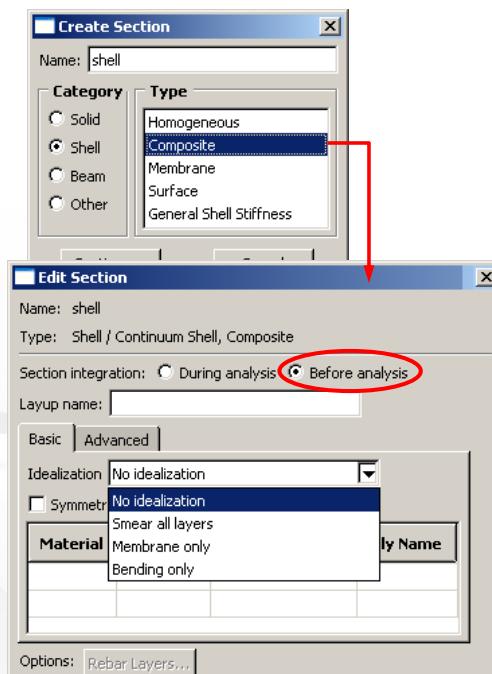
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Laminated Shell Section Definition

- Alternatively (and more commonly), the section is defined by giving the layers' thicknesses and material properties for layers.
 - Abaqus will then calculate the section's behavior automatically.
 - You can specify the idealization of the section response and the symmetric composites.
 - In this case (as with the *SHELL SECTION option) stress and strain output is available throughout the shell's thickness.
 - Usage:

*SHELL GENERAL SECTION, COMPOSITE, [SYMMETRIC],
[MEMBRANE ONLY|BENDING ONLY|SMEAR ALL LAYERS]



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

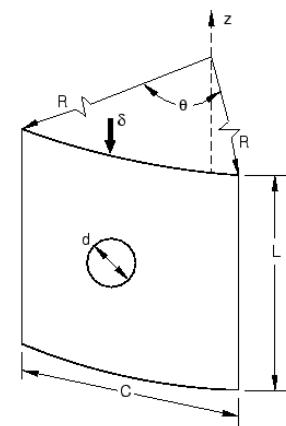
Laminated Shell Section Definition

- The shell section properties given through the *SHELL GENERAL SECTION option can be also defined via user subroutine **UGENS**.
 - This approach is particularly useful if the section response involves both geometric and material nonlinearity, such as may occur during section collapse.
 - Usage:

***SHELL GENERAL SECTION, USER**

Laminated Shell Section Definition

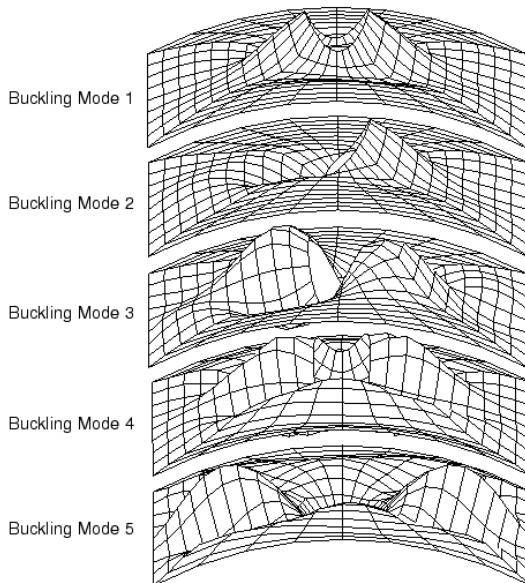
- Example of thin laminated shell analysis



- From *Laminated Composite Shells: Buckling of a Cylindrical Panel with a Circular Hole*, Abaqus Example Problems Manual, Section 1.2.2.

Laminated Shell Section Definition

- Results



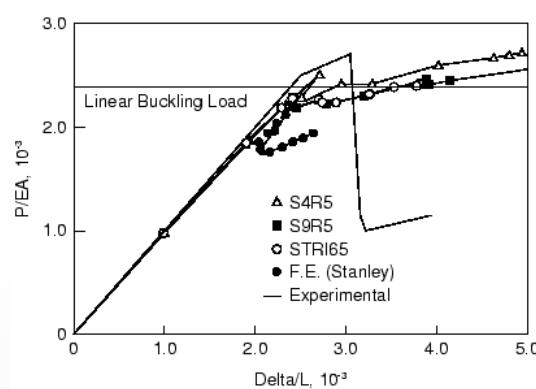
Buckling modes, element types S4R5, S9R5, and STR165

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Laminated Shell Section Definition



Composite panel: load-displacement response

- In this case delamination causes the shell to collapse suddenly (catastrophic failure).
 - The finite element models predict only the structural failure—there is no modeling of the delamination—and so fail to indicate the severity of the event.
 - Examining the values of TSHR13 and TSHR23 at the collapse load might alert the analyst to this possibility.

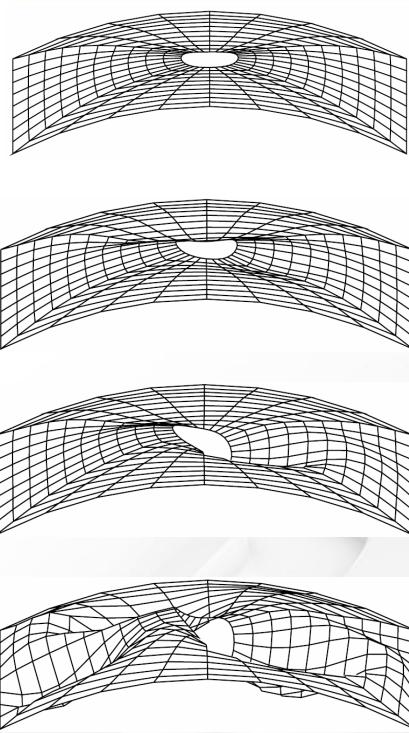
© DASSAULT SYSTEMES



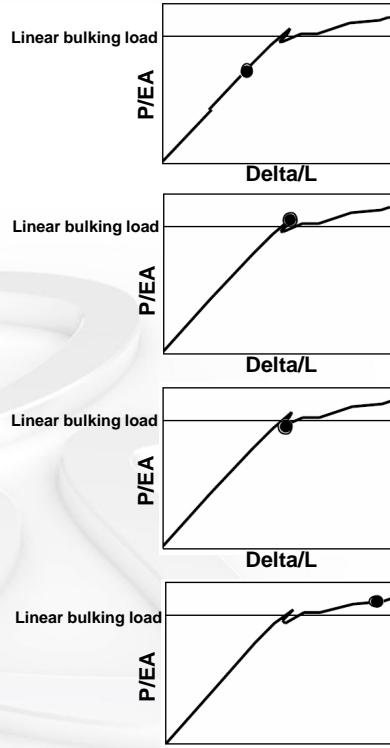
Analysis of Composite Materials with Abaqus

Laminated Shell Section Definition

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus



- Represents the current snapshot of the deformed shape as shown on the left side.

Postbuckling deformations: 10% h imperfection with S4R5

Note: 10% is a very large initial imperfection, and much smaller values are typically used. In this case 10% is specified for demonstration purposes.



Laminated Solid Section Definition

© DASSAULT SYSTEMES



Laminated Solid Section Definition

- Solid section definition

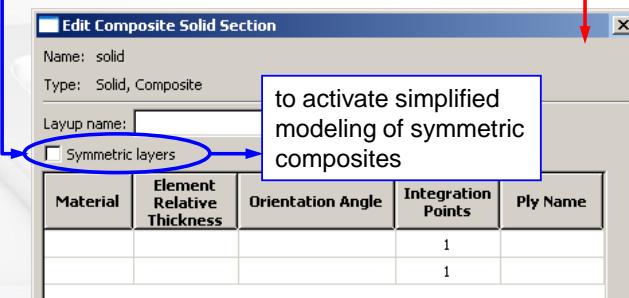
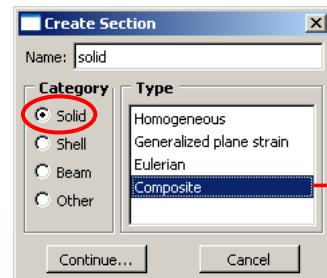
 - A solid section can be defined with:

***SOLID SECTION**

 - This section option supports multilayer composite solids:

***SOLID SECTION, COMPOSITE, [SYMMETRIC]**

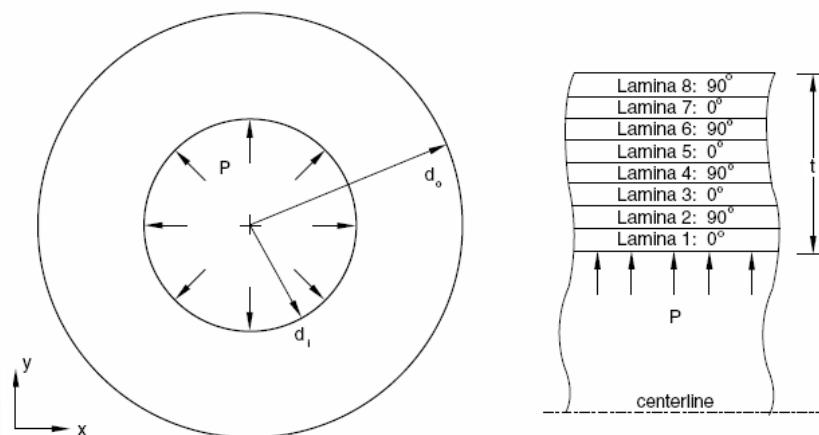
 - The section can have any number of layers with independent orientations and different materials in each layer.



Analysis of Composite Materials with Abaqus

Laminated Solid Section Definition

- Example of laminated solid analysis



Geometry of laminated cylinder

- From *Thick composite cylinder subjected to internal pressure*, Abaqus Benchmarks Manual, Section 1.1.4.



Analysis of Composite Materials with Abaqus

Laminated Solid Section Definition

- In this model the cylinder is discretized with two C3D20R elements (of different sizes) in the radial direction.

© DASSAULT SYSTEMES

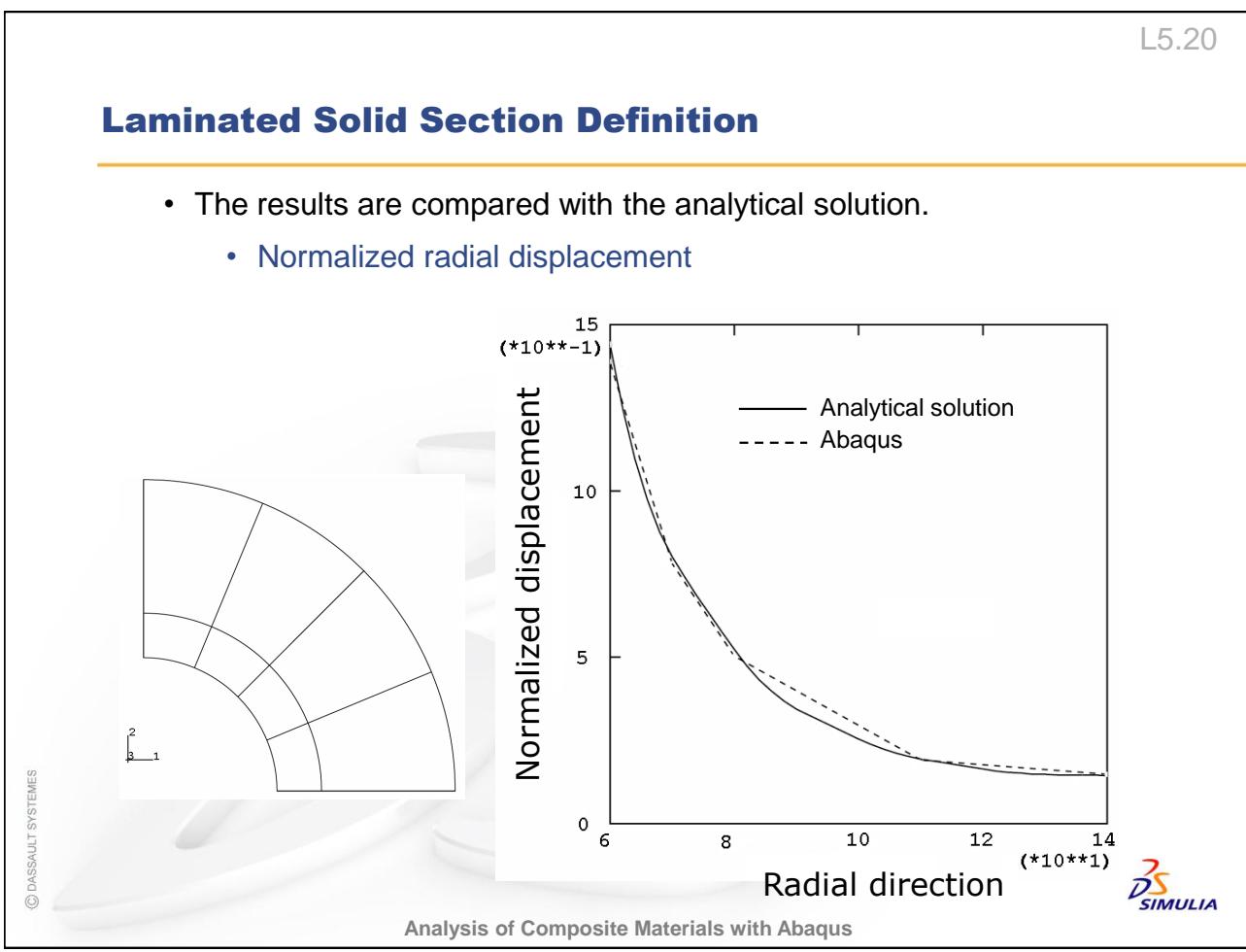
Edit Composite Solid Section				
Name: inner → assign to the set inner				
Type: Solid, Composite				
Layup name: inner layer				
<input type="checkbox"/> Symmetric layers				
Material	Element Relative Thickness	Orientation Angle	Integration Points	Ply Name
Lamina	1	0	3	Lamina_1
Lamina	1	90	3	Lamina_2

Edit Composite Solid Section				
Name: outer → assign to the set outer				
Type: Solid, Composite				
Layup name: outer layer				
<input type="checkbox"/> Symmetric layers				
Material	Element Relative Thickness	Orientation Angle	Integration Points	Ply Name
Lamina	1	0	3	Lamina_3
Lamina	1	90	3	Lamina_4
Lamina	1	0	3	Lamina_5
Lamina	1	90	3	Lamina_6
Lamina	1	0	3	Lamina_7
Lamina	1	90	3	Lamina_8

Analysis of Composite Materials with Abaqus

Laminated Solid Section Definition

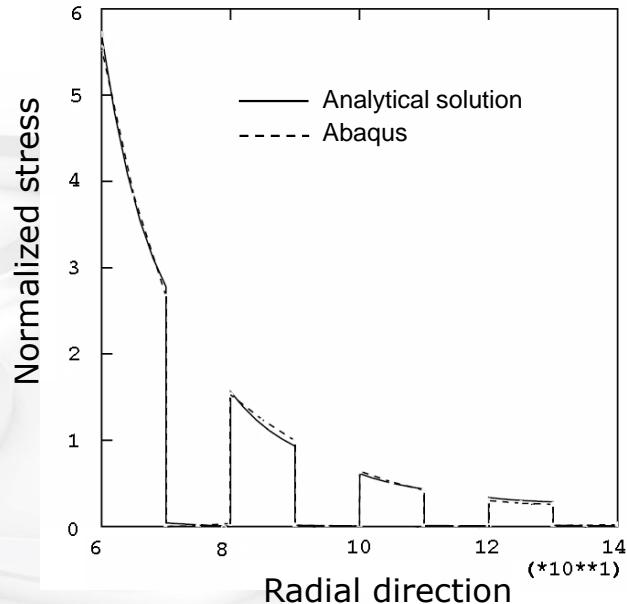
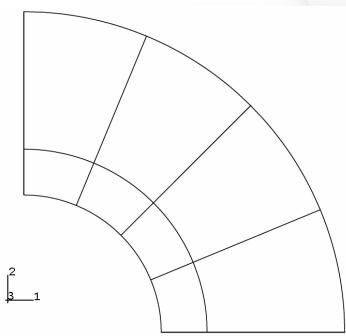
- The results are compared with the analytical solution.
 - Normalized radial displacement



Laminated Solid Section Definition

- Circumferential stress

© DASSAULT SYSTEMES



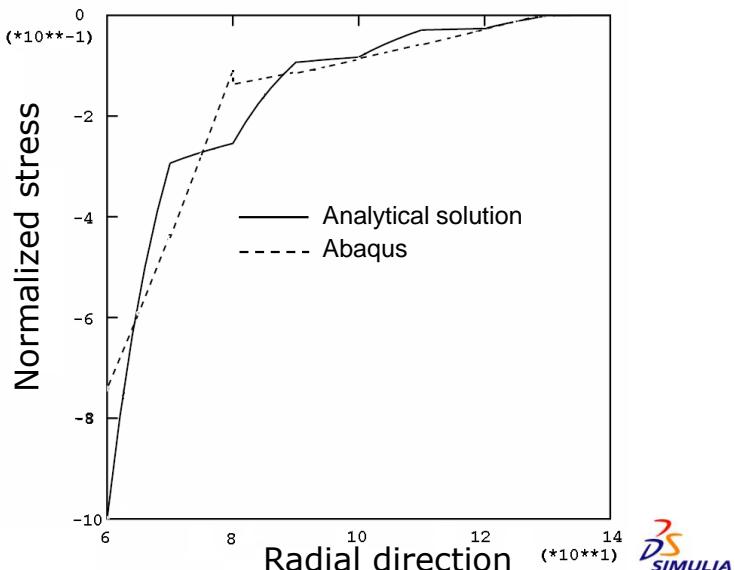
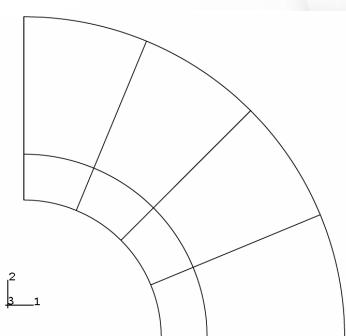
Analysis of Composite Materials with Abaqus

Laminated Solid Section Definition

- Radial stress

- The radial stress, especially near the inside of the cylinder, is not accurate due to the coarse mesh through the thickness. A refined mesh is required for an accurate result.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Section Point Based Postprocessing Technique

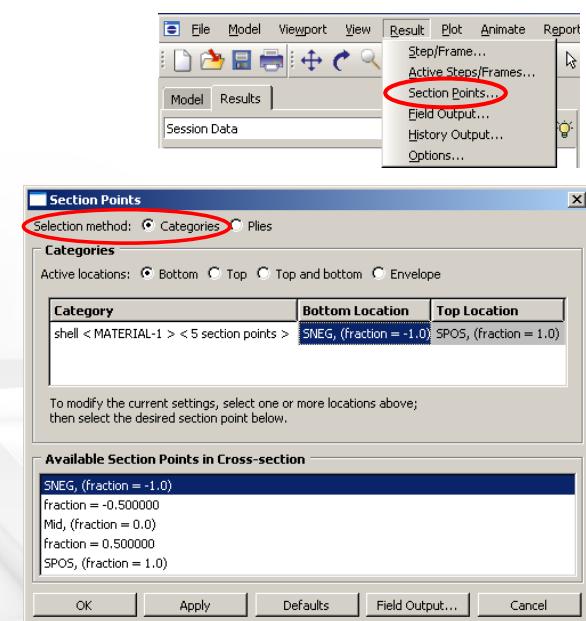
© DASSAULT SYSTEMES



L5.24

Section Point Based Postprocessing Technique

- As an alternative postprocessing technique, section point based postprocessing is available to visualize results.
 - Selecting section point data by category is used to control the section points from which Abaqus obtains integration point results and material orientations.
 - Reinforcement (rebar) layers in membrane, shell, composite solid, and surface elements are treated as section points for output purposes.
 - Note: Details of reinforcement (rebar) layers will be discussed in Lecture 6, "Reinforcement Modeling."



© DASSAULT SYSTEMES

Section Point Based Postprocessing Technique

- In many cases the location of a section point is described in terms of its position relative to the midpoint of the cross section.
 - For shells and composite solids this relative position is reported as a fraction of the distance between the midpoint of the cross section and the SPOS or SNEG surface of the section.
 - Reinforcement layers are indicated by their unique name.
- Contours of integration point values can be plotted as two-dimensional fronts located at the section points.
- Contours can be plotted for two section locations on the same shell (conventional or continuum) or composite solid.

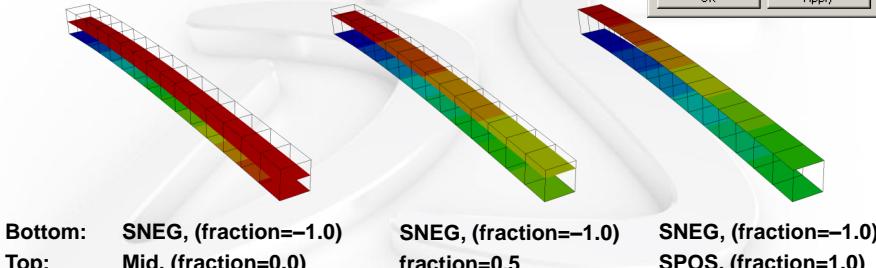
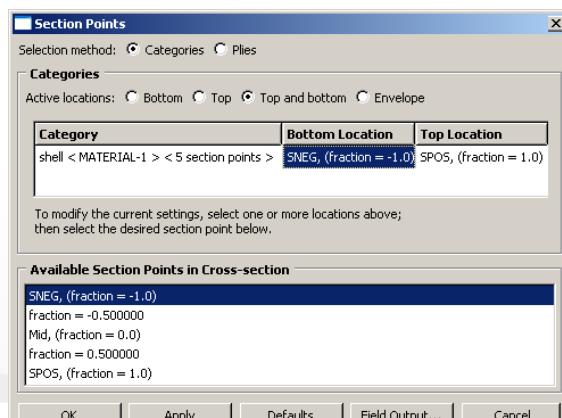
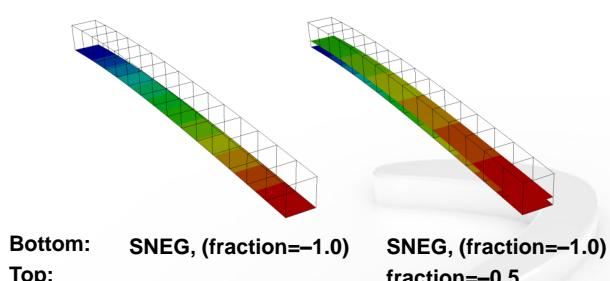
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Section Point Based Postprocessing Technique

- Example: Beam bending



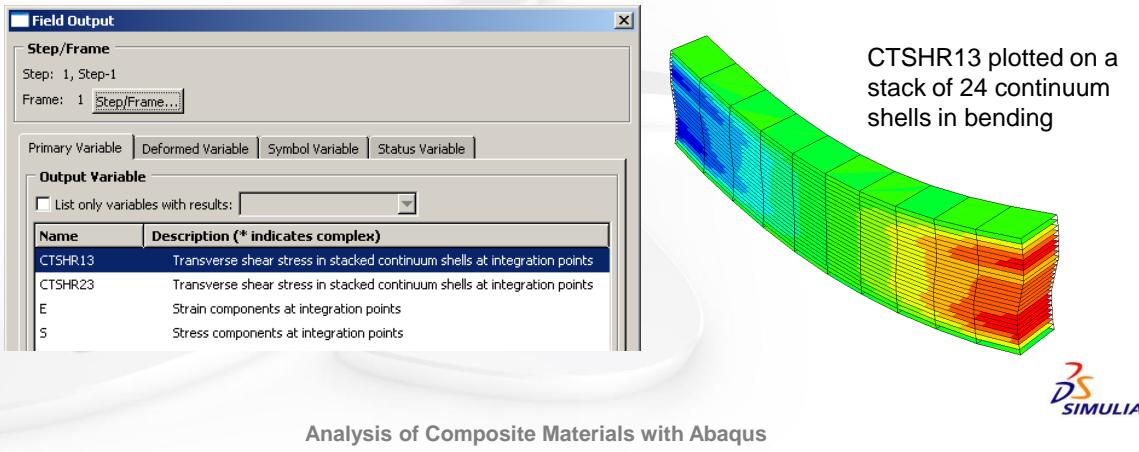
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Section Point Based Postprocessing Technique

- **Example: Contouring interlaminar transverse shear stress components**
 - Interlaminar transverse shear stress components can be a major cause of failure in layered composite plates.
 - Output variable CTSHR is an estimate of the through-the-thickness variation of transverse shear stress in continuum shells.
 - CTSHR enforces continuity of transverse shear between stacked continuum shell elements.



Notes

Notes

Reinforcement Modeling

Lecture 6

© DASSAULT SYSTEMES



L6.2

Overview

- Introduction
- Rebar Layers
- Embedded Elements

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES

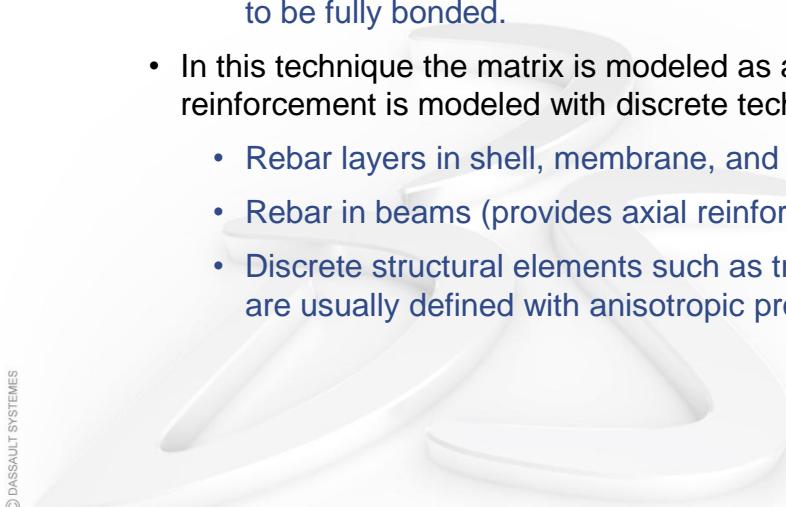


L6.4

Introduction

- **Discrete reinforcement**
 - Discrete analysis is used to model composites with large, distinct reinforcements (reinforced concrete, rubber tires with radial plies and belts, etc.).
 - The interface between the matrix and the reinforcement is assumed to be fully bonded.
 - In this technique the matrix is modeled as a continuum, but the reinforcement is modeled with discrete techniques such as:
 - Rebar layers in shell, membrane, and surface elements
 - Rebar in beams (provides axial reinforcement)
 - Discrete structural elements such as trusses or membranes, which are usually defined with anisotropic properties

© DASSAULT SYSTEMES



Introduction

- The material properties of the constituent materials can be elastic or inelastic.
- Limited inhomogeneity of the deformation is allowed, but separate stresses are always obtained in the matrix and the reinforcement.

© DASSAULT SYSTEMES



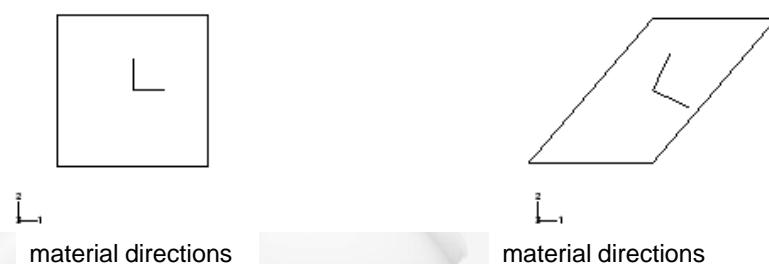
Analysis of Composite Materials with Abaqus

Introduction

• Structural elements vs. rebar layers

- Membranes with anisotropic properties can be used to model reinforcement. However, since material directions in the membrane element rotate with the average spin of the material, using anisotropic elasticity to model a material that is not truly a continuum can give rise to significant errors when large deformations occur.
 - For example, consider the rotation of material directions under finite shear:

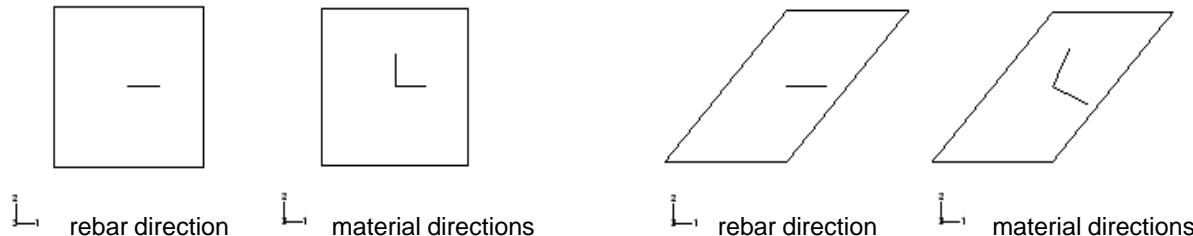
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Introduction

- Rebar directions rotate with the actual deformation and hence will provide more accurate results.
 - For example an individual fiber in a reinforcing belt of a tire can shear relatively easily compared to fibers in other directions.



Rebar direction vs. material directions under finite shear

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Introduction

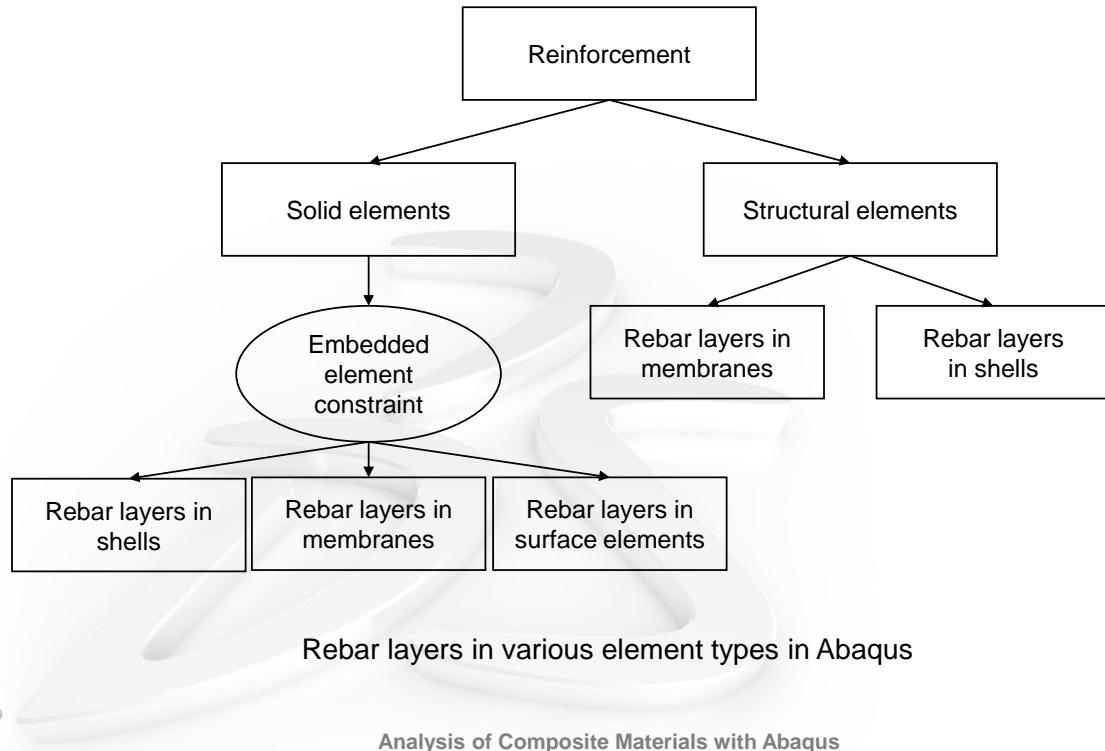
- **Reinforcement in Abaqus**
 - Rebar layers in Abaqus can be used to reinforce a matrix of either solid or structural elements such as shells and membranes; the overall organization of the use of rebar layers in Abaqus is illustrated in the following figure.
 - Shell, membrane, and surface elements are reinforced by directly specifying a rebar layer in the element.
 - Surface elements do not have any element properties other than the rebar layer and are used primarily as place-holders for rebar layers.
 - Solid elements are reinforced using the embedded element constraint.
 - In this technique, either shell, membrane, or surface elements reinforced with rebar layers are embedded in the host matrix comprised of solid elements.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Introduction



Rebar Layers

Rebar Layers

- Rebar layers are used for modeling layers of uniaxial reinforcement in shell, membrane and surface elements.
 - Frequently, rebar layers have high material stiffness compared to the matrix material so that elastic properties often are enough to characterize the rebar material behavior (except for yield and limit load calculations).
- Rebar layers have the following properties:
 - Rebar layers can be superposed on shells, membranes or surface elements.
 - They use the shape functions and integration schemes of the underlying elements, which implies that relative slip between the rebar and the matrix material (“dowel action”) is not considered.

Rebar Layers

- Their material properties are independent of those of the underlying elements.
- A rebar layer is smeared into an equivalent layer of constant thickness.
- As many different combinations and orientations of rebar layers as are needed can be defined within a single element.
- The rebar layer volume is not subtracted from the volume of the element to which the rebar layer is added.
 - Thus, rebar layers should be used only when the volume fraction of reinforcement is small (such as with reinforced concrete where the volume fraction of the rebar is between 1% and 4%).
- The mass of the rebar layer is taken into account for dynamic analysis as well as for distributed loads of type GRAV, CENTRIF, and ROTA.
- Rebar layers and associated results can be transferred between Abaqus/Standard and Abaqus/Explicit using the *IMPORT option.

Rebar Layers

- Specification of rebar layers

- The *REBAR LAYER option is used in conjunction with the

*SHELL SECTION,
 *MEMBRANE SECTION, or
 *SURFACE SECTION



options to specify reinforcement layers in shell, membranes, and surface elements, respectively.

© DASSAULT SYSTEMES

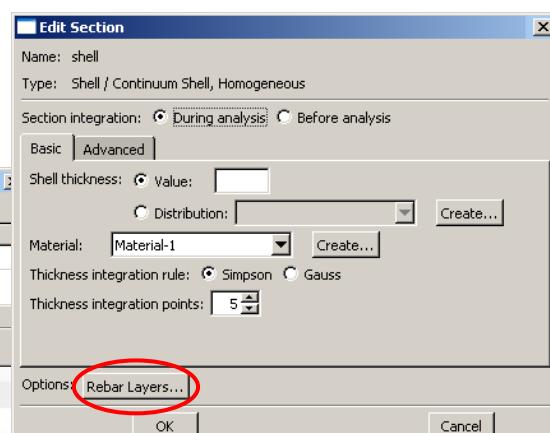
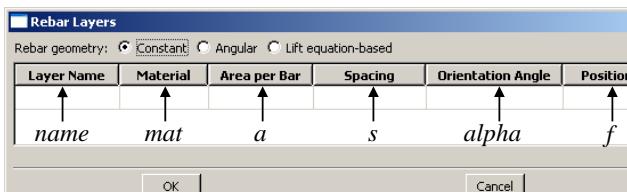


Analysis of Composite Materials with Abaqus

Rebar Layers

- Sample usage:

```
*SHELL SECTION, ELSET=...
*REBAR LAYER, ORIENTATION=ORI1
    name, a, s, f, mat, alpha
```



the Assign Rebar Reference Orientation tool



```
*SHELL SECTION, ELSET=...
*REBAR LAYER, ORIENTATION=ORI1
    name, a, s, f, mat, alpha
```

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Rebar Layers

- On the data line specify the
 - rebar layer name (used to identify the layer in the list of section points when postprocessing with Abaqus/Viewer);
 - cross-sectional area a of each rebar;
 - the rebar spacing s in the plane of the membrane, shell, or surface element;
 - the position of the rebars in the thickness direction ξ (for shell elements only), measured from the midsurface of the shell (positive in the direction of the positive normal to the shell);
 - the rebar material name; and
 - the angular orientation alpha, in degrees, measured relative to the local 1-direction, positive in the direction of the element normal.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Rebar Layers

- Repeat the data line to define each rebar layer. Also, note the following:
 - The local directions in the rebar layer are determined by the orientation specified on the keyword option
 - Any orientation defined on the membrane or shell element has no influence on the local direction in the rebar layer

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Rebar Layers

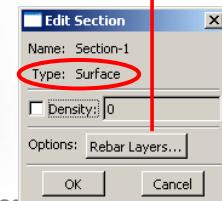
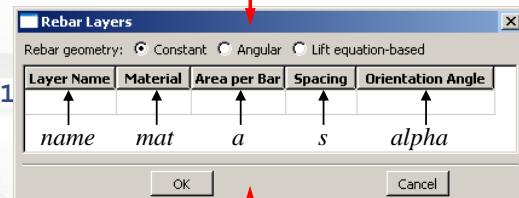
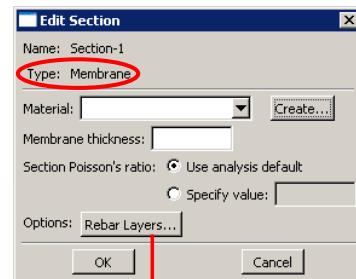
- Similar syntax is used to define rebar in membrane and surface elements.

```
*MEMBRANE SECTION, ELSET=...
*REBAR LAYER, ORIENTATION=ORI1
name, a, s, , mat, alpha, 1
```

and

```
*SURFACE SECTION, ELSET=...
*REBAR LAYER, ORIENTATION=ORI1
name, a, s, , mat, alpha, 1
```

Position meaningless for
membrane and surface sections;
leave blank.



Analysis of Composite Materials with Abaqus

Rebar Layers

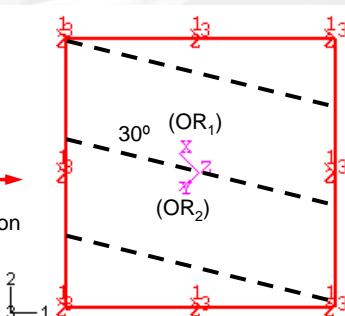
- Example:

```
*SURFACE SECTION, ELSET=BELT
*REBAR LAYER, ORIENTATION=ORIENT
RbName, 0.01, 0.1, , RbMat, 30., 1
*ORIENTATION, SYSTEM=RECTANGULAR, NAME=ORIENT
-0.7071, 0.7071, 0.0, -0.7071, -0.7071, 0.0
3, 0.0
```

Layer Name	Material	Area per Bar	Spacing	Orientation Angle
RbName	RbMat	0.01	0.1	30



Assign the rebar
reference orientation



OR_n = *ORIENTATION-defined directions
1, 2 = default local directions



Analysis of Composite Materials with Abaqus

Rebar Layers

- **Prestresses in rebar layers**

- Prestress can be defined in the rebars using the *INITIAL CONDITIONS, TYPE=STRESS, REBAR option, with or without using the *PRESTRESS HOLD option.
 - With the *PRESTRESS HOLD option the initial stress defined in the rebar is held constant.
 - While equilibrium iterations are performed to obtain the corresponding (self-equilibrating) stresses in the matrix material, the rebar layer will strain, but this strain is not allowed to cause changes in the stress in the rebar layer.
 - Without the *PRESTRESS HOLD option, the initial stresses are allowed to change during an equilibrating static analysis step as both the matrix and the rebar stresses adjust to the equilibrium configuration.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Rebar Layers

- An example of this is reinforced concrete:
 - The rebar are initially stretched to a desired tension before being covered by concrete.
 - After the concrete cures and bonds to the rebar, release of the initial rebar tension transfers load to the concrete, introducing compressive stresses in the concrete.
 - The resulting deformation in the concrete reduces the stress in the rebar.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Rebar Layers

- **Output**

- Abaqus/Viewer supports visualization of rebar layer orientations and results in rebar layers.
 - Both field and history output are supported.
- Output of variables such as stresses and strains at the rebar integration points is available; use the REBAR parameter on the *ELEMENT OUTPUT option to request such output for rebar layers.
 - Results can be viewed on a layer-by-layer basis.
 - To display results for a given rebar layer, select the named rebar layer from the list of available section points.

© DASSAULT SYSTEMES

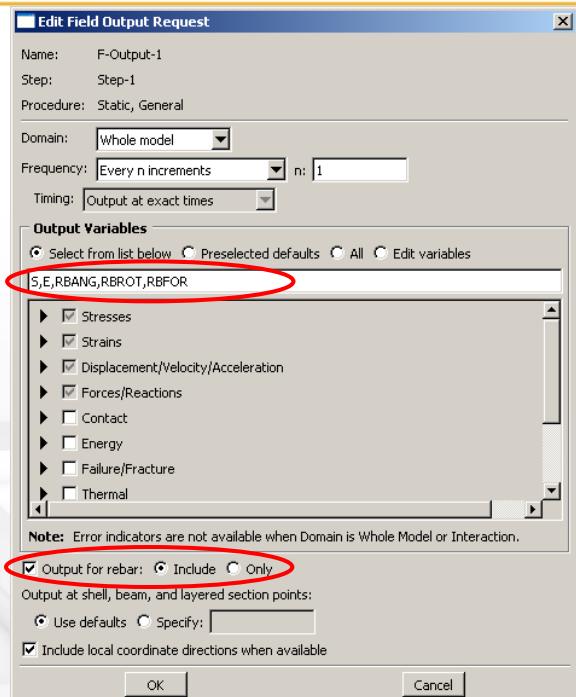


Analysis of Composite Materials with Abaqus

Rebar Layers

- The force in the rebar is available at the layer integration points as RBFOR, which is the rebar stress times the current cross-sectional area (see the figure on the following page).
- RBANG and RBROT identify the current orientation of rebar within the element and the relative rotation of the rebar layer as a result of finite deformation.
- Sample output request:

```
*ELEMENT OUTPUT, REBAR
S, E, RBANG, RBROT, RBFOR
```



© DASSAULT SYSTEMES



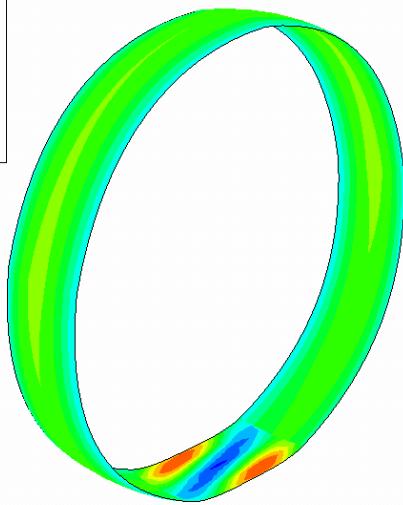
Analysis of Composite Materials with Abaqus

Rebar Layers

- Rebar force output

RBFOR
Multiple section points
(Ave. Crit.: 75%)

+	6.695e+01
+	5.905e+01
+	5.208e+01
+	4.509e+01
+	3.811e+01
+	3.113e+01
+	2.415e+01
+	1.717e+01
-	1.019e+01
-	3.017e+00
-	-1.080e+01
-	-1.779e+01



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Embedded Elements

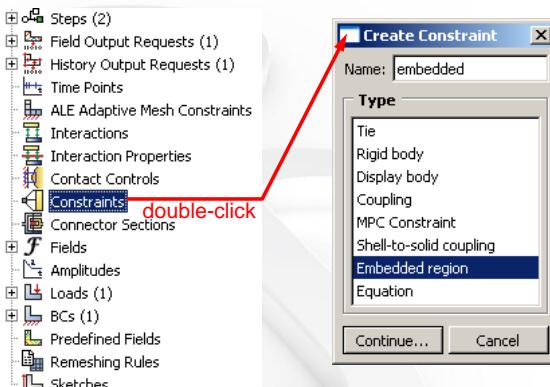
© DASSAULT SYSTEMES



Embedded Elements

- Shell, membrane, and surface elements reinforced with rebar layers can be embedded in continuum (solid) elements in an arbitrary manner such that the two meshes need not match.
 - This is accomplished using the *EMBEDDED ELEMENT option.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Embedded Elements

- Abaqus will search for the geometric relationships between nodes of the embedded elements and the host elements.
- If a node of an embedded element lies within a host element, the degrees of freedom at the node will be eliminated, and the node becomes an “embedded node.”
- The degrees of freedom of the embedded node are constrained to the interpolated values of the degrees of freedom of the host element.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Embedded Elements

- This enables a more natural geometric specification of rebar in structures.
 - It also enables visualization of rebar layers in solid elements.
- The embedded element constraint can also be used to model a set of truss or beam elements that lie embedded in a set of solid elements or a set of solid elements that lie embedded in another set of solid elements.
 - The constraint will not constrain rotational degrees of freedom of the embedded nodes when shell or beam elements are embedded in solid elements.

© DASSAULT SYSTEMES



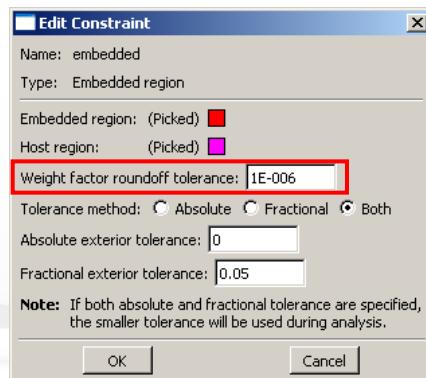
Analysis of Composite Materials with Abaqus

Embedded Elements

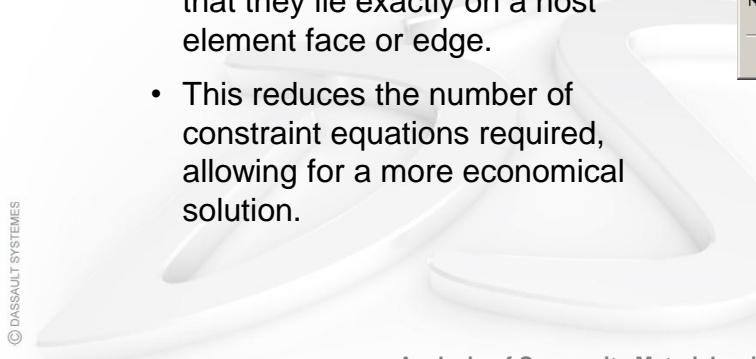
- Usage:

```
*EMBEDDED ELEMENT, HOST
ELSET=tread,
  ROUND OFF TOLERANCE=1.e-6
  belt1, belt2
```

- The ROUND OFF TOLERANCE parameter is used to adjust the position of embedded nodes such that they lie exactly on a host element face or edge.
- This reduces the number of constraint equations required, allowing for a more economical solution.

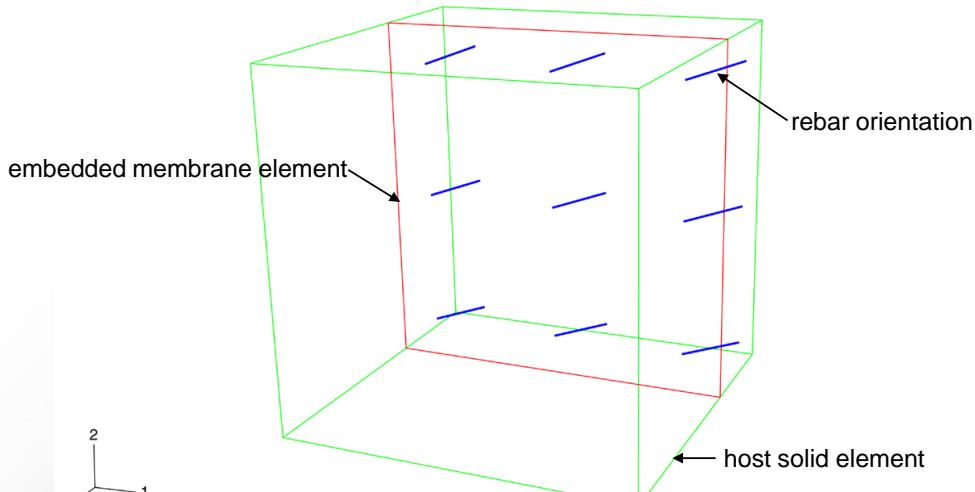


© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Embedded Elements



Reinforced solid element using the embedded element technique

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Embedded Elements

- **Choosing between membrane and surface elements**
 - The following guidelines are provided to help you choose between rebar-reinforced membrane elements and rebar-reinforced surface elements.
 - Use surface elements to model uniaxial behavior in the reinforcement layers of the solid elements.
 - Use membrane elements to introduce in-plane shear behavior in the reinforcement layers of the solid elements.
 - Note that membrane elements are more expensive than surface elements due to the reference surface section calculations.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Notes

Notes

Analysis of Sandwich Composites

Lecture 7

© DASSAULT SYSTEMES



L7.2

Overview

- **Introduction to Sandwich Composites**
- **Abaqus Usage**
- **Abaqus Examples**
 - Comparison to NAFEMS solution
 - Comparison of Conventional and Continuum Shells
 - Stacking Elements Through the Thickness
 - Tapered Sandwich Composite

© DASSAULT SYSTEMES



Introduction to Sandwich Composites

© DASSAULT SYSTEMES



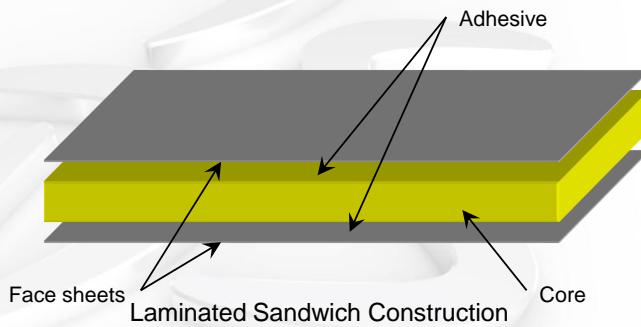
L7.4

Introduction to Sandwich Composites

- A laminated sandwich composite is a structure usually consisting of three features:
 - Two laminated faces sheets
 - Thick core material
- The design principle is the same as that of an I-beam



University of Maine's Human Power Submarine (Glass/Epoxy and foam core)

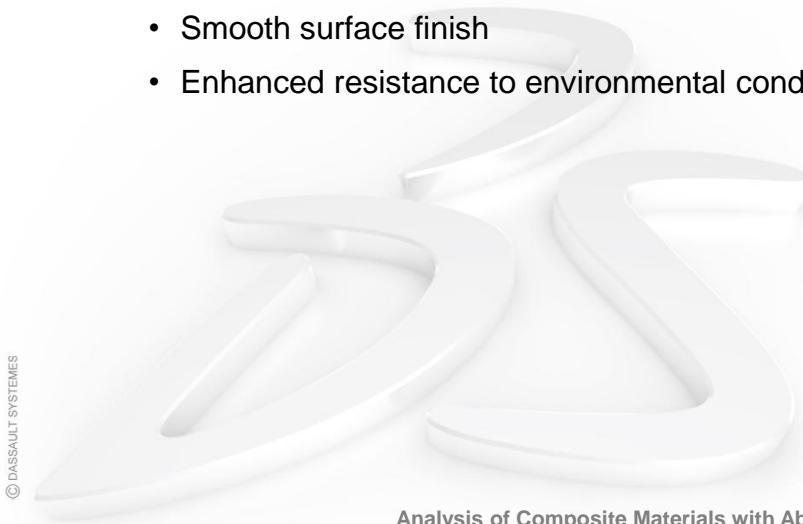


© DASSAULT SYSTEMES

Introduction to Sandwich Composites

- The face sheets act together to resist global bending moments and provide high in-plane longitudinal stiffness
- Primary features of the face sheets:
 - High tensile and compressive strength (i.e., high in-plane stiffness)
 - Impact resistance
 - Smooth surface finish
 - Enhanced resistance to environmental conditions and wear

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Introduction to Sandwich Composites

- The core acts to stabilize the face sheets over their entire surface
- Primary features of the core:
 - Low density
 - Shear modulus (transverse shear stiffness is VERY important)
 - Shear strength
 - Stiffness in transverse direction (perpendicular to face sheets)
 - Thermal and acoustic insulation
 - Easily mass produced
- Typical core materials used are:
 - Honeycombs
 - Foams
 - Balsa wood
- Core can have isotropic or anisotropic behavior

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Usage

© DASSAULT SYSTEMES



L7.8

Abaqus Usage

- Typically, the inclusion of the core requires that transverse shear deformation of the structure be considered to obtain an accurate solution (even if the length-to-thickness ratio is larger than 20)
- In Abaqus, this means “thick” shells are appropriate for modeling sandwich composites
 - E.g., S4R, S3R, S8R, SAX1, SAX2, SC8R, and SC6R elements
- Solid 3D continuum elements can also be utilized, but many elements may be needed through the thickness to accurately capture the shear deformation

© DASSAULT SYSTEMES



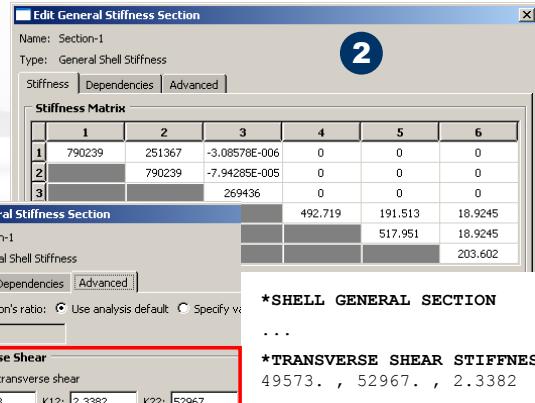
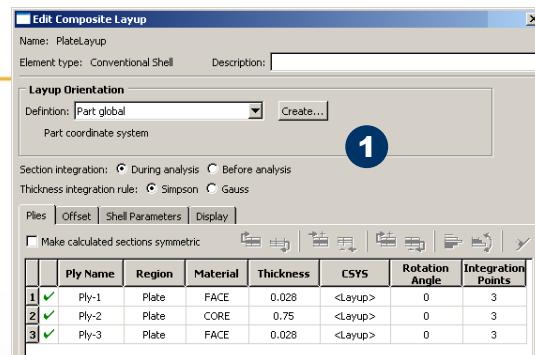
Abaqus Usage

- Conventional shell elements:

- The composite layup can be included utilizing one of the methodologies described in the lectures on composite modeling techniques.

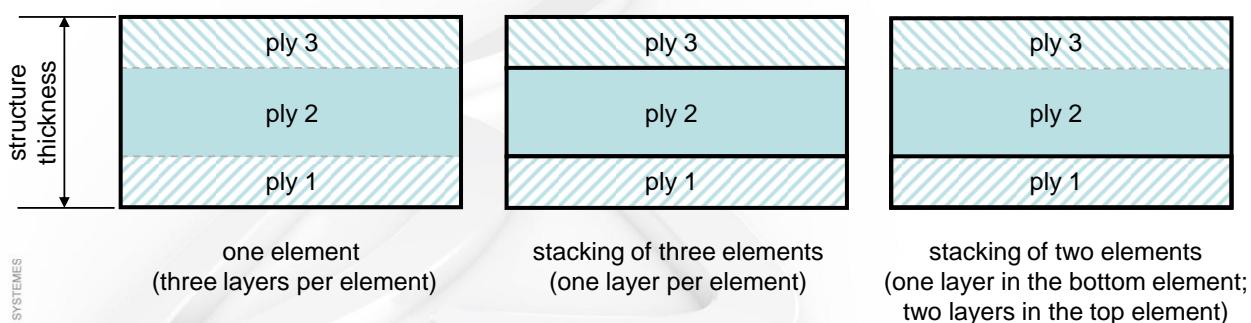
- For example:

- 1 Each ply of the layup is defined, and Abaqus calculates the section stiffness.
- 2 The section stiffness properties are assumed to be linear, and the ABD matrix is entered directly (the transverse shear stiffness must be included utilizing *TRANSVERSE SHEAR STIFFNESS).



Abaqus Usage

- Utilization of elements where “stacking” is involved (e.g., stacking of continuum shells or solids through the structure thickness):
 - Care must be taken that the correct layer of the composite is associated with the element;
 - This means that the element may consist of many layers;
 - Otherwise, the section definition is the same as that for conventional shell elements.



Abaqus Examples

© DASSAULT SYSTEMES



L7.12

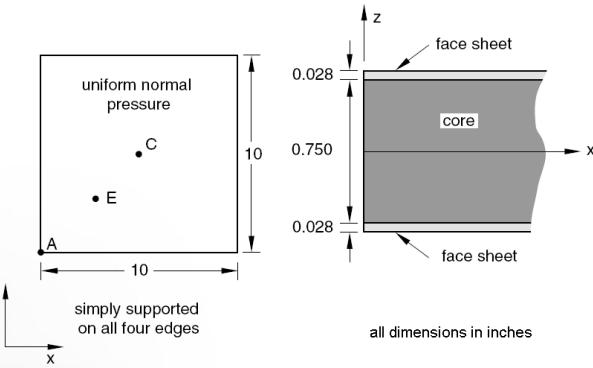
Abaqus Examples

1 Comparison to NAFEMS solution

- A quarter symmetry model of a simply-supported sandwich composite plate
- The plate is loaded with a uniform normal pressure load
- There are three distinct layers:
 - Two stiff face sheets
 - One soft core
- The material properties are as follows:

Face sheets:

$$E_1 = 1.0 \times 10^7 \text{ psi}, \\ E_2 = 4.0 \times 10^6 \text{ psi}, \\ v_{12} = 0.3, \text{ and} \\ G_{12} = G_{13} = G_{23} = 1.875 \times 10^6 \text{ psi}$$



Core:

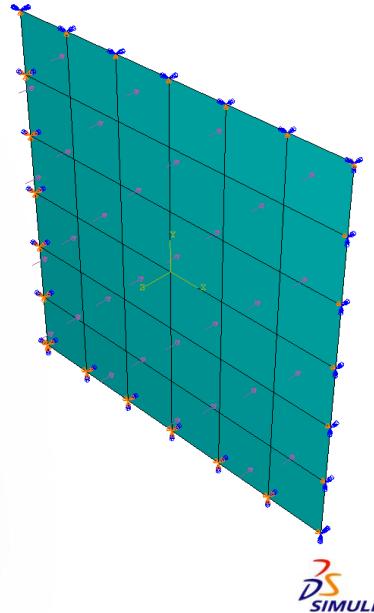
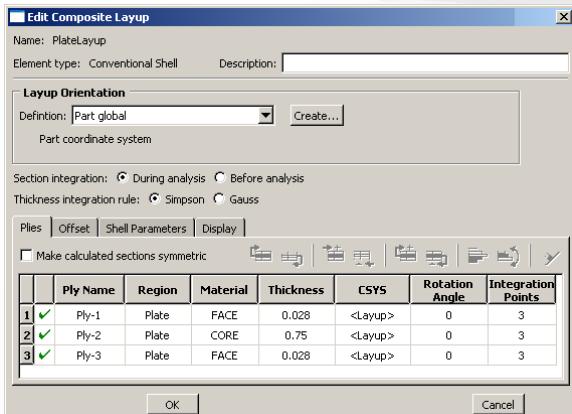
$$E_1 = E_2 = 10.0 \text{ psi}, \\ v_{12} = 0., \\ G_{12} = 10 \text{ psi}, \\ G_{13} = 3.04 \times 10^4 \text{ psi, and} \\ G_{23} = 1.2 \times 10^4 \text{ psi}$$

© DASSAULT SYSTEMES



Abaqus Examples

- The laminate is model utilizing S4R elements (with enhanced hourglass control)
 - The model consists of:
 - 36 S4R elements
 - 49 nodes

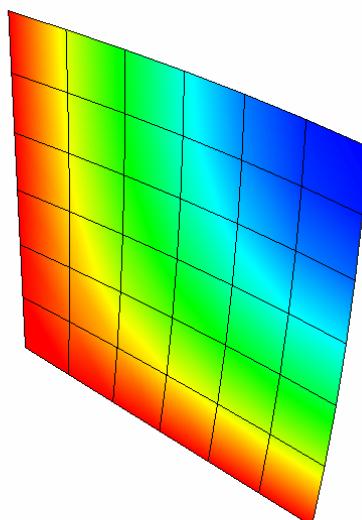
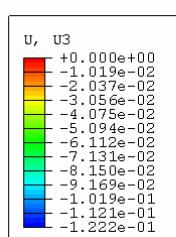


© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Abaqus Examples

- Transverse displacement contour plot



Step: LinearStatic
 Increment: 1. Step Time = 1.000
 Primary Var: U, U3
 Deformed Var: U Deformation Scale Factor: +4.090e+00

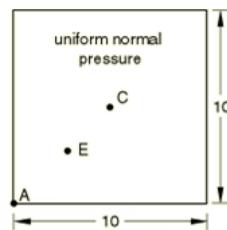
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Abaqus Examples

- Comparison of results:



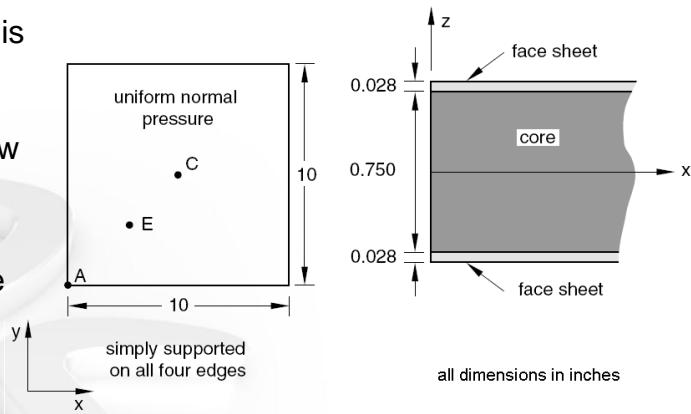
Model	u_z at C	S_{11} at C	S_{22} at C	S_{12} at E
NAFEMS*	-0.123	34449	13932	-5068
S4R	-0.122	35142.1	13696.8	-5210.29
% Diff	0.813 %	2.012 %	1.688 %	2.808 %

*This is a test recommended by the National Agency for Finite Element Methods and Standards (U.K.): Test R0031/3 from NAFEMS publication R0031, "Composites Benchmarks," February 1995.

Abaqus Examples

2 Comparison of Conventional and Continuum Shells

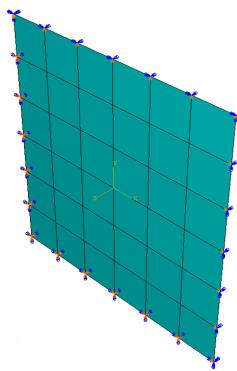
- The physical configuration is exactly the same as the previous example, with the exception that the plate now has fixed boundary conditions
- The material properties are exactly the same
- The plate is modeled with both S4R (conventional) and SC8R (continuum) shell elements to see how the results compare



Abaqus Examples

- The conventional shell model:
 - 36 S4R elements
 - 49 nodes
- The continuum shell model:
 - 36 SC8R elements
 - 98 nodes

© DASSAULT SYSTEMES



Note: only one element through the thickness

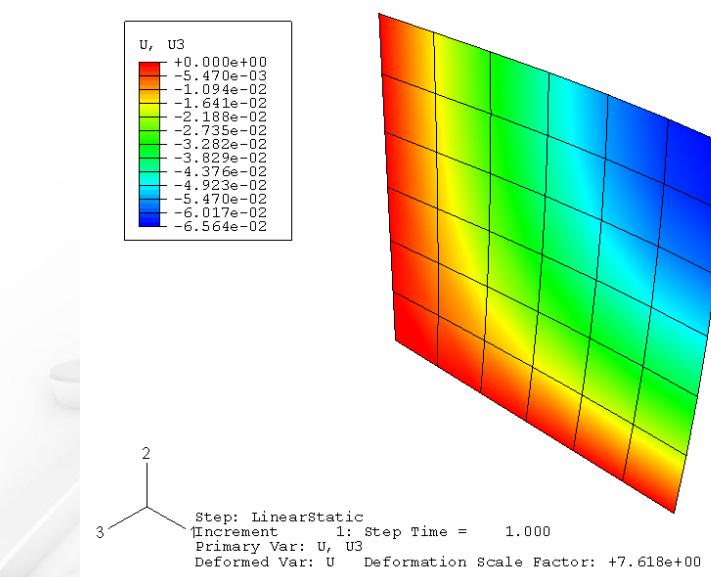


Analysis of Composite Materials with Abaqus

Abaqus Examples

- Transverse displacement contour plot of S4R elements

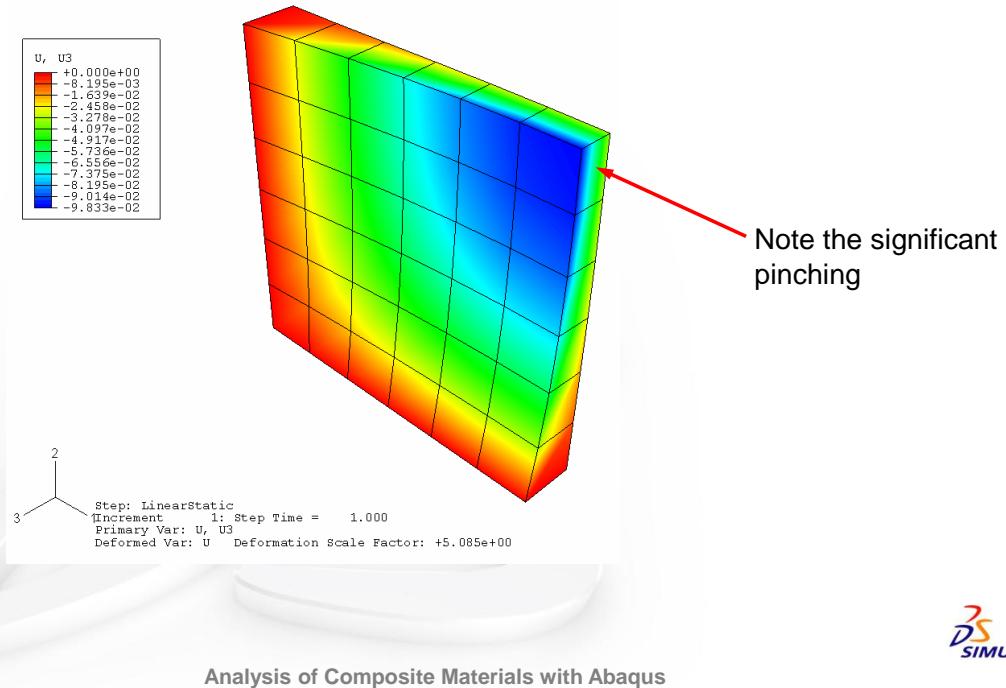
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

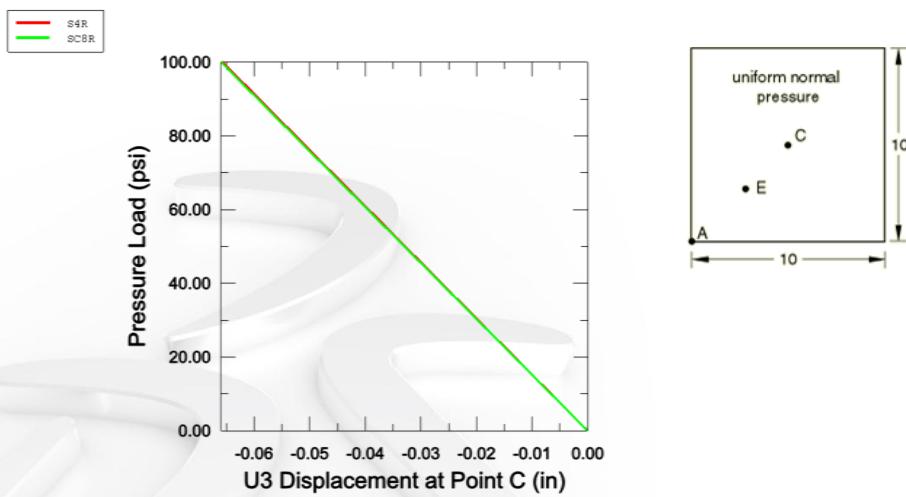
Abaqus Examples

- Transverse displacement contour plot of SC8R elements



Abaqus Examples

- Plots of the displacement at point C versus pressure load

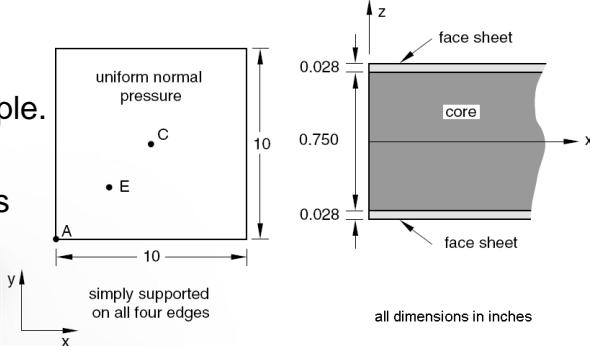


The displacement on the top and bottom of the SC8R element is averaged to produce the plot

Abaqus Examples

3 Stacking Elements Through the Thickness

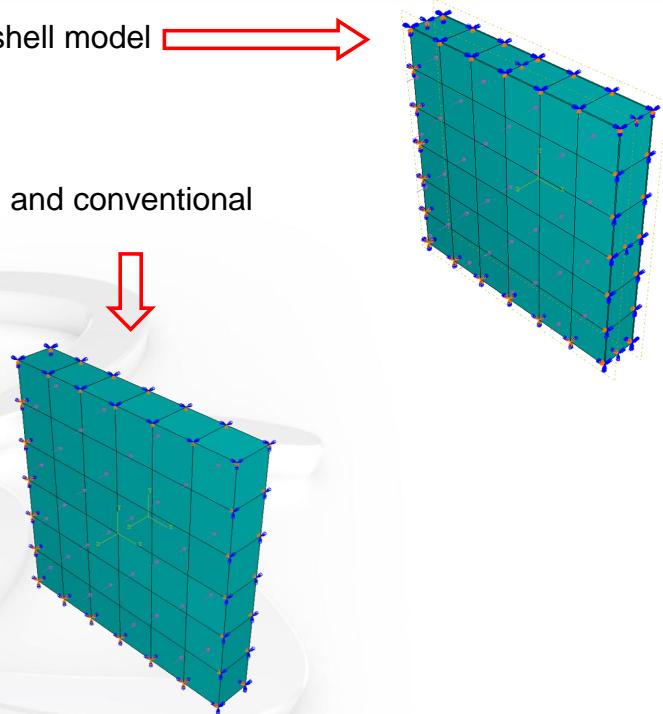
- Again, the same physical configuration as the previous example is utilized for this example.
- In this case, multiple elements are utilized through the thickness of the laminate.
- The two separate configurations are considered:
 - Three continuum shell (SC8R) elements through the thickness (one for each of the distinct layers)
 - One continuum shell element to model the core, with conventional shell elements to model the face sheets



all dimensions in inches

Abaqus Examples

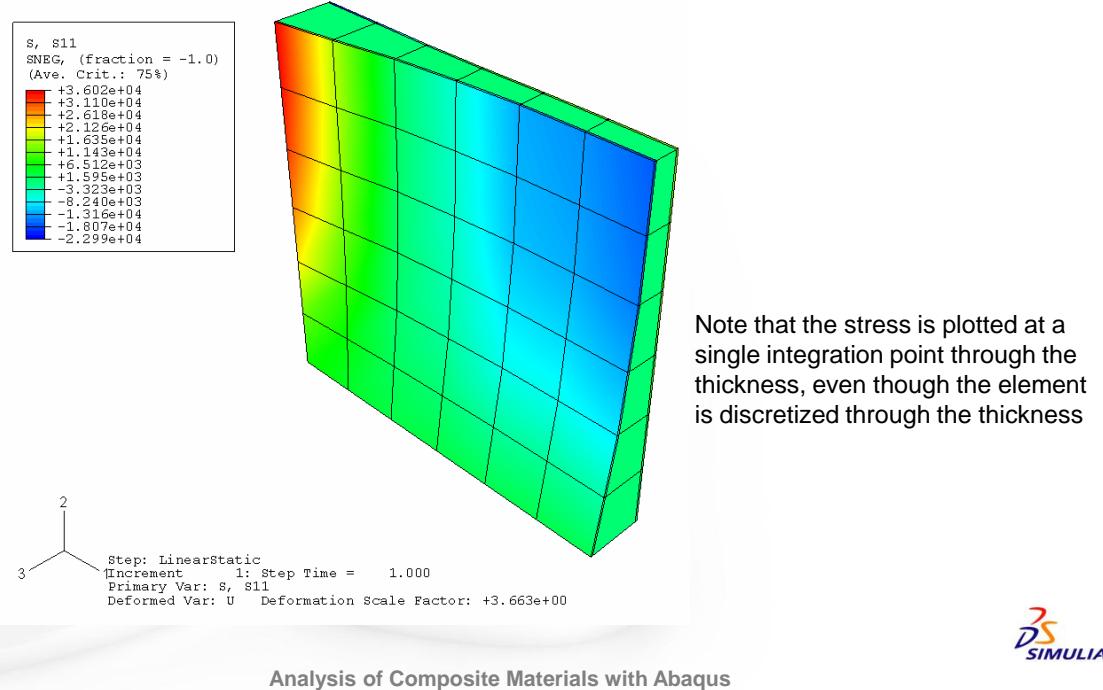
- The pure continuum shell model
- 108 elements
- 196 nodes
- The mixed continuum and conventional shell model
- 108 elements
- 98 nodes



Abaqus Examples

- The stress in the 1-direction for the pure continuum shell model

© DASSAULT SYSTEMES

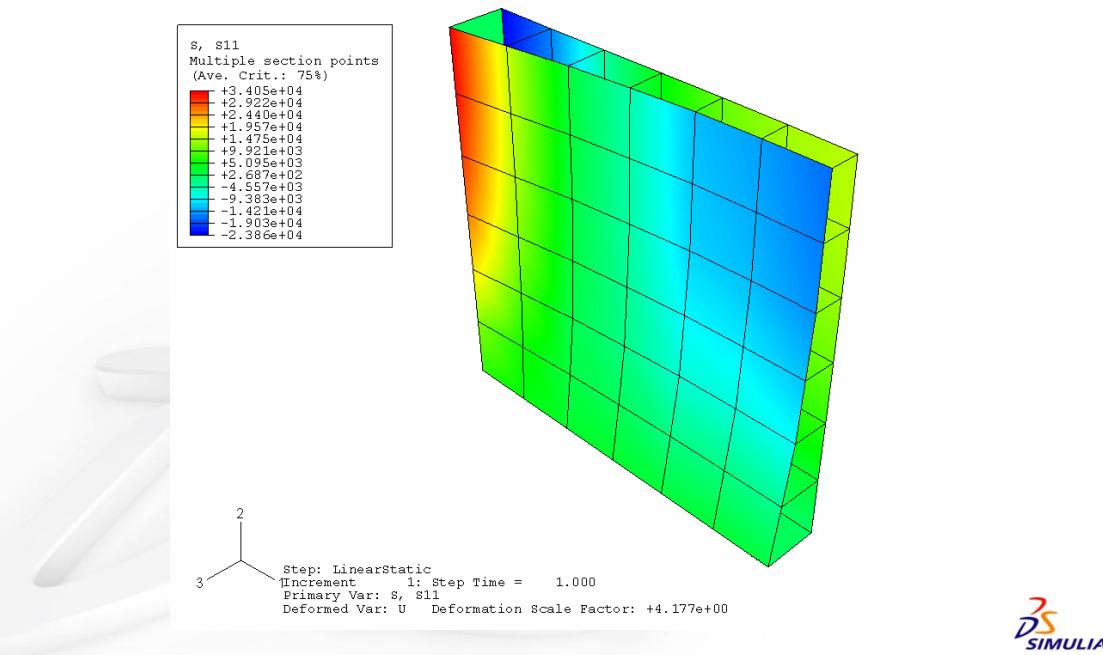


Analysis of Composite Materials with Abaqus

Abaqus Examples

- The stress in the 1-direction for the mixed conventional and continuum shell model.

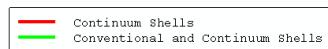
© DASSAULT SYSTEMES

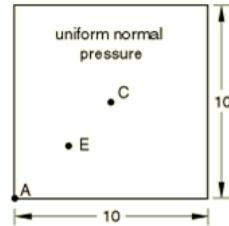
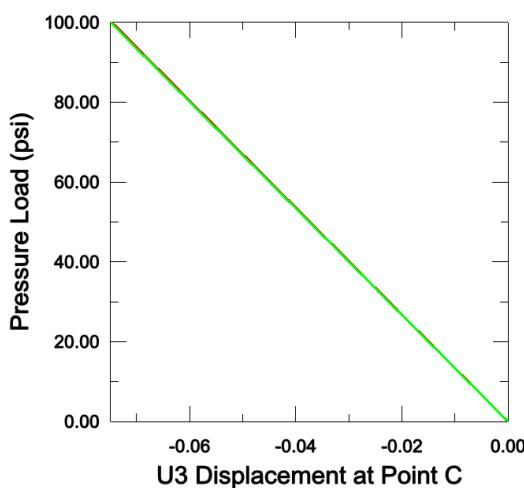


Analysis of Composite Materials with Abaqus

Abaqus Examples

- Plots of the displacement at point C versus pressure load





The displacement on the top and bottom of the SC8R element is averaged to produce the plot

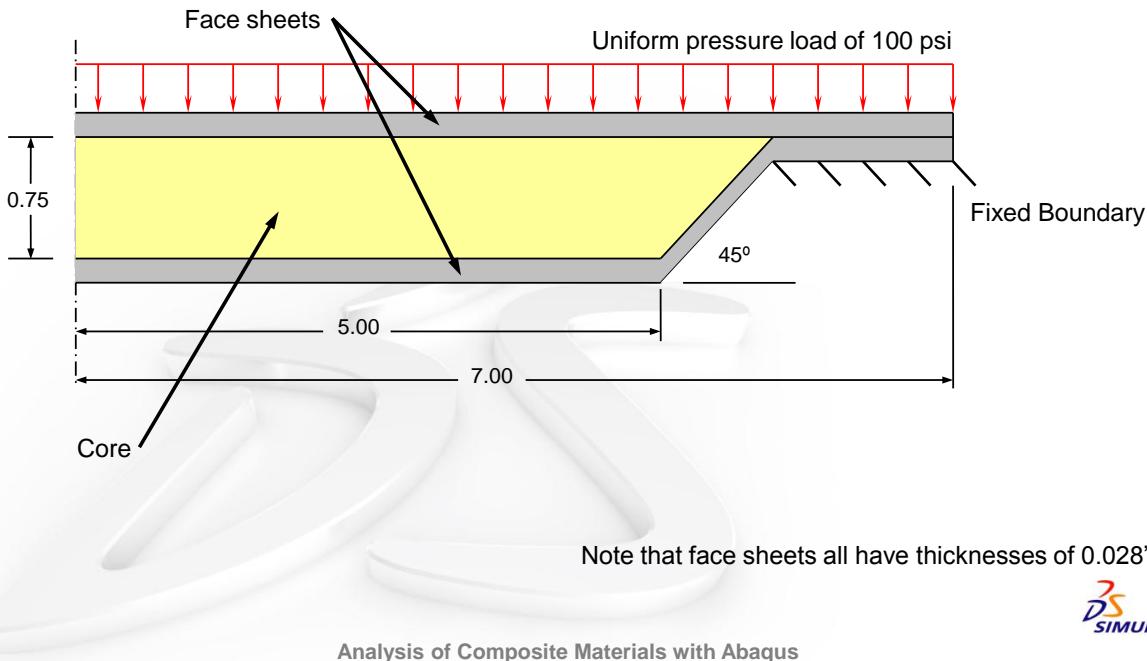
Abaqus Examples

4 Tapered Sandwich Composite

- The last example will utilize a tapered sandwich composite, which might be used in situations where you need to attach the laminate to a support structure.
 - For example, a sandwich composite laminate terminates at a bolted connection
- The material properties are the same as that for all the examples in this lecture
- Continuum shell elements (SC8R) will be utilized to discretize the core and face sheets

Abaqus Examples

- Geometric configuration of the sandwich composite:

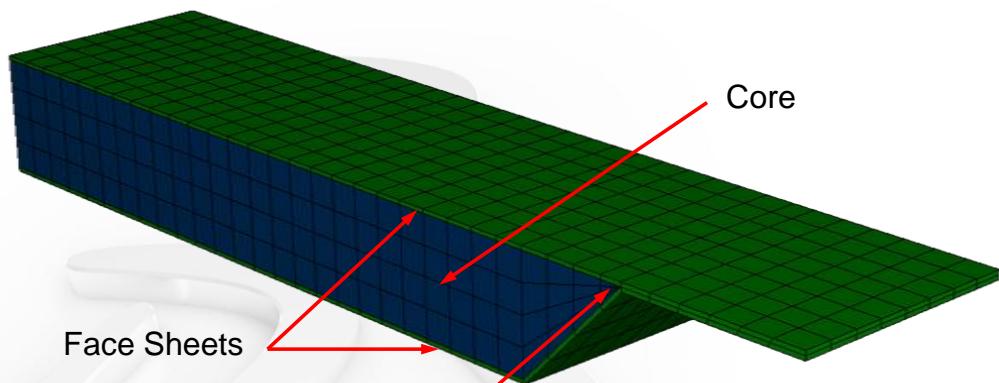


© DASSAULT SYSTEMES

Abaqus Examples

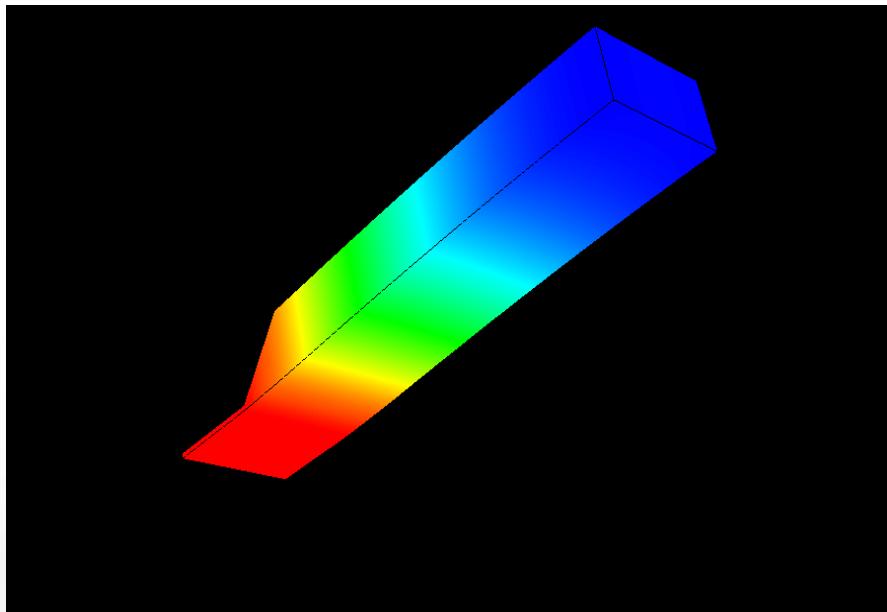
- The mesh of the tapered composite sandwich laminate:
 - 1488 SC8R and C3D6 elements
 - 2016 nodes

© DASSAULT SYSTEMES



Abaqus Examples

- Displacement plot of uniformly loaded panel



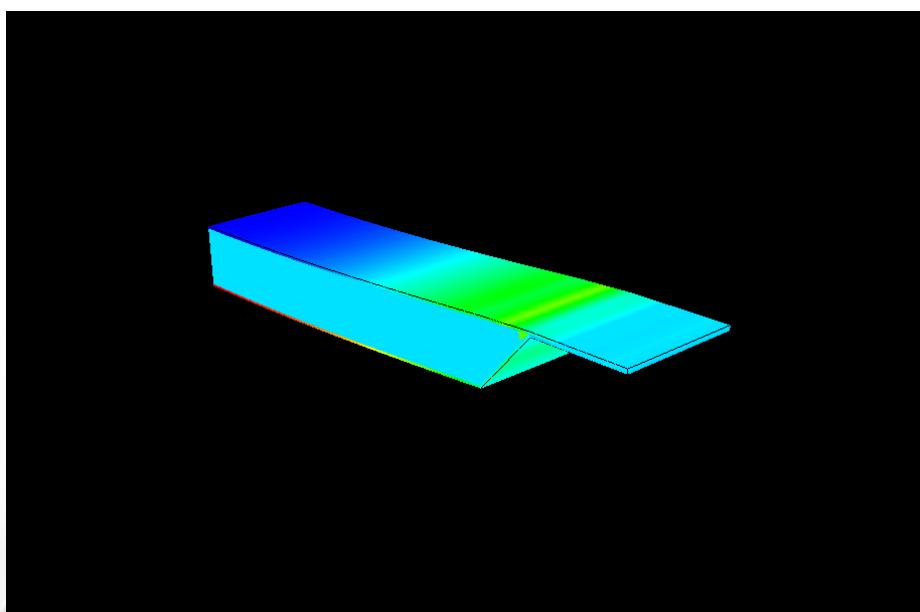
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Examples

- Stress plot of uniformly loaded panel in the longitudinal direction



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Notes

Notes

Analysis of Stiffened Composite Panels

Lecture 8

© DASSAULT SYSTEMES



L8.2

Overview

- **Stiffened Composite Panels**
- **Abaqus Usage**
- **Abaqus Example**

© DASSAULT SYSTEMES



Stiffened Composite Panels

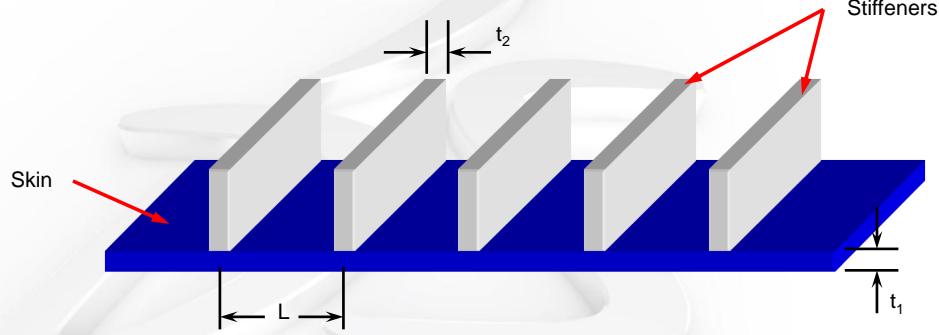
© DASSAULT SYSTEMES



L8.4

Stiffened Composite Panels

- Discretely stiffened composite laminates will be the focus of this lecture.
 - This structure is created by attaching bracing elements to a composite plate or shell.
 - This structural configuration is utilized extensively in many industries (e.g., stiffening the fuselage of an aircraft).



© DASSAULT SYSTEMES

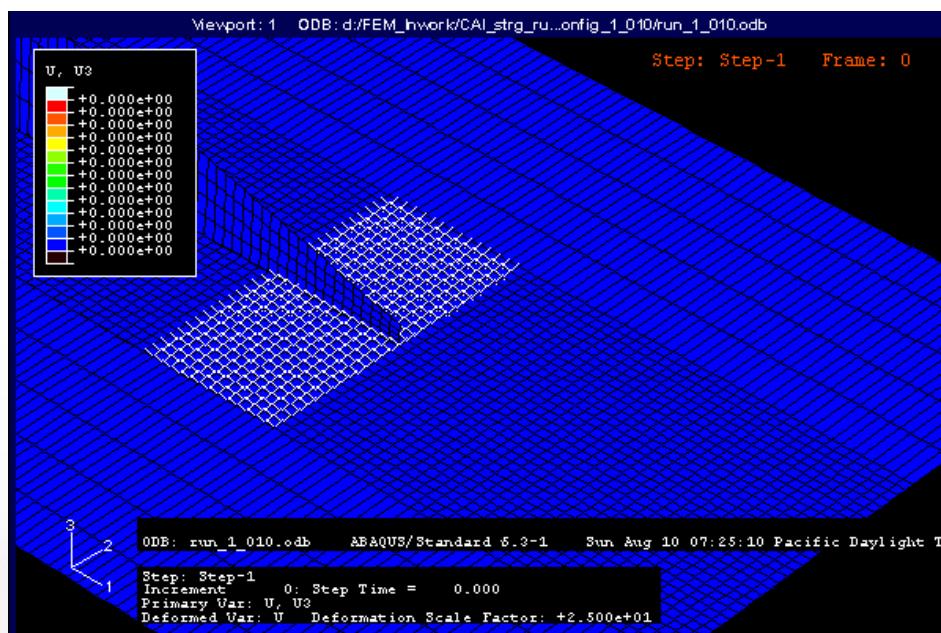


Stiffened Composite Panels

- Many stiffened composite plates are manufactured by co-curing or adhesively bonding the stiffener to the panel.
- The connection between the skin and stiffener is usually of importance due to integrity concerns.
- Abaqus provides certain features for analyzing the failure of the interface between a skin and stringer (specifically, these techniques model progressive fracture along a path).
 - Cohesive elements and cohesive contact
 - discussed in Lecture 10, “Cohesive Connections.”
 - Virtual Crack Closure Technique (VCCT)
 - discussed in Lecture 11, “VCCT for Abaqus.”

Stiffened Composite Panels

- Example of a stringer pop-off analyzed with VCCT:



Stiffened Composite Panels

- The main purpose of this lecture, however, is to study how to actually model the skin/stiffener combination itself.
 - This will be the focus of the rest of the lecture.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Usage

© DASSAULT SYSTEMES



Abaqus Usage

- The modeling of the skin is fairly straightforward in terms of the techniques that would be utilized.
 - The skin is typically a thin composite layup; therefore, it would be modeled utilizing a conventional or continuum shell element, or perhaps a composite solid utilizing the mixed modeling techniques described previously in Lectures 3, 4 and 5.
- The modeling of the stiffener is the primary focus of this lecture.
- The geometry of the stiffener, as well as the pattern of the stiffener deployment, can influence the modeling strategy.
 - For example, if the spacing of the stiffener is small, as compared to the in-plane dimensions of the plate, then perhaps we could smear the stiffness properties of the stiffener over the skin, rather than modeling the stiffeners discretely

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Abaqus Usage

- The following are possible ways in which a stiffener might be modeled discretely:
 - Beam elements
 - Shell elements (both conventional and continuum)
 - Solid elements

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Abaqus Usage

- **Using beam elements to model a stiffener**
 - Beam elements might be utilized as a cost-effective way to model the stiffeners accurately.
 - Geometry must conform to the beam assumption that the cross-sectional dimensions are small compared to the length of the beam.
 - For isotropic materials, the material definition for the beam element is fairly straightforward.
 - A composite stiffener will, in general, be composed of different materials throughout the cross section.
 - The meshed beam cross-section technique mentioned in Lecture 3 may provide a solution to determining the section behavior in this instance.
 - Beam elements preclude the use of layers that may induce bending-twisting coupling (e.g., an unsymmetric laminate).

© DASSAULT SYSTEMES



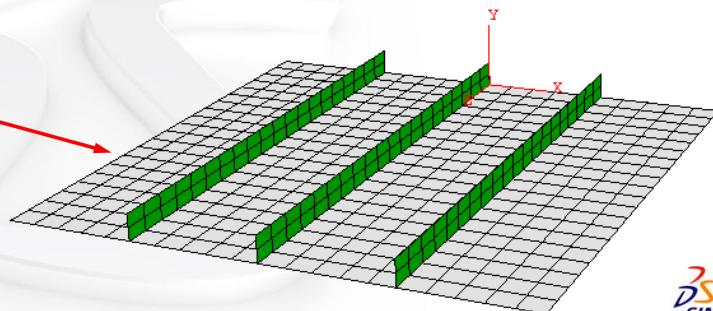
Analysis of Composite Materials with Abaqus

Abaqus Usage

- **Using shell elements to model a stiffener**
 - Shell elements may be needed to model the stiffeners accurately if the restrictions regarding the utilization of beam elements are too great for your purposes
 - Again, utilization of shell elements requires that the geometry of the stiffener adhere to the underlying assumption of a shell:
 - The thickness dimension is much smaller than the other dimensions
 - This analysis methodology does allow for orientations of the lamina such that deformation couplings are possible (e.g., bending-twisting coupling)

© DASSAULT SYSTEMES

From the blast loading on a stiffened plate example in the Getting Started with Abaqus manual

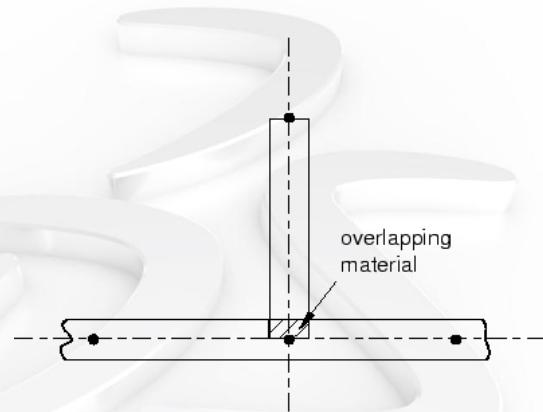


Analysis of Composite Materials with Abaqus

Abaqus Usage

- Shell elements allow you to offset the reference surface (location of the nodes) from the mid-surface
- This feature can be utilized to correct the material overlap issue that can occur when two shell elements are attached perpendicularly to each other

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Usage

- **Using solid elements to model a stiffener**
 - Utilization of solid elements represents the most general way to model the stiffeners, as well as the most computationally expensive (3D discretization)
 - Local refinement can be accomplished if you utilize solid elements in the area of interest, and some other type of element globally
 - The two regions would be coupled together utilizing one of Abaqus' coupling constraints (shell-to-solid, kinematic or distributing coupling, etc.)

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Example

© DASSAULT SYSTEMES



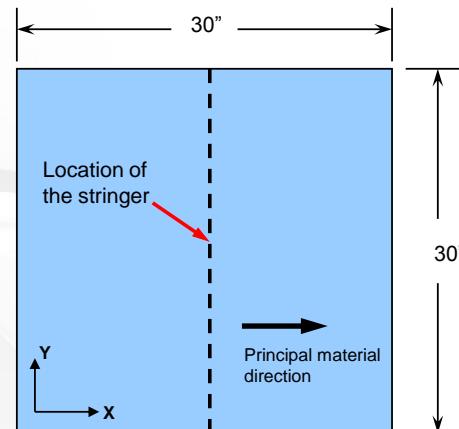
L8.16

Abaqus Example

- **Composite Laminate with Single Stiffener**
 - A composite plate is analyzed with a single stringer to act as a bracing element
 - The stringer itself has the profile of a T-joint, which is adhesively bonded to the skin
 - All four edges of the plate are simply supported, and a uniform pressure load of 10 psi is applied to the face of the skin
 - The stiffener is assumed isotropic with $E = 1e7$ psi and $\nu = 0.3$
 - The following material properties are utilized for the skin:

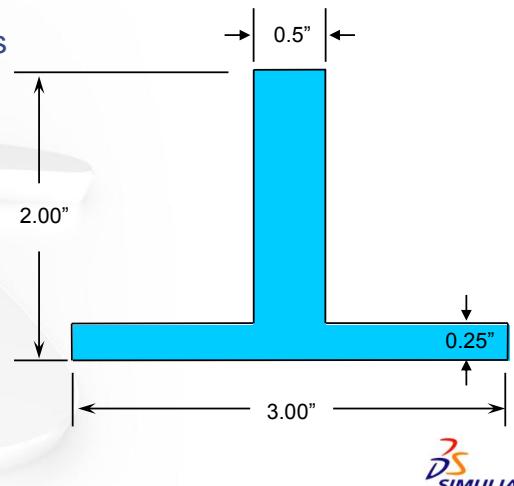
$$E_1 = 1.0 \times 10^7 \text{ psi}, E_2 = 4.0 \times 10^6 \text{ psi}, \nu_{12} = 0.3,$$

$$G_{12} = G_{13} = G_{23} = 1.875 \times 10^6 \text{ psi}$$



Abaqus Example

- In this example, the stringer is modeled using beam elements as well as conventional and continuum shell elements
- Utilizing a beam or conventional shell element to model this type of stringer leads to an idealization
 - The continuous transition at the intersection cannot be captured
 - The transition can be captured utilizing continuum shell elements
- The stiffener will be offset from the plate to eliminate material overlap using the following as appropriate:
 - Beam profile offset
 - Shell offsets



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Abaqus Example

- The stringer is ‘bonded’ to the skin utilizing a TIE constraint when it is modeled with shell elements (when modeled with beam elements, shared nodes are used instead)
 - If stringer pop-off is of importance, tied contact may be a better choice, if applicable
- The model with conventional shell and beam elements consists of:
 - 400 S4R elements + 20 B31 elements
 - 441 nodes (shared nodes between the beam and the shell)
- The model with only conventional shell elements consists of:
 - 625 S4R elements
 - 701 nodes
- The model with both conventional and continuum shell elements consists of:
 - 1040 S4R and SC8R elements
 - 1548 nodes

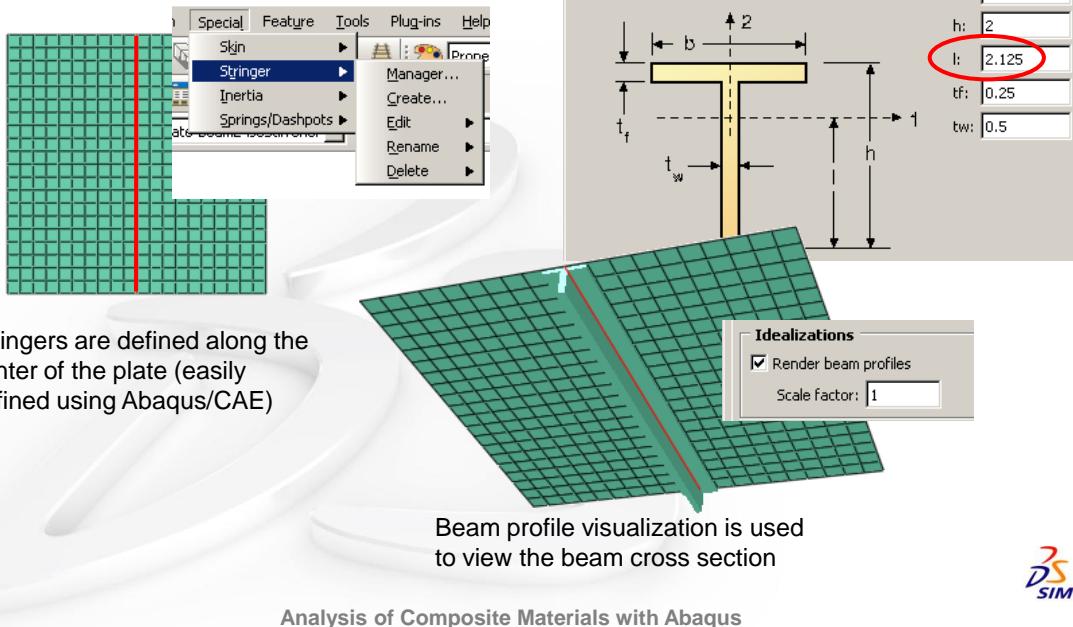
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Example

- Mesh of the geometry
 - The beam and conventional shell element mesh:

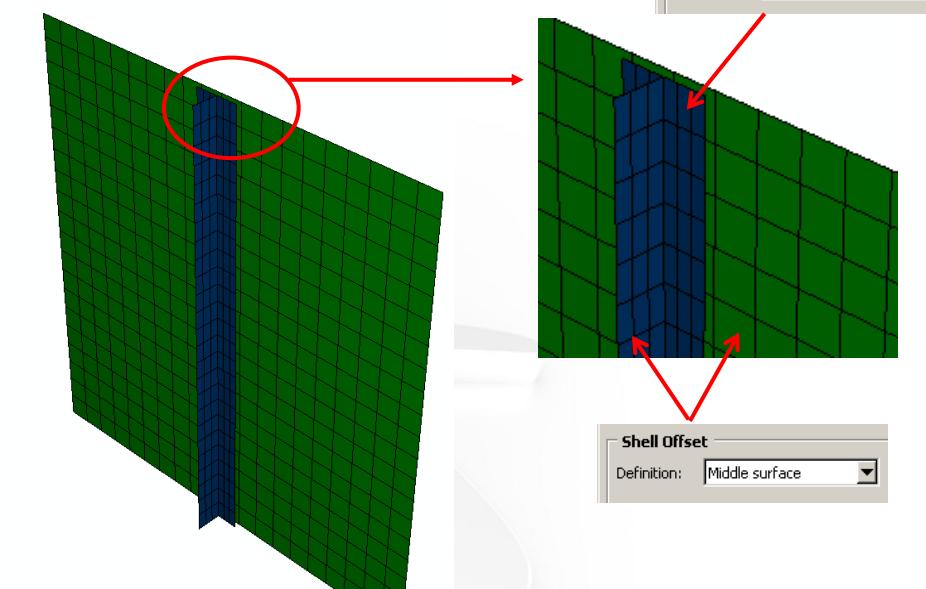


© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Abaqus Example

- The conventional shell element mesh:



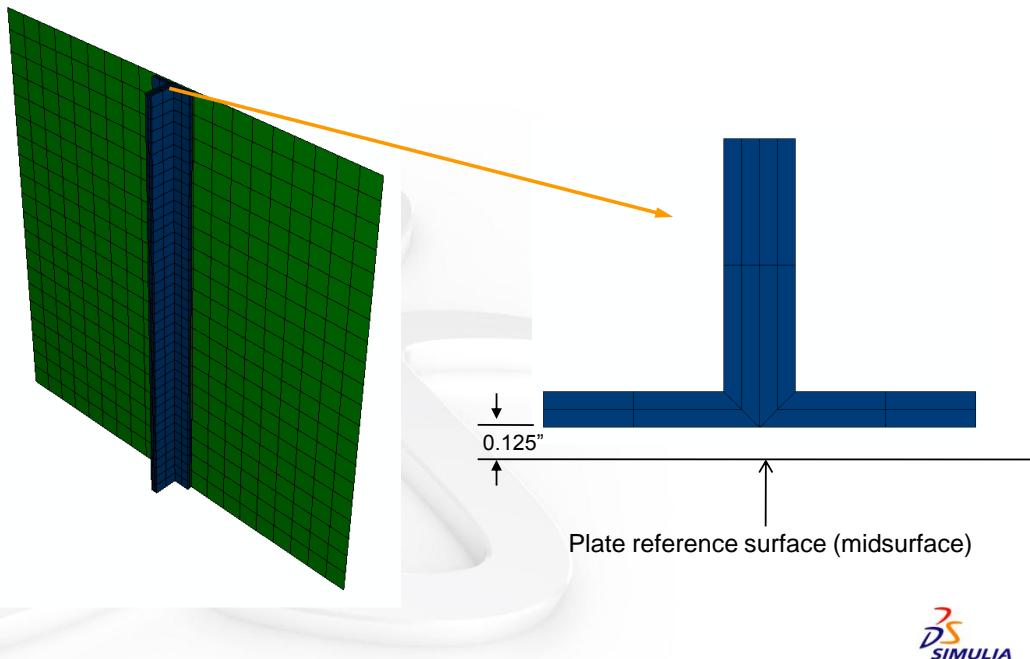
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Abaqus Example

- The conventional and continuum shell element mesh:

© DASSAULT SYSTEMES



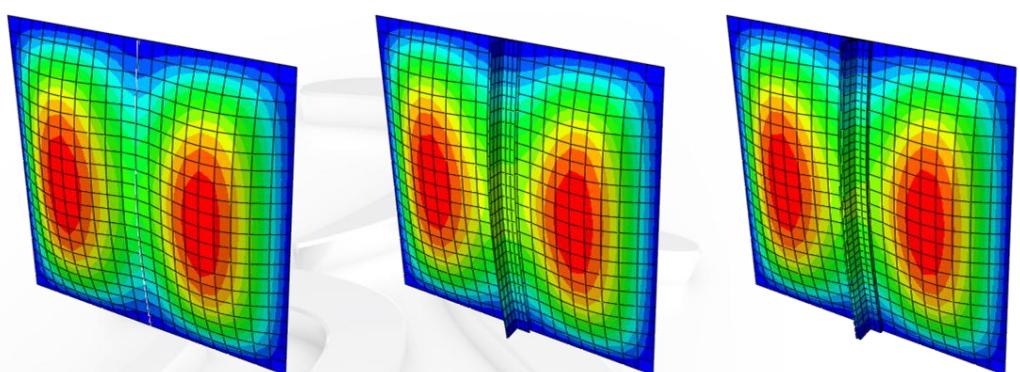
Analysis of Composite Materials with Abaqus

Abaqus Example

- Results:**

- Statically displaced shapes (contours of transverse displacement)

© DASSAULT SYSTEMES



Beam stiffener

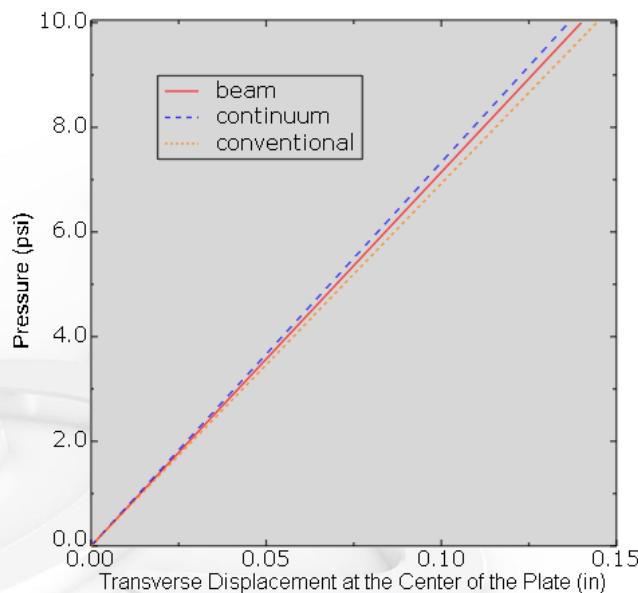
Conventional shell stiffener

Continuum shell stiffener

Analysis of Composite Materials with Abaqus

Abaqus Example

- A comparison of the transverse displacement of the center of the plate versus applied load



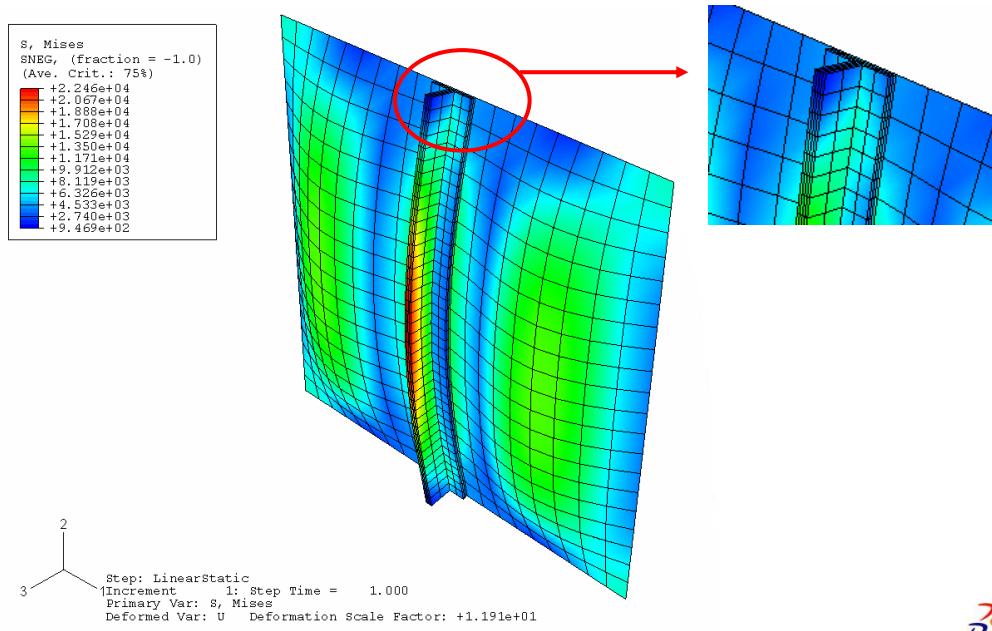
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Abaqus Example

- Static stress solution (conventional/continuum shell model with 10 psi load)



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Notes

Notes

Modeling Damage and Failure in Composites

Lecture 9

© DASSAULT SYSTEMES



L9.2

Overview

- Failure Criteria in Laminates
- Failure Theories
- Progressive Damage of Fiber-Reinforced Composites
- Example
- Import of Composite Damage Model

© DASSAULT SYSTEMES



Failure Criteria in Laminates

© DASSAULT SYSTEMES

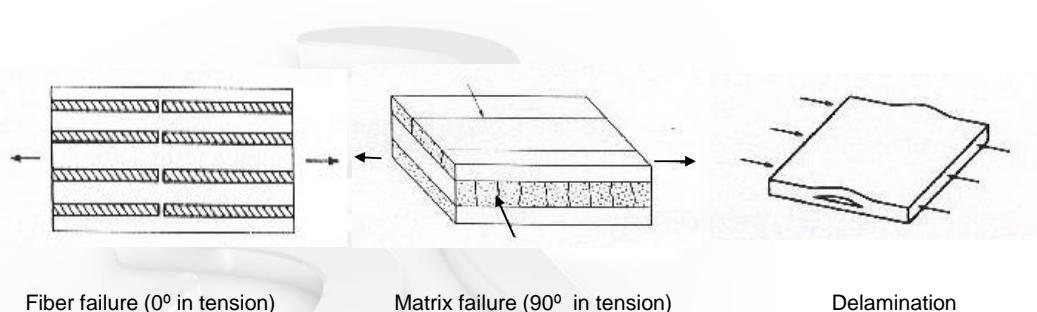


L9.4

Failure Criteria in Laminates

• Failure modes

- A composite material can fail in two basic failure modes:
 - Failure of individual layers (plies) in tension, compression, or shear
 - Delamination between plies



© DASSAULT SYSTEMES



Failure Criteria in Laminates

- Ply failure is associated with membrane and bending stresses and is dependent on the loading and orientation of the reinforcement.
- Delamination is associated with the presence of transverse shear stresses.
 - This mode can be important in laminates under fatigue loading and static overloading.
 - Estimates of the transverse shear stresses, obtained as described previously in this course, can help assess this failure.
 - Delamination can be modeled using cohesive elements or VCCT (discussed in subsequent lectures).
- A laminate should be designed so that its tendency for ply and delamination failure is minimized.
 - Failure due to delamination is more insidious, as it is less predictable than ply failure, but ultimately, the laminate needs to be designed not to fail.

Failure Criteria in Laminates

- **Ply failure**
 - The in-plane failure mode depends on the sign and direction of the stresses and strains relative to the direction of reinforcement.
 - As a result, the first failure does not necessarily occur in one of the extreme layers. Hence, all layers need to be monitored for failure.
 - The ply failure can occur due to any or all of the following failure modes:
 - Fiber tension and fiber compression failure modes
 - Matrix tension and matrix compression failure modes

Failure Criteria in Laminates

- Ply failure modes can occur sequentially and are likely to interact, which makes it very difficult to model the behavior of the laminate subsequent to initial failure.
 - Common analysis practice determines only the point of first failure.
 - Several such “initial failure envelopes” are available in Abaqus.
- Abaqus provides five standard plane stress orthotropic failure measures.
 - These failure measures are purely postprocessed output requests and do not cause any material degradation.
 - All failure measures take values greater than or equal to 0.0, with values greater than or equal to 1.0 implying failure.
- Abaqus also offers a damage model to predict the onset of damage and to model damage evolution.
 - This takes into account all four ply failure modes

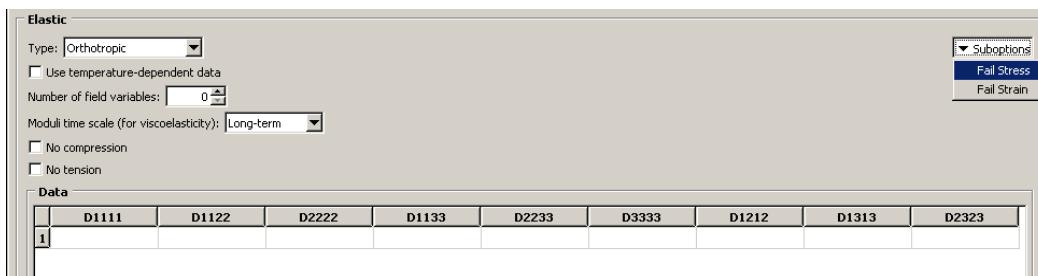
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Criteria in Laminates

- The failure criteria are defined as suboptions of the elastic material properties.



- Note that Abaqus denotes the orthotropic material directions by 1 and 2, with the 1-direction aligned with the fibers and the 2-direction transverse to the fibers.

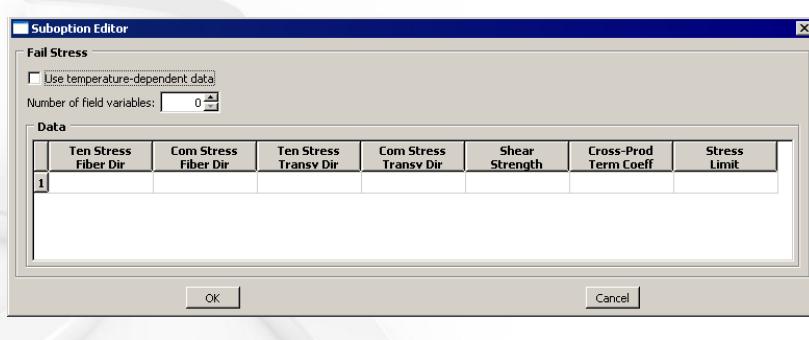
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Criteria in Laminates

- The **stress-based** failure option defines tensile and compressive stress limits in the 1- and 2-directions; shear strength in the 1-2 plane; a scaling factor used to define the Tsai-Wu coefficient, F_{12} , if no biaxial stress limit is given; and the biaxial stress limit.
- Usage:



***FAIL STRESS**

$X_p, X_c, Y_p, Y_c, S, f^*, \sigma_{biax}$

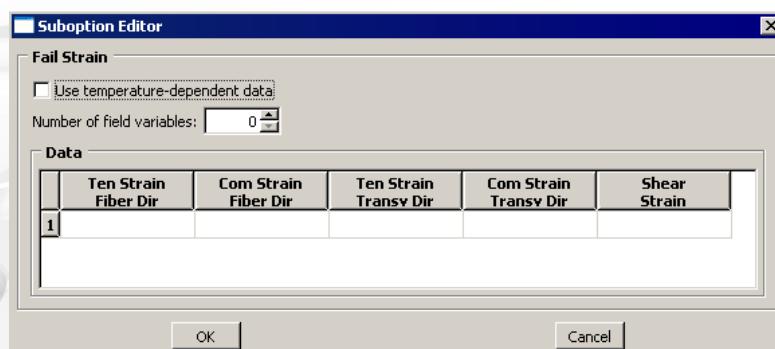
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Criteria in Laminates

- The **strain-based** failure option defines tensile and compressive strain limits in the 1- and 2-directions and the shear strain limit in the 1-2 plane.
- Usage:



***FAIL STRAIN**

$X_{\epsilon_t}, X_{\epsilon_c}, Y_{\epsilon_t}, Y_{\epsilon_c}, S_{\epsilon}$

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories

© DASSAULT SYSTEMES



L9.12

Failure Theories

- Stress-based failure theories

- Maximum stress theory

- This is the simplest stress-based failure model. It measures each stress component against the corresponding stress limit.
 - It is defined:
 - If $\sigma_{11} > 0$, set $X = X_t$; otherwise, set $X = X_c$.
 - If $\sigma_{22} > 0$, set $Y = Y_t$; otherwise, set $Y = Y_c$.
 - The maximum stress failure criterion requires that

$$I_F = \max\left(\frac{\sigma_{11}}{X}, \frac{\sigma_{22}}{Y}, \left|\frac{\sigma_{12}}{S}\right|\right) < 1.0.$$

- Since this theory provides no interaction between the stress components in different directions, its accuracy is limited.

© DASSAULT SYSTEMES



Failure Theories

- **Tsai-Hill theory**

- This model is widely used as a simple failure criterion for composite lamina since it was first proposed by Tsai in 1968 as an extension of Hill's anisotropic plasticity model.
- The model is:
 - If $\sigma_{11} > 0$, set $X = X_t$; otherwise, set $X = X_c$.
 - If $\sigma_{22} > 0$, set $Y = Y_t$; otherwise, set $Y = Y_c$.
- The Tsai-Hill failure criterion requires that

$$I_F = \frac{\sigma_{11}^2}{X^2} - \frac{\sigma_{11}\sigma_{22}}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} < 1.0.$$

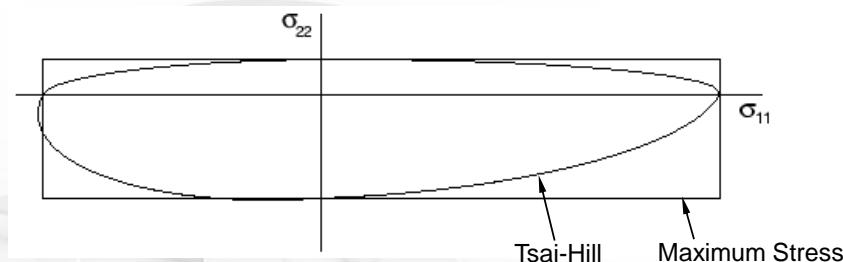
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories

- This is a piecewise-continuous failure surface.
 - At any given value of shear stress, σ_{12} , it provides four separate elliptical arcs in $(\sigma_{11}, \sigma_{22})$ space.
 - The shape of the arc depends on the signs of σ_{11} and σ_{22} .



Tsai-Hill versus Maximum Stress failure envelope

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories

- **Tsai-Wu theory**

- The Tsai-Wu failure criterion was introduced to provide a smooth form of the Tsai-Hill criterion. It uses an additional parameter (F_{12} below), which is defined in Abaqus by specifying either f^* or σ_{biax} .

- The criterion is

$$I_F = F_1\sigma_{11} + F_2\sigma_{22} + F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\sigma_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22} < 1.0.$$

- The Tsai-Wu coefficients are defined:

$$\begin{aligned} F_1 &= \frac{1}{X_t} + \frac{1}{X_c}, & F_2 &= \frac{1}{Y_t} + \frac{1}{Y_c}, \\ F_{11} &= -\frac{1}{X_t X_c}, & F_{22} &= -\frac{1}{Y_t Y_c}, & F_{66} &= \frac{1}{S^2}. \end{aligned}$$

Failure Theories

- If σ_{biax} is given,

$$F_{12} = \frac{1}{2\sigma_{biax}^2} \left[1 - \left(\frac{1}{X_t} + \frac{1}{X_c} + \frac{1}{Y_t} + \frac{1}{Y_c} \right) \sigma_{biax} + \left(\frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} \right) \sigma_{biax}^2 \right];$$

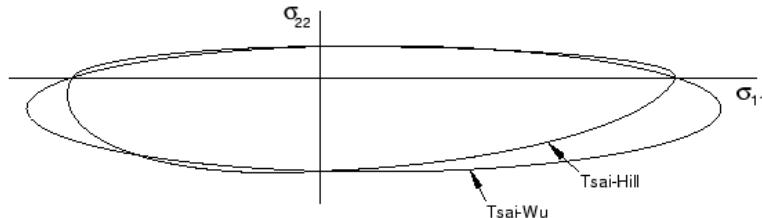
otherwise,

$$F_{12} = f^* \sqrt{F_{11}F_{22}},$$

where $-1.0 \leq f^* \leq 1.0$ and the default value of f^* is zero.

Failure Theories

- If $F_{12} = 0$, the Tsai-Wu model appears as below compared to the Tsai-Hill model:



Tsai-Hill versus Tsai-Wu failure envelope ($F_{12} = 0.0$)

- Proper choice of F_{12} can provide slightly more accurate results compared to experimental data, although the difference usually is not large.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories

- Azzi-Tsai-Hill theory**

- The Azzi-Tsai-Hill failure theory is the same as the Tsai-Hill theory, except that the absolute value of the cross product term is taken:

$$I_F = \frac{\sigma_{11}^2}{X^2} - \frac{|\sigma_{11}\sigma_{22}|}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} < 1.0.$$

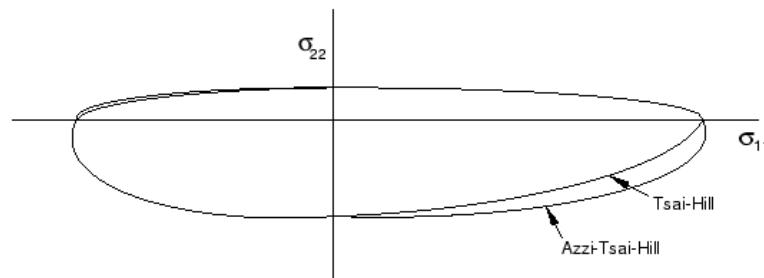
- This difference shows up only when σ_{11} and σ_{22} have opposite signs.
 - Thus, for a given σ_{12} the Azzi-Tsai-Hill surface differs from the Tsai-Hill surface only in the second and fourth quadrants of the $(\sigma_{11}, \sigma_{22})$ plane, where it is the outside bounding surface (i.e., further from the origin).

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories



Tsai-Hill versus Azzi-Tsai-Hill failure envelope

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories

- Strain-based failure theory
 - Maximum strain failure theory
 - This is a simple model that defines failures from the largest strain component relative to the corresponding strain limit:
 - If $\varepsilon_{11} > 0$, set $X_\varepsilon = X_{\varepsilon_t}$; otherwise, set $X_\varepsilon = X_{\varepsilon_c}$.
 - If $\varepsilon_{22} > 0$, set $Y_\varepsilon = Y_{\varepsilon_t}$; otherwise, set $Y_\varepsilon = Y_{\varepsilon_c}$.
 - The maximum strain failure criterion requires that

$$I_F = \max \left(\frac{\varepsilon_{11}}{X_\varepsilon}, \frac{\varepsilon_{22}}{Y_\varepsilon}, \left| \frac{\gamma_{12}}{S_\varepsilon} \right| \right) < 1.0.$$

- Experimental comparison has shown that this model generally is not as accurate as the maximum stress failure theory in general strain states.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Failure Theories

- **Output failure indices**

- When failure measures are defined and output is requested, Abaqus outputs the failure index R .
- Each of the stress-based failure theories defines a failure surface surrounding the origin in the three-dimensional space $\{\sigma_{11}, \sigma_{22}, \sigma_{12}\}$.
 - Failure occurs any time a state of stress is either on or outside this surface.
 - To measure the proximity to the failure surface, the failure index R is used.
 - R is defined as the scaling factor so that for the given stress state $\{\sigma_{11}, \sigma_{22}, \sigma_{12}\}$,

$$\left\{ \frac{\sigma_{11}}{R}, \frac{\sigma_{22}}{R}, \frac{\sigma_{12}}{R} \right\} \Rightarrow I_F = 1.0.$$

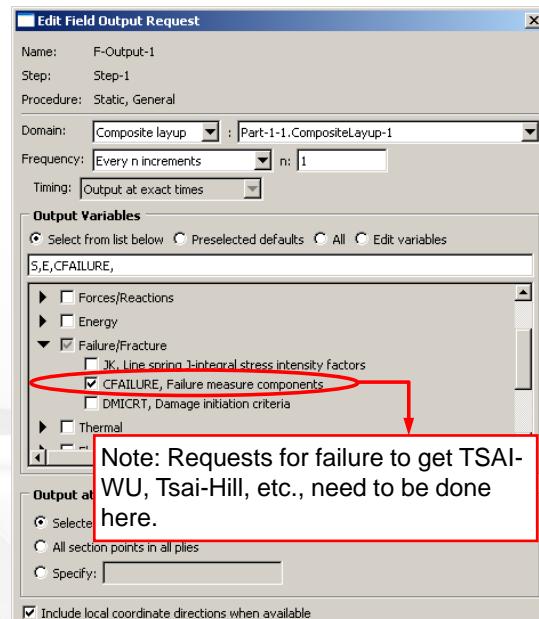
Failure Theories

- That is, $1/R$ is the scaling factor by which we need to multiply all of the stress components simultaneously to hit the failure surface.
- Values of $R < 1.0$ indicate that the state of stress is within the failure surface, while values of $R \geq 1.0$ indicate failure.
 - For the maximum stress theory $R \equiv I_F$.
- For the maximum strain failure theory the failure index is defined similarly.
 - The failure index R is the scaling factor such that for the given strain state $\{\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{12}\}$,

$$\left\{ \frac{\varepsilon_{11}}{R}, \frac{\varepsilon_{22}}{R}, \frac{\varepsilon_{12}}{R} \right\} \Rightarrow I_F = 1.0.$$

Failure Theories

- Output variable names
 - CFAILURE
All failure measure components
 - MSTRS
Maximum stress theory failure measure
 - TSAIH
Tsai-Hill theory failure measure
 - TSAIW
Tsai-Wu theory failure measure
 - AZZIT
Azzi-Tsai-Hill theory failure measure
 - MSTRIN
Maximum strain theory failure measure

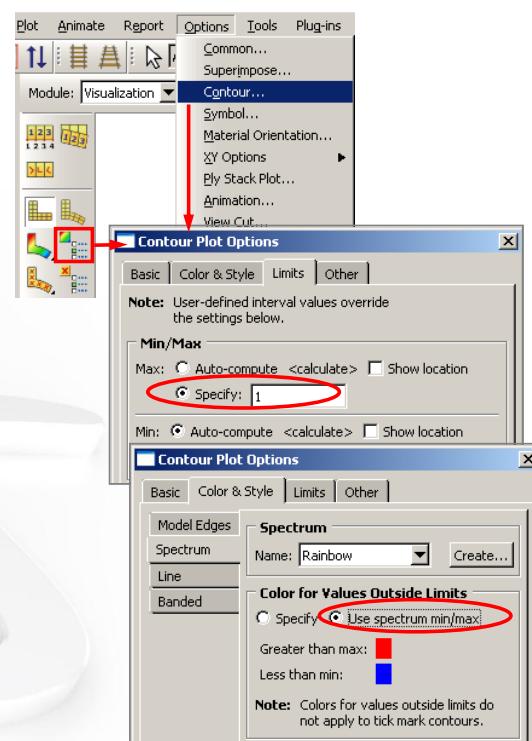


```
*OUTPUT, FIELD
*ELEMENT OUTPUT, ELSET=CompLayup-1
CFAILURE
```

Analysis of Composite Materials with Abaqus

Failure Theories

- Visualization tip:
 - Set the maximum contour level to 1.0 in Abaqus/Viewer and use the spectrum min/max colors as the color for values outside the limits.
 - Then, plot the failure measure contours for any given layer.
 - Anything that appears in red has failed.
 - The failure measures are indications of material failure, but do not cause any material degradation.
 - To model material degradation, use alternative damage and failure models that will be discussed in next section "Progressive Damage of Fiber-Reinforced Composites."



Analysis of Composite Materials with Abaqus

Progressive Damage of Fiber-Reinforced Composites

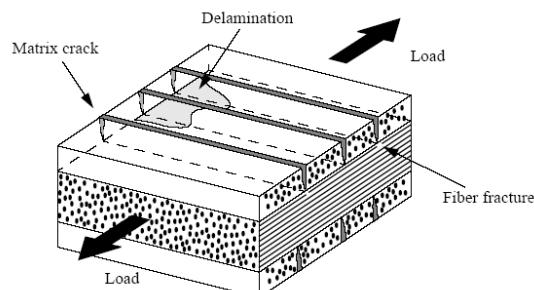
© DASSAULT SYSTEMES



L9.26

Progressive Damage of Fiber-Reinforced Composites

- Abaqus offers a general capability for modeling progressive damage and failure in fiber-reinforced composites.
 - Material failure refers to the complete loss of load carrying capacity that results from progressive degradation of the material stiffness.
 - Stiffness degradation is modeled using damage mechanics.
 - Elements with a plane stress formulation (plane stress, shell, continuum shell, and membrane elements) must be used for modeling.
- 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



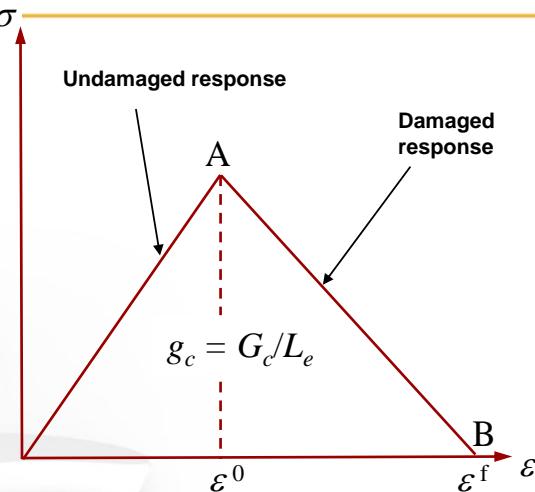
Common damage types
in composite laminates

© DASSAULT SYSTEMES

Progressive Damage of Fiber-Reinforced Composites

- Components of material definition**

- Undamaged constitutive behavior (e.g. Elastic: orthotropic or anisotropic)
- Damage initiation (point A)
- Damage evolution (path A–B)
- Choice of element removal (point B)



Keywords

*MATERIAL

*ELASTIC, TYPE=ORTHOTROPIC

*DAMAGE INITIATION, CRITERION=criterion

*DAMAGE EVOLUTION

*SECTION CONTROLS, ELEMENT DELETION=YES

Typical material response showing progressive damage

Multiple damage definitions are allowed



Analysis of Composite Materials with Abaqus

Progressive Damage of Fiber-Reinforced Composites

- Damage initiation criteria**

- Damage initiation defines the point of initiation of degradation of stiffness
- The behavior of the undamaged material is linearly elastic
- The damage initiation criteria for fiber reinforced composites are based on Hashin's theory
- It does not actually lead to damage unless damage evolution is also specified
 - Output variables associated with each criterion
 - Useful for evaluating the severity of current deformation state
- Output
 - HSNFTCRT – tensile fiber Hashin's criterion
 - HSNFCCRT – compressive fiber Hashin's criterion
 - HSNMTCRT – tensile matrix Hashin's criterion
 - HSNMCCRT – compressive matrix Hashin's criterion

Progressive Damage of Fiber-Reinforced Composites

- Hashin's damage initiation criteria

Mode I: Fiber Tension ($\hat{\sigma}_{11} \geq 0$)

$$F_{ft} = \left(\frac{\hat{\sigma}_{11}}{X^T} \right)^2 + \alpha \left(\frac{\hat{\sigma}_{12}}{S^L} \right)^2 = 1, \text{ where } 0 \leq \alpha \leq 1$$

Mode II: Fiber Compression ($\hat{\sigma}_{11} < 0$)

$$F_{fc} = \left(\frac{\hat{\sigma}_{11}}{X^C} \right)^2 = 1$$

Mode III: Matrix Tension ($\hat{\sigma}_{22} \geq 0$)

$$F_{mt} = \left(\frac{\hat{\sigma}_{22}}{Y^T} \right)^2 + \left(\frac{\hat{\sigma}_{12}}{S^L} \right)^2 = 1$$

Mode IV: Matrix Compression ($\hat{\sigma}_{22} < 0$)

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

Effective stress: $\hat{\sigma} = \begin{bmatrix} \frac{1}{1-d_f} & 0 & 0 \\ 0 & \frac{1}{1-d_m} & 0 \\ 0 & 0 & \frac{1}{1-d_s} \end{bmatrix} \sigma$

where,

$\hat{\sigma}$ – effective stress

σ – true stress

d_f, d_m, d_s – damage variables

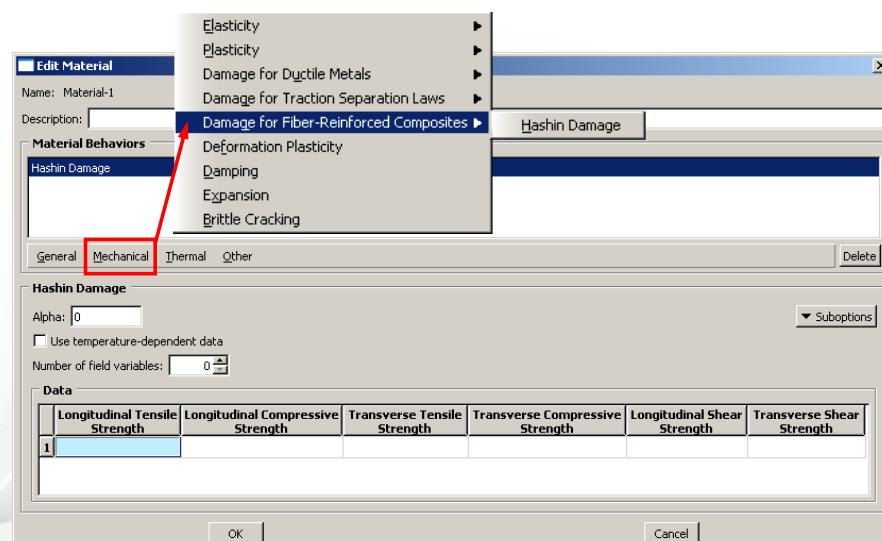


Analysis of Composite Materials with Abaqus

Progressive Damage of Fiber-Reinforced Composites

- User interface for damage initiation

*DAMAGE INITIATION, CRITERION=HASHIN, ALPHA=<alpha>
 $X^T, X^C, Y^T, Y^C, S^L, S^T$



Analysis of Composite Materials with Abaqus

Progressive Damage of Fiber-Reinforced Composites

- **Damage evolution**

- Damage evolution defines the post damage-initiation material behavior.
 - That is, it describes the rate of degradation of the material stiffness once the initiation criterion is satisfied.
 - Response of the material after damage initiation is of the form: $\sigma = \mathbf{C}(d)\varepsilon$, where $\mathbf{C}(d)$ is the damaged elasticity matrix

$$\mathbf{C}(d) = \frac{1}{D} \begin{bmatrix} (1-d_f)E_1 & (1-d_f)(1-d_m)v_{21}E_1 & 0 \\ (1-d_f)(1-d_m)v_{12}E_2 & (1-d_m)E_2 & 0 \\ 0 & 0 & D(1-d_s)G \end{bmatrix}$$

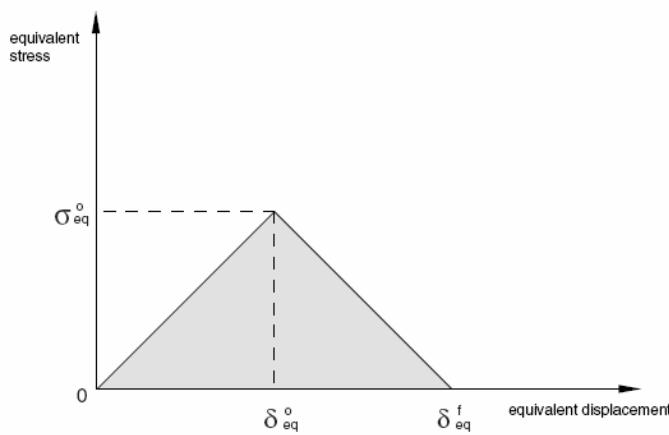
where

$$D = 1 - (1-d_f)(1-d_m)v_{12}v_{21} > 0;$$

$d_s = 1 - (1-d_{ft})(1-d_{fc})(1-d_{mt})(1-d_{mc})$, and d_{ft} , d_{fc} , d_{mt} , and d_{mc} are damage variables corresponding to the four failure modes previously discussed.

Progressive Damage of Fiber-Reinforced Composites

- The evolution law is based on the energy dissipated during the process (based on the work of C. Davila and P. Camanho related to cohesive elements)
- Linear material softening is assumed

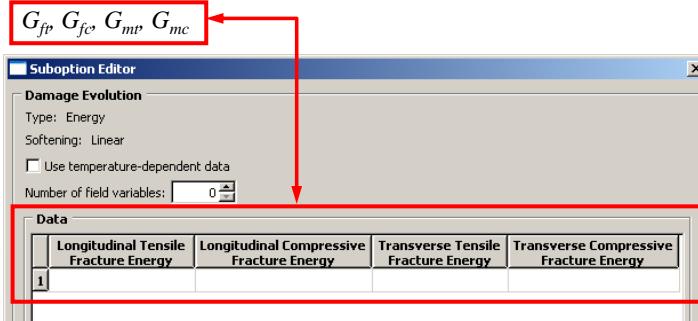


$$d = \frac{\delta_{eq}^f(\delta_{eq} - \delta_{eq}^0)}{\delta_{eq}(\delta_{eq}^f - \delta_{eq}^0)}$$

Progressive Damage of Fiber-Reinforced Composites

- Usage

*DAMAGE EVOLUTION, TYPE=ENERGY, SOFTENING=LINEAR



- Output

- Damage Variables

- DAMAGEFT – tensile fiber damage
 - DAMAGEFC – compressive fiber damage
 - DAMAGEMT – tensile matrix damage
 - DAMAGEMC – compressive matrix damage
 - DAMAGESHR – shear damage

Progressive Damage of Fiber-Reinforced Composites

- By default, an element is removed (deleted) once the damage variables for all failure modes at all material points in the element reach a value of D_{max}

*SECTION CONTROLS, ELEMENT DELETION={YES/ NO},
MAX DEGRADATION= D_{max} ,

- Output

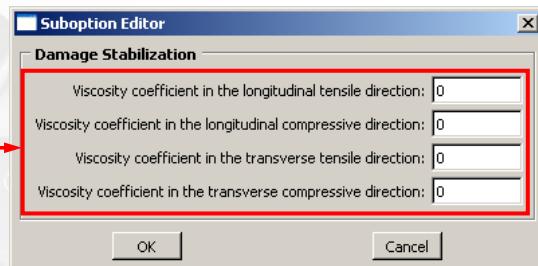
- STATUS – element status (1 – active, 0 – removed)

Progressive Damage of Fiber-Reinforced Composites

- **Viscous regularization**

- Stiffness degradation and material softening give rise to convergence problems.
- Viscous regularization can facilitate solution convergence
- In this regularization scheme, a viscous damage variable is defined for each damage mode:

***DAMAGE STABILIZATION**
 $\eta_{ft}, \eta_{fc}, \eta_{mt}, \eta_{mc}$



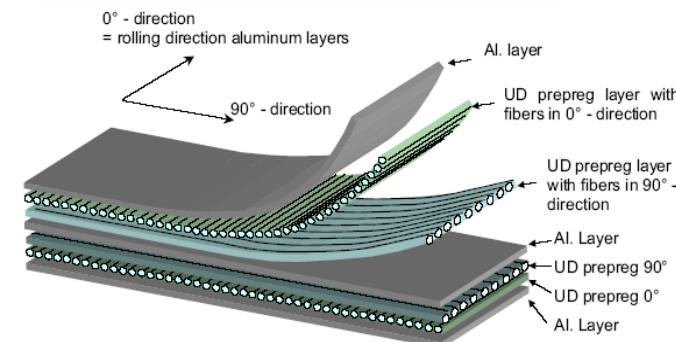
- Output

- Damage energy (ALLDMD, DMENER, ELDMD, EDMDDEN)
- Viscous regularization (ALLCD, CENER, ELCD, ECDDEN)

Progressive Damage of Fiber-Reinforced Composites

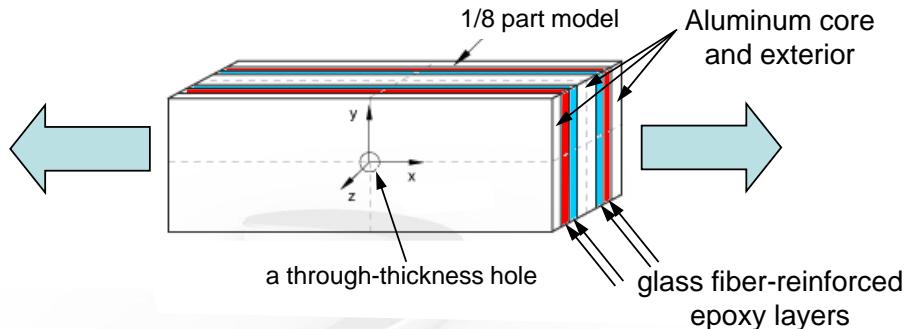
- **Example: Analysis of blunt notched fiber metal laminate**

- Fiber metal laminates (FMLs) are composed of:
 - laminated thin aluminum layers
 - Intermediate glass fiber-reinforced epoxy layers

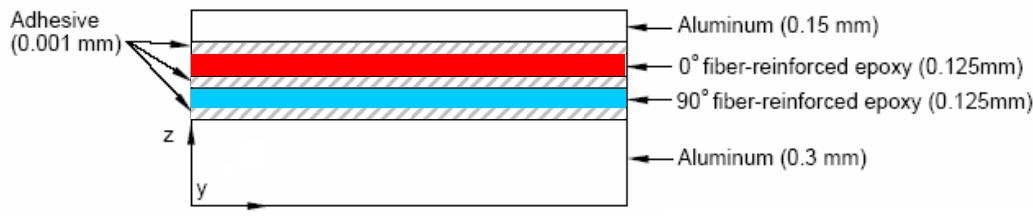


Progressive Damage of Fiber-Reinforced Composites

- Geometry of blunt notched fiber metal laminate (Glare 3 3/2-0.3)



- Through-thickness view of the laminate:

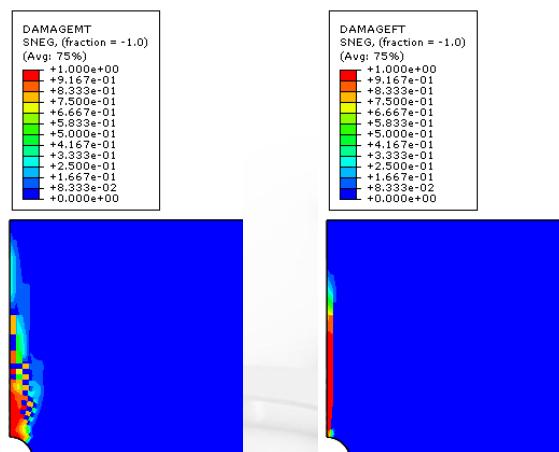


Analysis of Composite Materials with Abaqus

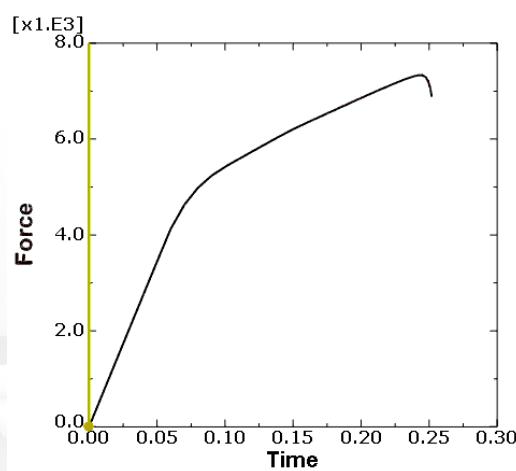
Example Problem 1.4.6, "Failure of blunt notched fiber metal laminates"

Progressive Damage of Fiber-Reinforced Composites

- Results



damage in matrix and damage in fibers
for one of glass fiber-reinforced epoxy layers



Net blunt notch strength (MPa)	
Test (De Vries, 2001)	446
Abaqus	453

Import of Composite Damage Model

© DASSAULT SYSTEMES

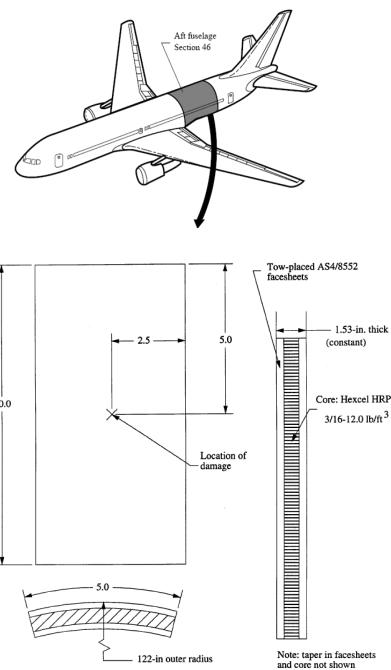
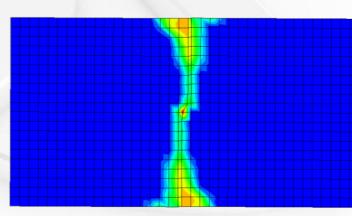
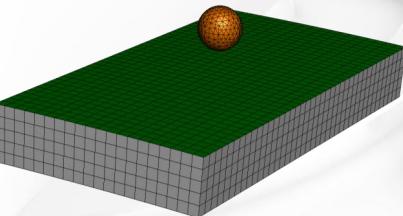


L9.40

Import of Composite Damage Model

- Abaqus allows the import of the damage model for fiber-reinforced composites from Abaqus/Explicit to Abaqus/Standard.
 - Details of the import capability will not be covered in this lecture (please refer to "Importing and transferring results," Section 9.2 of the Abaqus Analysis User's Manual).
- One typical application is the analysis of Barely Visible Impact Damage (BVID) in composite structures used in aerospace applications.
 - Non-visible damage to composite structures is a significant concern in the aerospace industry.

© DASSAULT SYSTEMES



from McGowan, D.M., and Ambur, D.R., NASA TM-110303
*Damage-Tolerance Characteristics of Composite Fuselage
Sandwich Structures With Thick Facesheets*

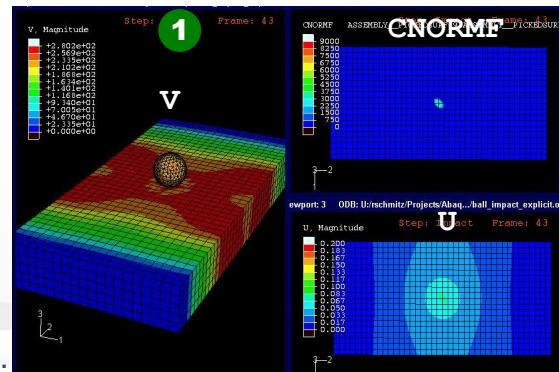
Import of Composite Damage Model

- Issues such as maintenance tool drop, hail impact, and other sources of damage must be dealt with as part of composite structure design.

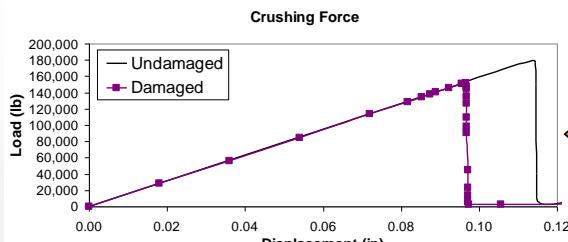
1 ABAQUS/Explicit is used to model low speed impact which results in damage.

- How much strength does a structure retain when small flaws are present?

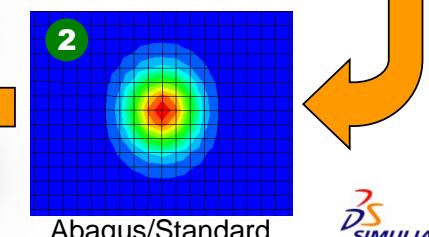
2 Further compression analysis of the damaged plate is conducted in Abaqus/Standard.



Abaqus/Explicit



Analysis of Composite Materials with Abaqus

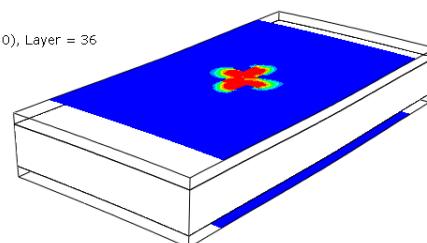
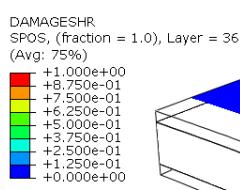


© DASSAULT SYSTEMES

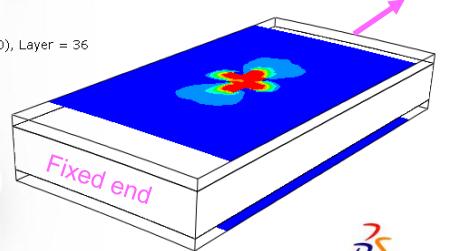
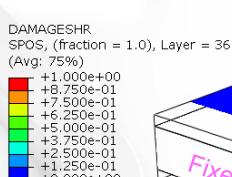
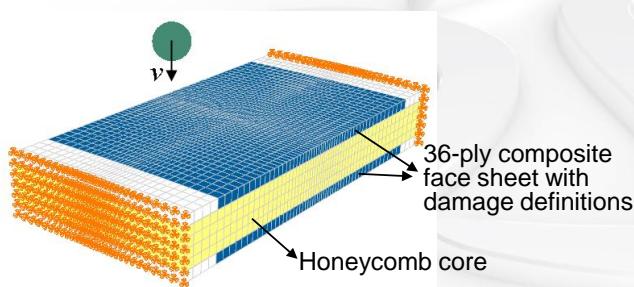
Import of Composite Damage Model

- Example: Damaged sandwich composites**
 - Low speed impact analysis is performed in ABAQUS/Explicit.
 - Damage and associated state are imported into Abaqus/Standard.
 - Residual strength analysis under tensile loading which leads to further damage evolution is performed in Abaqus/Standard.

Impact Analysis – Abaqus/Explicit



Tensile Analysis – Abaqus/Standard



Analysis of Composite Materials with Abaqus

Notes

Notes

Cohesive Behavior

Lecture 10

© DASSAULT SYSTEMES



L10.2

Overview

- Introduction
- Cohesive Element Technology
- Constitutive Response in Cohesive Elements
- Viscous Regularization for Cohesive Elements
- Cohesive Element Examples
- Surface-based Cohesive Behavior
- Element- vs. Surface-based Cohesive Behavior

Note: Appendix 2 contains an in-depth discussion of modeling techniques for cohesive elements using both the interactive and keywords interfaces.

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES

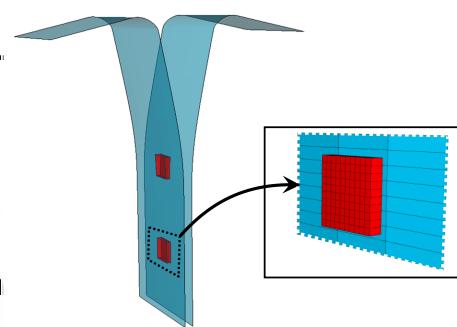


 SIMULIA

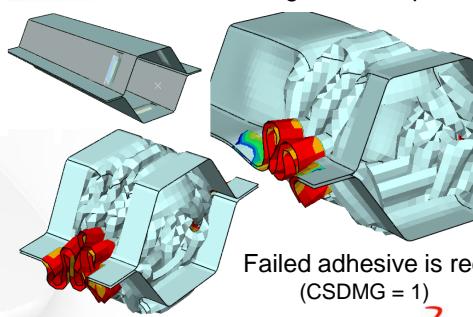
L10.4

Introduction

- Cohesive behavior is useful in modeling adhesives, bonded interfaces, and gaskets.
 - Models separation between two initially bonded surfaces
 - Progressive failure of adhesives
 - Delamination in composites
- Idealize complex fracture mechanisms with a macroscopic “cohesive law,” which relates the traction across the interface to the separation.
- The cohesive behavior can be:
 - Element-based
 - Modeled with cohesive elements
 - Surface-based
 - Modeled with contact pairs in Abaqus/Standard and general contact in Abaqus/Explicit



T-peel analysis: Cohesive elements are used for modeling adhesive patches



Failed adhesive is red
(CSDMG = 1)

 SIMULIA

© DASSAULT SYSTEMES

Introduction

- **Element-based cohesive behavior—cohesive elements**
 - Cohesive elements allow very detailed modeling of adhesive connections, including
 - specification of detailed adhesive material properties, direct control of the connection mesh, modeling of adhesives of finite thickness, etc.
 - Cohesive elements in Abaqus primarily address two classes of problems:
 - Adhesive joints
 - Adhesive layer with finite thickness
 - Typically the bulk material properties are known
 - Delamination
 - Adhesive layer of “zero” thickness
 - Typically the bulk material properties are not known

Introduction

- The constitutive modeling depends on the class of problem:
 - Based on macroscopic properties (stiffness, strength) for adhesive joints
 - Continuum description: any Abaqus material model can be used
 - Modeling technique is relatively straightforward: cohesive layer has finite thickness; standard material models (including damage).
 - The continuum description is not discussed further in this lecture.
 - Based on a traction-separation description for delamination
 - Linear elasticity with damage
 - Modeling technique is less straightforward: typical applications use zero-thickness cohesive elements; non-standard constitutive law
 - **This application is the primary focus of this lecture**

Introduction

- **Surface-based cohesive behavior—cohesive surfaces**
 - This is a simplified and easy way to model cohesive connections, using the traction-separation interface behavior.
 - It offers capabilities that are very similar to cohesive elements modeled with the traction-separation constitutive response.
 - However, it does not require element definitions.
 - In addition, cohesive surfaces can bond anytime contact is established (“sticky” contact)
 - It is primarily intended for situations in which interface thickness is negligibly small.
 - It must be defined as a surface interaction property.
 - Damage for cohesive surfaces is an interaction property, not a material property.
 - The kinematics of cohesive surfaces is different from that of cohesive elements.
 - By default, the initial stiffness of the interface is computed automatically.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



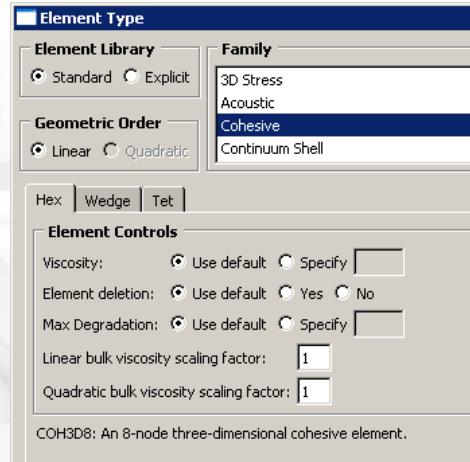
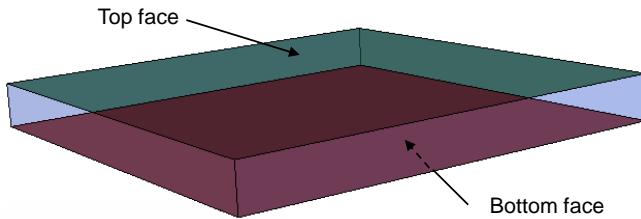
Cohesive Element Technology

© DASSAULT SYSTEMES



Cohesive Element Technology

- **Element types***
 - 3D elements
 - COH3D8
 - COH3D6
 - 2D element
 - COH2D4
 - Axisymmetric element
 - COHAX4
- **These elements can be embedded in a model via**
 - shared nodes or
 - tie constraints.



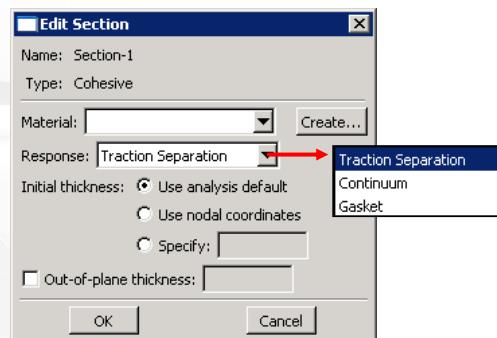
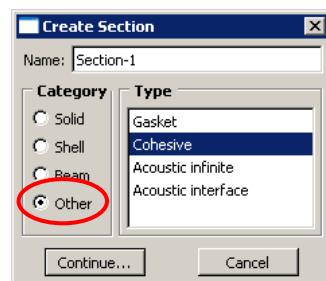
*Cohesive pore pressure elements are also available.

Cohesive Element Technology

- **Element and section definition**

```
*ELEMENT, TYPE = COH3D8
*COHESIVE SECTION, ELSET =....,
RESPONSE = {TRACTION SEPARATION, CONTINUUM,
GASKET },
THICKNESS = { SPECIFIED, GEOMETRY},
MATERIAL = ...
```

Specify thickness in dataline (default is 1.0)



Cohesive Element Technology

- Default thickness of cohesive elements
 - Traction-separation response:
 - Unit thickness
 - Continuum and gasket response
 - Geometric thickness based on nodal coordinates

© DASSAULT SYSTEMES

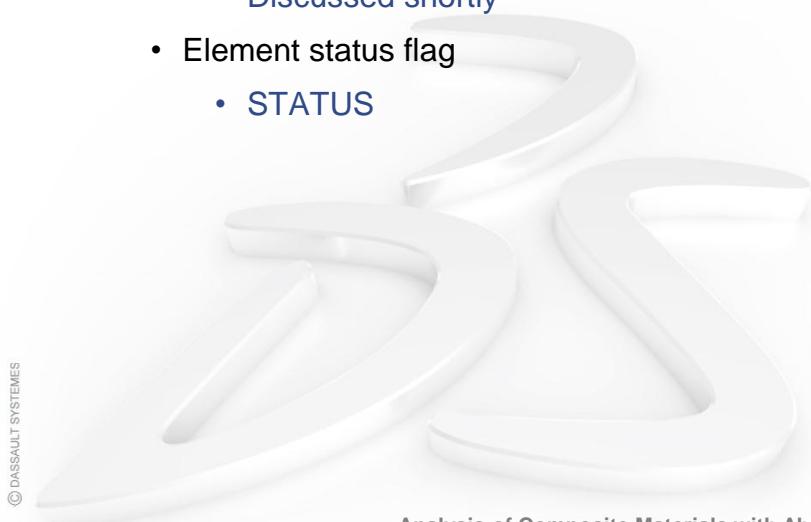


Analysis of Composite Materials with Abaqus

Cohesive Element Technology

- Output variables
 - Scalar damage (i.e., degradation) variable
 - SDEG
 - Variables indicating whether damage initiation criteria met or exceeded
 - Discussed shortly
 - Element status flag
 - STATUS

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Cohesive Element Technology

- Import of cohesive elements

- The combination of Abaqus/Standard and Abaqus/Explicit expands the range of applications for cohesive elements.
- For example, you can simulate the damage in a structure due to an impact event then study the effect of the damage on the structure's load carrying capacity.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



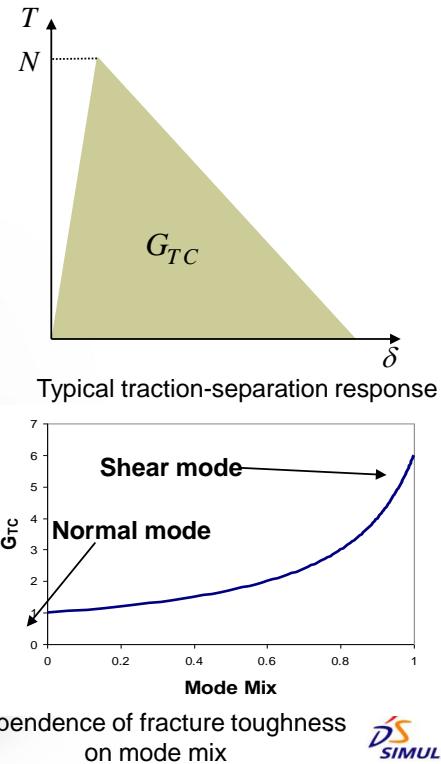
Constitutive Response in Cohesive Elements

© DASSAULT SYSTEMES



Constitutive Response in Cohesive Elements

- Delamination applications
 - Traction separation law
 - Typically characterized by peak strength (N) and fracture energy (G_{TC})
 - Mode dependent
 - Linear elasticity with damage
 - Available in both Abaqus/Standard and Abaqus/Explicit
 - Modeling of damage under the general framework introduced earlier
 - Damage initiation
 - Traction or separation-based criterion
 - Damage evolution
 - Removal of elements

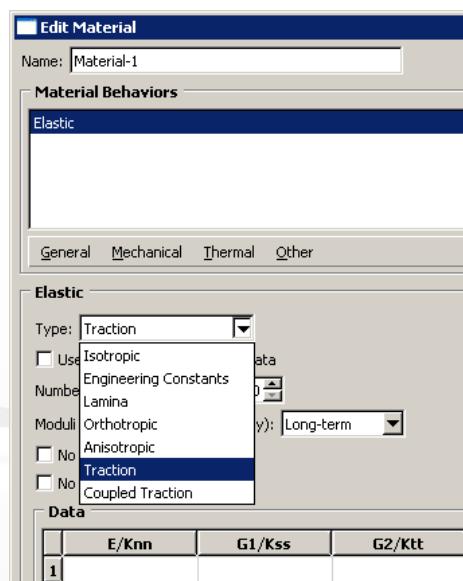


Analysis of Composite Materials with Abaqus



Constitutive Response in Cohesive Elements

- Linear elasticity with damage
 - Linear elasticity
 - Defines behavior before the initiation of damage
 - Relates nominal stress to nominal strain
 - Nominal traction to separation with default choice of unit thickness
 - Uncoupled traction behavior: nominal stress depends only on corresponding nominal strain
 - Coupled traction behavior is more general



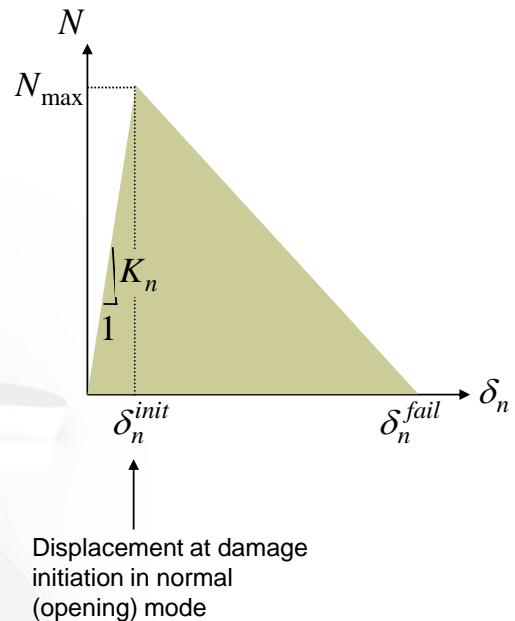
```
*ELASTIC, TYPE = { TRACTION,
COUPLED TRACTION }
```



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- The elastic modulus for the traction separation law should be interpreted as a **penalty stiffness**.
 - For example, for the opening mode:
$$K_n = N_{\max} / \delta_n^{\text{init}}$$
- In Abaqus, nominal stress and strain quantities are used for the traction separation law.
 - If unit thickness is specified for the element, then the nominal strain corresponds to the separation value.
- Elastic response governed by K_n .
 - If you specify a non-unit thickness for the cohesive element, you must scale your data to obtain the correct stiffness K_n . Example on next slide.



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Example: Peel test model

A

Name: Section-1	Data		
Type: Cohesive	E/Knn	G1/Kss	G2/Ktt
Material: elastic	6.9E9	6.9E9	6.9E9
Response: Traction Separation			
Initial thickness: <input type="radio"/> Use analysis default			
<input checked="" type="radio"/> Use nodal coordinates			
<input type="radio"/> Specify: []			
<input type="checkbox"/> Out-of-plane thickness: []			
OK		Cancel	

$$\begin{aligned} N &= E_n \varepsilon_n && \text{Abaqus evaluates this...} \\ &= K_n \delta_n && \dots \text{which is equivalent to this} \\ \varepsilon_n &= \delta_n / h_{\text{eff}} \Rightarrow K_n = E_n / h_{\text{eff}} \end{aligned}$$

Assume separation at initiation $\delta_n^{\text{init}} = 1e-3$ and $N_{\max} = 6.9e9$.

For **model A**: use geometric thickness

$$\begin{aligned} h_{\text{eff}} &= h_{\text{geom}} = 1e-3 \rightarrow \varepsilon_n^{\text{init}} = \delta_n^{\text{init}} / h_{\text{eff}} = 1; \\ N_{\max} &= E_n = 6.9e9 \rightarrow K_n = 6.9e12 \end{aligned}$$

For **model B**: specify unit thickness

$$\begin{aligned} h_{\text{eff}} &= 1 \rightarrow \varepsilon_n^{\text{init}} = \delta_n^{\text{init}} / h_{\text{eff}} = 1e-3; \\ N_{\max} &= 6.9e9 \rightarrow E_n = K_n = 6.9e12 \end{aligned}$$

B

Name: Section-1	Data		
Type: Cohesive	E/Knn	G1/Kss	G2/Ktt
Material: elastic	6.9E12	6.9E12	6.9E12
Response: Traction Separation			
Initial thickness: <input type="radio"/> Use analysis default			
<input checked="" type="radio"/> Use nodal coordinates			
<input type="radio"/> Specify: 1.0			
<input type="checkbox"/> Out-of-plane thickness: []			
OK		Cancel	

Geometric thickness (based on nodal coordinates) of the adhesive $h_{\text{geom}} = 1e-3$



Analysis of Composite Materials with Abaqus

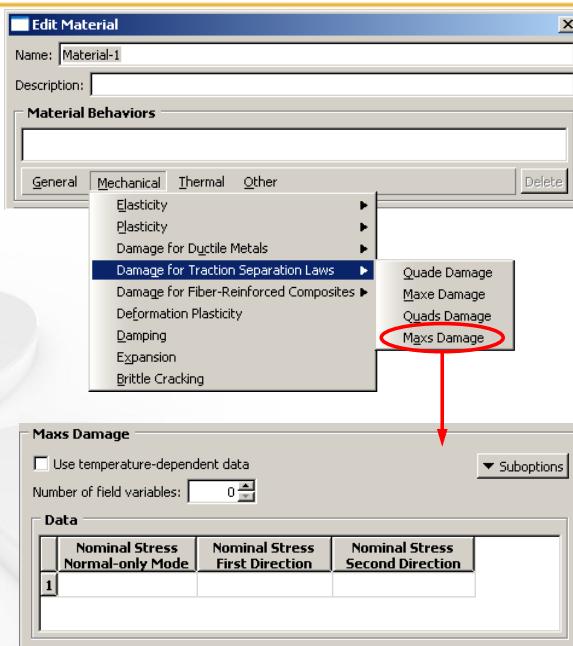
Constitutive Response in Cohesive Elements

- Damage initiation
 - Mixed mode conditions
 - Maximum stress (or strain) criterion:

$$\text{MAX} \left\{ \frac{\langle \sigma_n \rangle}{N_{\max}}, \frac{\sigma_t}{T_{\max}}, \frac{\sigma_s}{S_{\max}} \right\} = 1$$

$$\langle \sigma_n \rangle = \begin{cases} \sigma_n & \text{for } \sigma_n > 0 \\ 0 & \text{for } \sigma_n < 0 \end{cases}$$

- Output:
 - MAXSCRT
 - MAXECRT



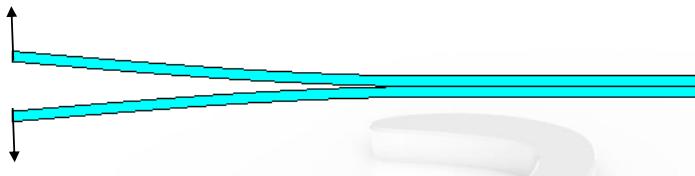
* DAMAGE INITIATION, CRITERION = { MAXS, MAXE }



Analysis of Composite Materials with Abaqus

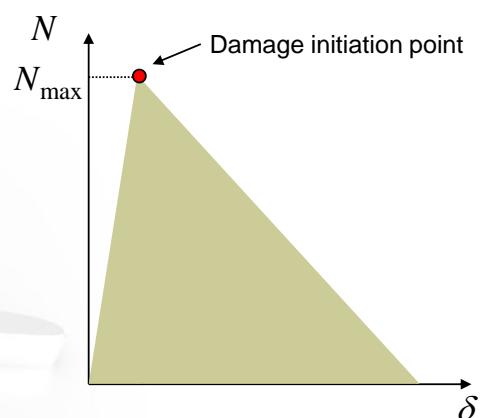
Constitutive Response in Cohesive Elements

- For example, for Mode I (opening mode) the MAXS condition implies damage initiates when $\sigma_n = N_{\max}$.



*Damage initiation,criterion=MAXS
290.0E6, 200.0E6, 200.0E6

N_{\max} T_{\max} S_{\max}



Analysis of Composite Materials with Abaqus

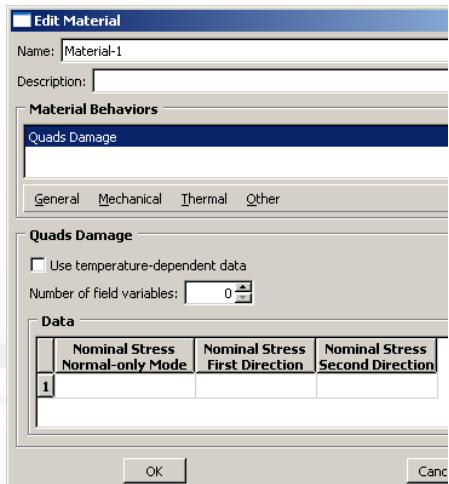


Constitutive Response in Cohesive Elements

- Quadratic stress (or strain) interaction criterion:

$$\left(\frac{\langle \sigma_n \rangle}{N_{\max}} \right)^2 + \left(\frac{\sigma_t}{T_{\max}} \right)^2 + \left(\frac{\sigma_s}{S_{\max}} \right)^2 = 1$$

- No damage initiation under pure compression
- Output:
 - QUADSCRT
 - QUADECRT



* DAMAGE INITIATION,
CRITERION = { QUADS, QUADE }



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Summary of damage initiation criteria

Maximum nominal stress criterion

$$\text{MAX} \left\{ \frac{\langle \sigma_n \rangle}{N_{\max}}, \frac{\sigma_s}{S_{\max}}, \frac{\sigma_t}{T_{\max}} \right\} = 1$$

*DAMAGE INITIATION, CRITERION=MAXS
 $N_{\max}, S_{\max}, T_{\max}$

Quadratic nominal stress criterion

$$\left(\frac{\langle \sigma_n \rangle}{N_{\max}} \right)^2 + \left(\frac{\sigma_s}{S_{\max}} \right)^2 + \left(\frac{\sigma_t}{T_{\max}} \right)^2 = 1$$

*DAMAGE INITIATION, CRITERION=QUADS
 $N_{\max}, S_{\max}, T_{\max}$

σ_n : nominal stress in the pure normal mode

σ_s : nominal stress in the first shear direction

σ_t : nominal stress in the second shear direction

Note: $\varepsilon_n = \frac{\delta_n}{T_o}$, $\varepsilon_s = \frac{\delta_s}{T_o}$, $\varepsilon_t = \frac{\delta_t}{T_o}$

where δ_n , δ_s , and δ_t are components of relative displacement between the top and bottom of the cohesive element; and T_o is the original thickness of the cohesive element.

Maximum nominal strain criterion

$$\text{MAX} \left\{ \frac{\langle \varepsilon_n \rangle}{\varepsilon_n^{\max}}, \frac{\varepsilon_s}{\varepsilon_s^{\max}}, \frac{\varepsilon_t}{\varepsilon_t^{\max}} \right\} = 1$$

*DAMAGE INITIATION, CRITERION=MAXE
 $\varepsilon_n^{\max}, \varepsilon_s^{\max}, \varepsilon_t^{\max}$

Quadratic nominal stress criterion

$$\left(\frac{\langle \varepsilon_n \rangle}{\varepsilon_n^{\max}} \right)^2 + \left(\frac{\varepsilon_s}{\varepsilon_s^{\max}} \right)^2 + \left(\frac{\varepsilon_t}{\varepsilon_t^{\max}} \right)^2 = 1$$

*DAMAGE INITIATION, CRITERION=QUADE
 $\varepsilon_n^{\max}, \varepsilon_s^{\max}, \varepsilon_t^{\max}$

ε_n : nominal strain in the pure normal mode

ε_s : nominal strain in the first shear direction

ε_t : nominal strain in the second shear direction



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- **Damage evolution**
 - Post damage-initiation response defined by:

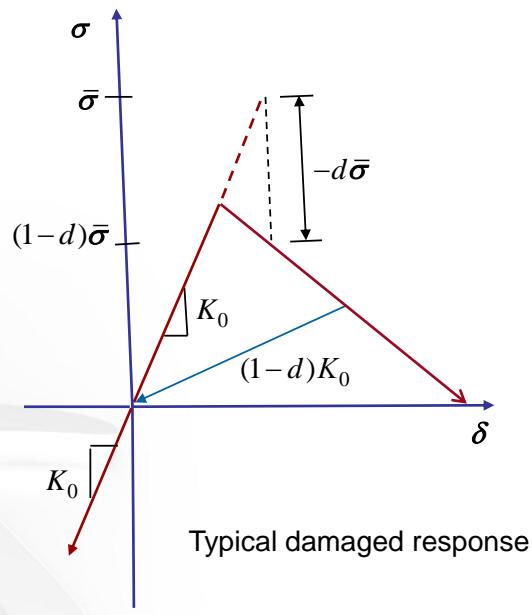
$$\sigma = (1-d)\bar{\sigma}$$

- d is the scalar damage variable

$d = 0$: undamaged

$d = 1$: fully damaged

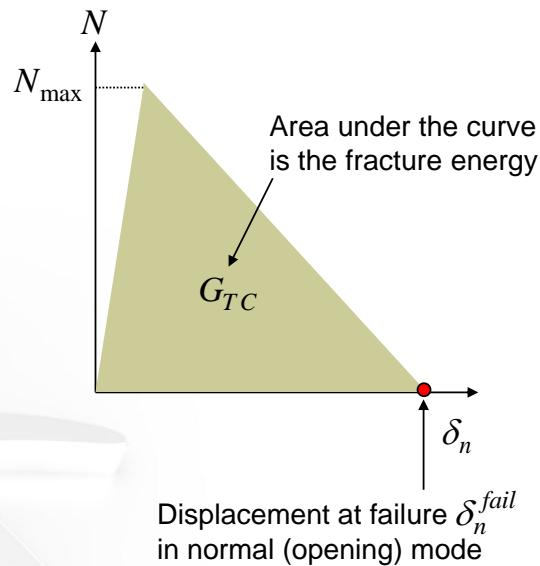
d monotonically increases



Typical damaged response

Constitutive Response in Cohesive Elements

- Damage evolution is based on energy or displacement
 - Specify either the total fracture energy or the post damage-initiation effective displacement at failure
 - May depend on mode mix
 - Mode mix may be defined in terms of energy or traction



Displacement at failure δ_n^{fail} in normal (opening) mode

Constitutive Response in Cohesive Elements

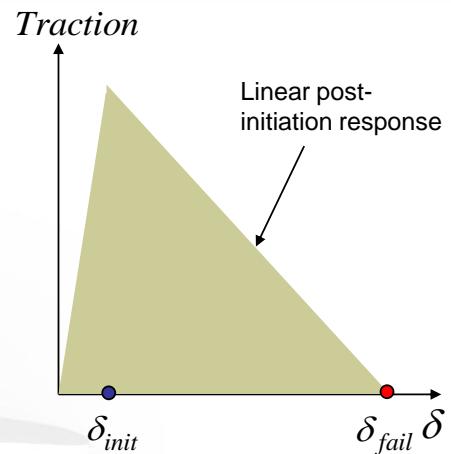
- Displacement-based damage evolution

- Damage is a function of an effective displacement:

$$\delta = \sqrt{(\delta_n)^2 + \delta_s^2 + \delta_t^2}$$

- The post damage-initiation softening response can be either

- Linear
- Exponential
- Tabular



Constitutive Response in Cohesive Elements

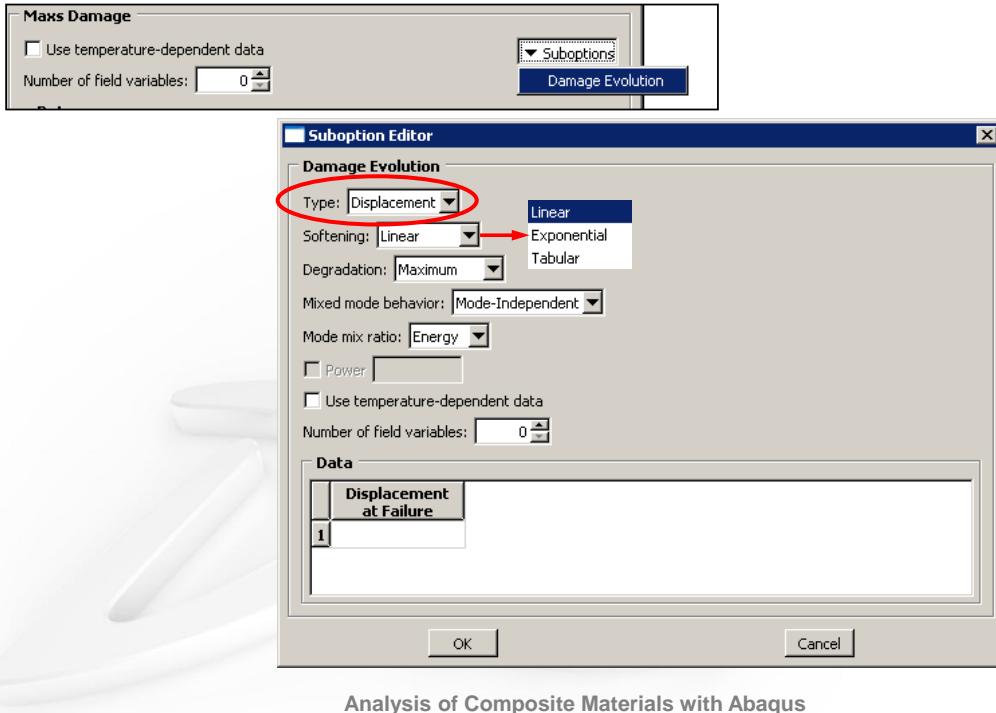
- Keywords interface for displacement-based damage evolution

```
*DAMAGE EVOLUTION, TYPE = DISPLACEMENT,
SOFTENING = { LINEAR | EXPONENTIAL | TABULAR },
MIXED MODE BEHAVIOR = TABULAR
```

- For LINEAR and EXPONENTIAL softening:
 - Specify the effective displacement at complete failure δ_{fail} relative to the effective displacement at initiation δ_{init} .
- For TABULAR softening:
 - Specify the scalar damage variable d directly as a function of $\delta - \delta_{init}$.
- Optionally specify the effective displacement as function of mode mix in tabular form.
 - Abaqus assumes that the damage evolution is mode independent otherwise.

Constitutive Response in Cohesive Elements

- Abaqus/CAE interface for displacement-based damage evolution



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Energy-based damage evolution

- The fracture energy can be defined as a function of mode mix using either a tabular form or one of two analytical forms:

- Power law

$$\left(\frac{G_I}{G_{IC}}\right)^\alpha + \left(\frac{G_{II}}{G_{IIC}}\right)^\alpha + \left(\frac{G_{III}}{G_{III}}\right)^\alpha = 1$$

- BK (Benzeggagh-Kenane)

$$G_{IC} + (G_{IIC} - G_{IC}) \left(\frac{G_{shear}}{G_T} \right)^\eta = G_{TC}$$

where $G_{shear} = G_{II} + G_{III}$

$$G_T = G_I + G_{shear}$$

For isotropic failure ($G_{IC} = G_{IIC}$), the response is insensitive to the value of η .



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Keywords interface for energy-based damage evolution

```
*DAMAGE EVOLUTION, TYPE = ENERGY,
SOFTENING = { LINEAR | EXPONENTIAL },
MIXED MODE BEHAVIOR = { TABULAR | POWER LAW | BK },
POWER = value
```

- Specify fracture energy as function of mode mix in tabular form, or
- Specify the fracture energy in pure normal and shear deformation modes and choose either the POWER LAW or the BK mixed mode behavior

© DASSAULT SYSTEMES

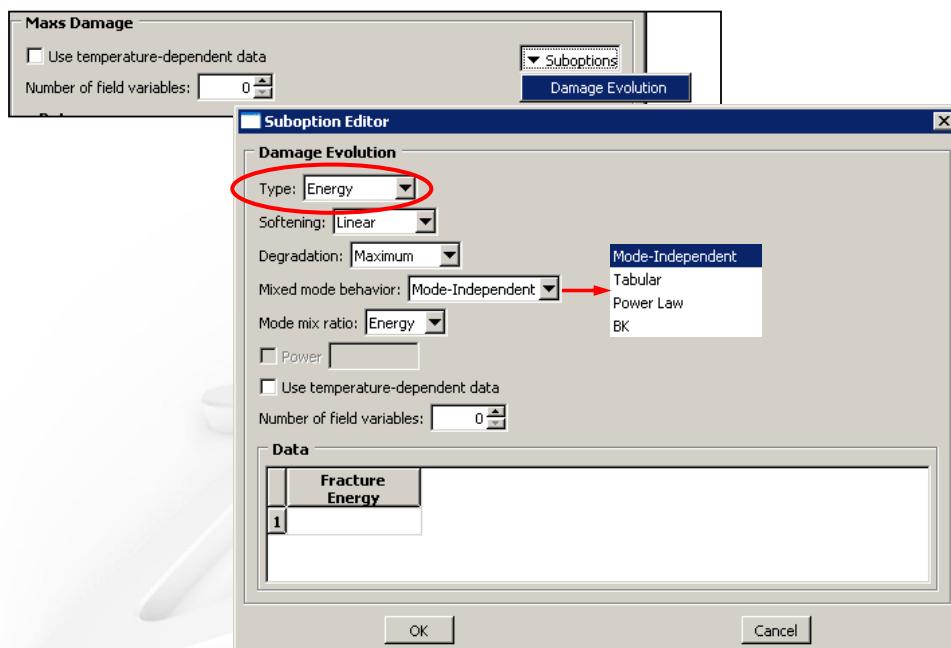


Analysis of Composite Materials with Abaqus

L10.30

Constitutive Response in Cohesive Elements

- Abaqus/CAE interface for energy-based damage evolution



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Example

- The preceding discussion was very general in the sense that the full range of options for modeling the constitutive response of cohesive elements was presented.
- In the simplest case, Abaqus requires that you input the adhesive thickness h_{eff} and 10 material parameters:

`*Elastic, type=traction`

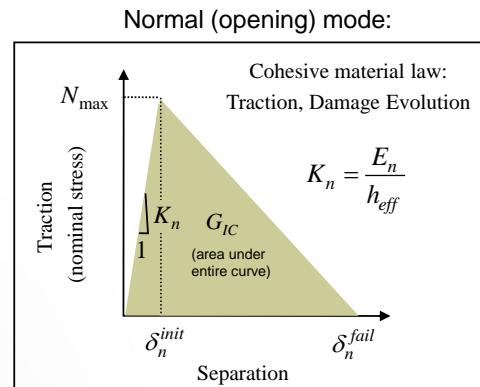
`En, Et, Es`

`*Damage initiation, criterion = maxs`

`Nmax, Tmax, Smax`

`*Damage evolution, type=energy, mixed mode behavior=bk, power=η` ABAQUS Users' Conference, Stockholm, 2005.

`GIC, GIIC, GIIIC`



What do you do when you only have 1 property and the adhesive thickness is essentially zero?

Constitutive Response in Cohesive Elements

- Example (cont'd)

- Common case: you know G_{TC} for the surface bond.

- Assume isotropic behavior

$$G_{IC} = G_{IIC} = G_{IIIC} = G_{TC}$$

- For MIXED MODE BEHAVIOR = BK, this makes the response independent of $η$ term, so set $η$ = any valid input value (e.g., 1.0)

- Bond thickness is essentially zero

- Specify the cohesive section property thickness $h_{eff} = 1.0$

⇒ Nominal strains = separation; elastic moduli = stiffness

- Isotropy also implies the following:

$$E_n = E_t = E_s = E_{eff} \quad (=K_{eff} \text{ since we chose } h_{eff} = 1.0)$$

$$N_{max} = T_{max} = S_{max} = T_{ult}$$

Constitutive Response in Cohesive Elements

- Example (cont'd)

- Introduce concept of damage initiation ratio:

$$\delta_{ratio} = \delta_{init} / \delta_{fail}, \quad \text{where } 0 < \delta_{ratio} < 1.$$

- Use G_C and equation of a triangle to relate back to K_{eff} and T_{ult} :

$$K_{eff} = \frac{2G_{TC}}{\delta_{ratio} \delta_{fail}^2} \quad T_{ult} = \frac{2G_{TC}}{\delta_{fail}}$$

- The problem now reduces to two penalty terms: δ_{fail} and δ_{ratio} .
 - Assume $\delta_{ratio} = 1/2$.
 - Choose δ_{fail} as a fraction of the typical cohesive element mesh size.
 - For example, use $\delta_{fail} = 0.050 \times$ typical cohesive element size as a starting point.

© DASSAULT SYSTEMES



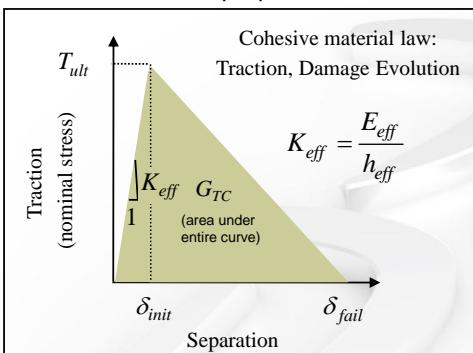
Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Example (cont'd)

- Thus, after choosing the two penalty terms, a single (effective) traction-separation law applies to all modes (normal + shear):

Effective properties:



```
*Cohesive section, thickness=SPECIFIED, ...
1.0,
:
:
*Elastic, type=TRACTION
K_eff, K_eff, K_eff
*Damage initiation, criterion = MAXS
T_ult, T_ult, T_ult
*Damage evolution, type=ENERGY,
mixed mode behavior=BK, power=1
G_TC, G_TC , G_TC
```

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Constitutive Response in Cohesive Elements

- Example (cont'd)

- What if the response is dynamic? What about the density?
 - The density of the cohesive layer should also be considered a penalty quantity.
 - For Abaqus/Explicit, the effective density should not adversely affect the stable time increment. Diehl suggests the following rule:

$$\rho_{eff} = E_{eff} \cdot \left(\frac{\Delta t_{stable}}{f_{t2D} h_{eff}} \right)^2$$

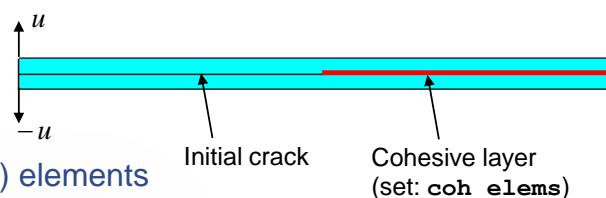
- Δt_{stable} = stable time increment without cohesive elements in the model
- $f_{t2D} = 0.32213$ (for cohesive elements whose original nodal coordinates relate to zero element thickness)

- The Abaqus Analysis User's Manual provides additional guidelines for determining a cohesive element density that minimizes the effect on the stable time increment in Abaqus/Explicit.

Constitutive Response in Cohesive Elements

- Example: Double-cantilever beam (DCB)

- Alfano and Crisfield (2001)
 - Pure Mode I
 - Displacement control
 - Analyzed using 2D (CPE4I) elements
 - Delamination assumed to occur along a straight line
 - Beams: Orthotropic material
 - Cohesive layer: Traction-separation with damage
 - The cohesive properties are given next slide.



Constitutive Response in Cohesive Elements

- Properties: adhesive
 - Interactive interface

The screenshot displays several Abaqus dialog boxes related to cohesive element properties:

- Edit Material**: Shows a material named "cohesive". A red arrow points from the "Name" field in this dialog to the "Material" dropdown in the "Edit Section" dialog.
- Edit Section**: Shows a cohesive section named "cohesive" with "Traction Separation" response and "Elastic" material type.
- Edit Material**: Shows a material named "cohesive" with "Quads Damage" behavior selected. A red circle highlights the "Quads Damage" tab.
- Suboption Editor**: Shows damage evolution settings: Type=Energy, Softening=Linear, Degradation=Maximum, Mixed mode behavior=BK, Mode mix ratio=Energy, Power=2.284. A red circle highlights the "Damage Evolution" tab.

Below the dialogs, the text "Analysis of Composite Materials with Abaqus" is visible. The Dassault Systèmes SIMULIA logo is in the bottom right corner.

Constitutive Response in Cohesive Elements

- Keywords interface

```
*COHESIVE SECTION, ELSET=coh_elems, MATERIAL=cohesive,
RESPONSE=TRACTION SEPARATION
, 0.02
*MATERIAL, NAME=cohesive
*ELASTIC, TYPE=TRACTION
5.7e14, 5.7e14 , 5.7e14
*DAMAGE INITIATION, CRITERION=QUADS
5.7e7, 5.7e7 , 5.7e7
*DAMAGE EVOLUTION, TYPE=ENERGY, MIXED MODE BEHAVIOR=BK, POWER=2.284
280, 280 , 280
```

- Note:** More details on modeling of this problem using cohesive elements are discussed in Appendix 2 “Cohesive Element Modeling Techniques;” the relevant results will be discussed later in section “Surface-based Cohesive Behavior.”

Viscous Regularization for Cohesive Elements

© DASSAULT SYSTEMES

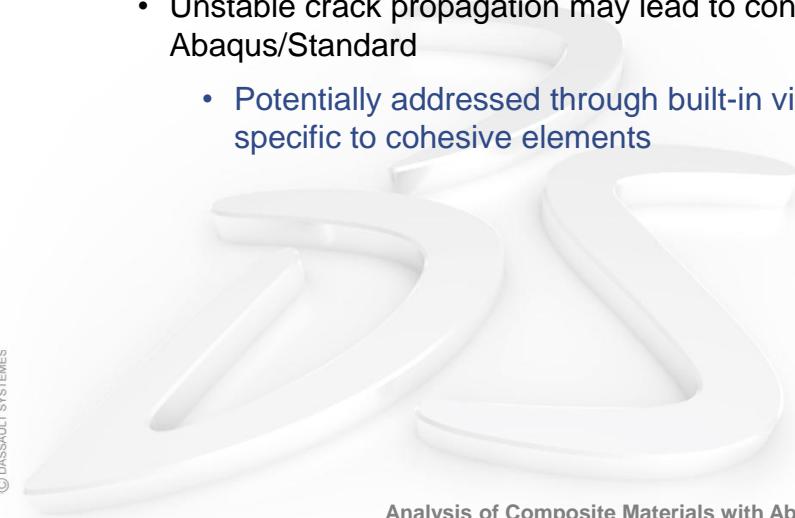


L10.40

Viscous Regularization for Cohesive Elements

- Cohesive elements have the potential to cause numerical difficulties in the following cases
 - Stiff cohesive behavior may lead to reduced maximum stable time increment in Abaqus/Explicit
 - Potentially addressed through selective mass scaling
 - Unstable crack propagation may lead to convergence difficulties in Abaqus/Standard
 - Potentially addressed through built-in viscous regularization option specific to cohesive elements

© DASSAULT SYSTEMES



Viscous Regularization for Cohesive Elements

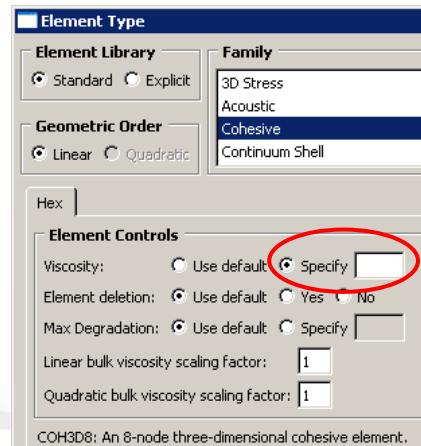
- User interface for viscous regularization

***COHESIVE SECTION, CONTROLS = control1**
***SECTION CONTROLS, NAME = control1,**
VISCOSITY = factor

- Add-on transverse shear stiffness may provide additional stability

***COHESIVE SECTION**

***TRANSVERSE SHEAR STIFFNESS**



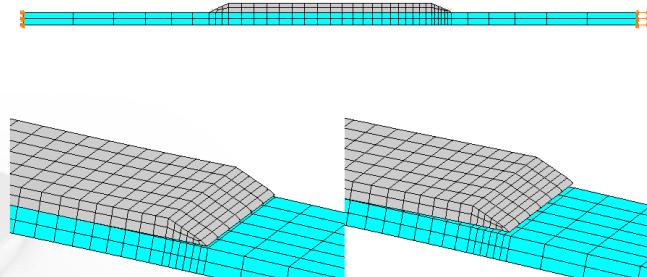
- Output

- Energy associated with viscous regularization: ALLCD
- More details on viscous regularization are discussed in Appendix 2

Cohesive Element Examples

Cohesive Element Examples

- **Composite components in aerospace structures**
(Courtesy: NASA)
 - Stress concentrations around stiffener terminations and flanges
 - Residual thermal strains at the interface at room temperature
 - Analysis of the effects of residual strains on skin/stiffener debonding
 - Delamination initiation and propagation



Beginning of separation After separation
Abaqus/Standard simulation of skin/stiffener debonding

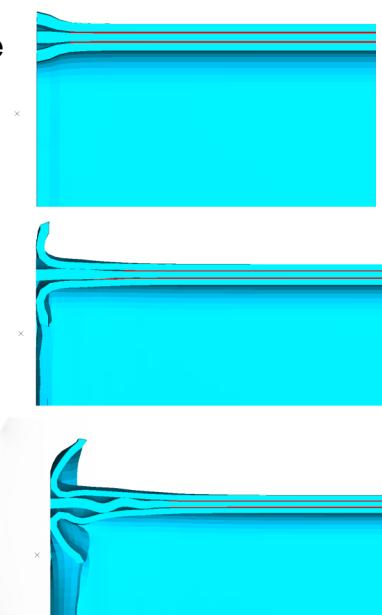
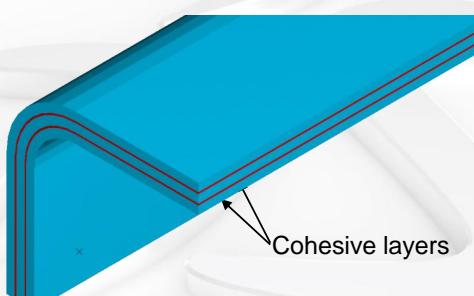
Example Problem 1.4.5



Analysis of Composite Materials with Abaqus

Cohesive Element Examples

- **Delamination of a composite**
 - This model is a representative of composite delamination.
 - It comprises 3 layers of composite with adhesive layers applied between composite layers.
 - The composite delaminates under the impact of a heavy mass displayed in light greenish shade in the animation.



Analysis of Composite Materials with Abaqus

Cohesive Element Examples

• Lap joint analysis

- Lap joints are created by laying one material on top of another and bonding them together
 - For example: bonding materials using an adhesive or fasteners
- Connection type influences characteristic of joint
 - Adhesive connection (covered here)
 - Compliance and thickness of adhesive
 - Fastener connection
 - Stiffness of fastener

© DASSAULT SYSTEMES

Material A

Material B

Single-Lap Joint



Analysis of Composite Materials with Abaqus

Cohesive Element Examples

- Mesh of the lap joint modeled using both solid and shell elements
 - 2536 C3D8I and S4R elements
 - Transition from solid to shell elements is accomplished using the surface-based shell-to-solid coupling constraint.
 - 2116 COH3D8 elements
- Linear elastic material is used for the cohesive layer
 - 100 psi modulus of elasticity (compared to 10.E6 for aluminum)
 - 0.4 Poisson's ratio

© DASSAULT SYSTEMES

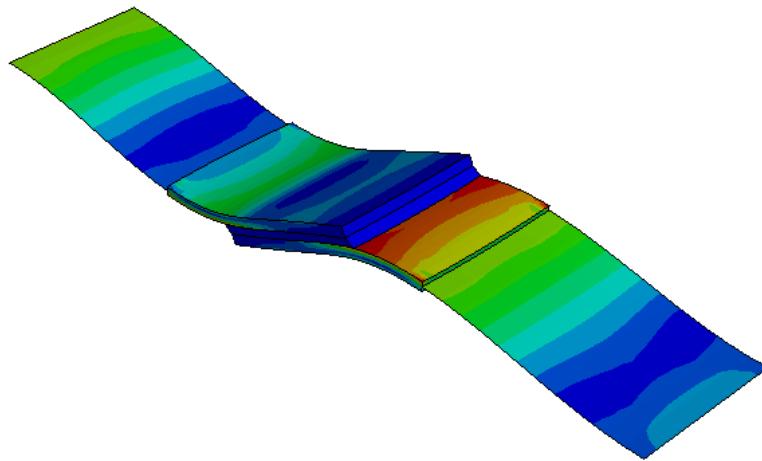
Note that the cohesive element layer is initially of zero thickness, and the mesh density is finer than the connected regions



Analysis of Composite Materials with Abaqus

Cohesive Element Examples

- The evolution of deformation of the lap joint with a compliant adhesive layer



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior



© DASSAULT SYSTEMES



Surface-based Cohesive Behavior

- Surface-based cohesive behavior provides a simplified way to model cohesive connections with negligibly small interface thicknesses using the traction-separation constitutive model.
 - It can also model “sticky” contact (surfaces can bond after coming into contact).
 - The cohesive surface behavior can be defined for general contact in Abaqus/Explicit and contact pairs in Abaqus/Standard (with the exception of the finite-sliding, surface-to-surface formulation).
 - Cohesive surface behavior is defined as a surface interaction property.
 - To prevent overconstraints in Abaqus/Explicit, a pure master-slave formulation is enforced for surfaces with cohesive behavior.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- User interface

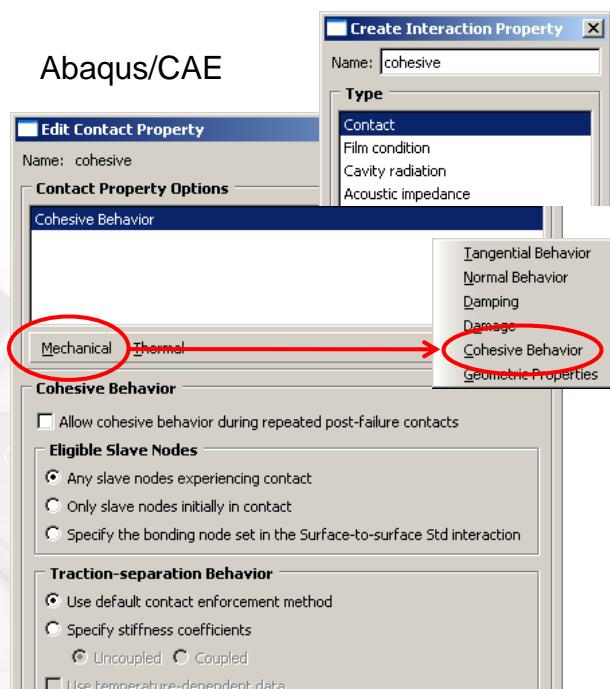
Abaqus/Standard

```
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR
...
*CONTACT PAIR, INTERACTION=cohesive
surface1, surface2
```

Abaqus/Explicit

```
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR
...
*CONTACT
*CONTACT PROPERTY ASSIGNMENT
surface1, surface2, cohesive
```

Abaqus/CAE



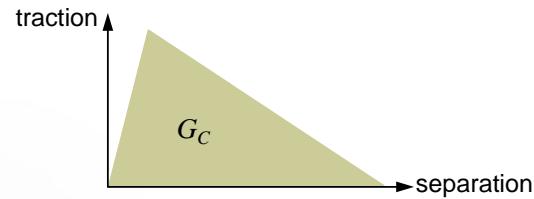
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- The formulae and laws that govern surface-based cohesive behavior are very similar to those used for cohesive elements with traction-separation behavior:
 - linear elastic traction-separation,
 - damage initiation criteria, and
 - damage evolution laws.
- However, it is important to recognize that damage in surface-based cohesive behavior is an interaction property, not a material property.
- Traction and separation are interpreted differently for cohesive elements and cohesive surfaces:



© DASSAULT SYSTEMES

	Cohesive elements	Cohesive surfaces
separation	Relative displacement (δ) between the top and bottom of the cohesive layer Nominal strain (ϵ) = $\frac{\delta}{T_o}$	Contact separation (δ)
traction	Nominal stress (σ)	Contact stress (t) = $\frac{F}{A}$

Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- Linear elastic traction-separation behavior**
 - Relates normal and shear stresses to the normal and shear separations across the interface before the initiation of damage.
 - By default, elastic properties are based on underlying element stiffness.
 - Can optionally specify the properties.
 - Recall this specification is required for cohesive elements.
 - The traction-separation behavior can be uncoupled (default) or coupled.

*COHESIVE BEHAVIOR, TYPE= { UNCOUPLED, COUPLED }

Optional data line to specify Knn, Kss, Ktt

Traction-separation Behavior

<input type="radio"/> Use default contact enforcement method		
<input checked="" type="radio"/> Specify stiffness coefficients		
<input checked="" type="radio"/> Uncoupled	<input type="radio"/> Coupled	
<input type="checkbox"/> Use temperature-dependent data		
Number of field variables: <input type="text" value="0"/>		
Knn	Kss	Ktt

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

• Controlling the cohersed nodes

- The slave nodes to which cohesive behavior is applied can be controlled to define a wider range of cohesive interactions: Can include:
 - All slave nodes
 - Only slave nodes initially in contact
 - Initially bonded node set

① Applying cohesive behavior to all slave nodes (default)

- Cohesive constraint forces potentially act on all nodes of the slave surface.
- Slave nodes that are not initially contacting the master surface can also experience cohesive forces if they contact the master surface during the analysis.

***COHESIVE BEHAVIOR,
ELIGIBILITY = CURRENT CONTACTS**

Eligible Slave Nodes
<input checked="" type="radio"/> Any slave nodes experiencing contact
<input type="radio"/> Only slave nodes initially in contact
<input type="radio"/> Specify the bonding node set in the Surface-to-surface Std interaction



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

② Applying cohesive behavior only to slave nodes initially in contact

- Restrict cohesive behavior to only those slave nodes that are in contact with the master surface at the start of a step.
- Any new contact that occurs during the step will not experience cohesive constraint forces.
 - Only compressive contact is modeled for new contact.

***COHESIVE BEHAVIOR,
ELIGIBILITY = ORIGINAL CONTACTS**

Eligible Slave Nodes
<input type="radio"/> Any slave nodes experiencing contact
<input checked="" type="radio"/> Only slave nodes initially in contact
<input type="radio"/> Specify the bonding node set in the Surface-to-surface Std interaction



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

3 Applying cohesive behavior only to an initially bonded node set (Abaqus/Standard only)

- Restrict cohesive behavior to a subset of slave nodes defined using *INITIAL CONDITIONS, TYPE=CONTACT.
- All slave nodes outside of this set will experience only compressive contact forces during the analysis.
 - This method is particularly useful for modeling crack propagation along an existing fault line.

*COHESIVE BEHAVIOR,
ELIGIBILITY = SPECIFIED CONTACTS

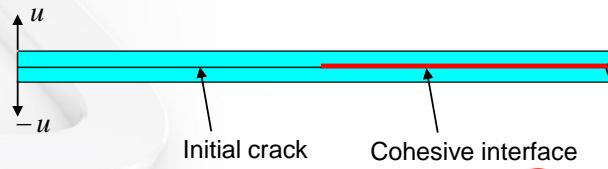
Eligible Slave Nodes

Any slave nodes experiencing contact
 Only slave nodes initially in contact
 Specify the bonding node set in the Surface-to-surface Std interaction

Surface-based Cohesive Behavior

• Example: Double cantilever beam (DCB)

- Analyze debonding of the DCB model using the surface-based cohesive behavior in Abaqus/Standard.
- To model debonding using surface-based cohesive behavior,
 - you must define:
 - 1 contact pairs and initially bonded crack surfaces;
 - 2 the traction-separation behavior;
 - 3 the damage initiation criterion; and
 - 4 the damage evolution.
 - You may also
 - 5 specify viscous regularization to facilitate solution convergence in Abaqus/Standard.
- **Note:** Steps 3, 4, and 5, will be covered later in this lecture.



Surface-based Cohesive Behavior

- 1 Define contact pairs and initially bonded crack surfaces
 - The initially bonded portion of the slave surface (i.e., node set `bond`) is identified with the *INITIAL CONDITIONS, TYPE=CONTACT option.

Note: Frictionless contact is assumed.

```

*NSET, NSET=bond, GENERATE
1, 121, 1
*SURFACE, NAME=TopSurf
_TopBeam_S1, S1
*SURFACE, NAME=BotSurf
_BotBeam_S1, S1
*CONTACT PAIR, INTER=cohesive
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
  
```

slave surface master surface a list of slave nodes that are initially bonded

© DASSAULT SYSTEMES SIMULIA

Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- 2 Define traction-separation behavior
 - In this model, the cohesive behavior is only enforced for the node set `bond`.
 - Use the ELIGIBILITY=SPECIFIED CONTACTS parameter to enforce this behavior.
 - Recall the default elastic properties are based on underlying element stiffness. Here we specify the properties.

...

```

*CONTACT PAIR, INTER=cohesive
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR,
ELIGIBILITY=SPECIFIED CONTACTS
5.7e14, 5.7e14, 5.7e14 ← Optional
  
```

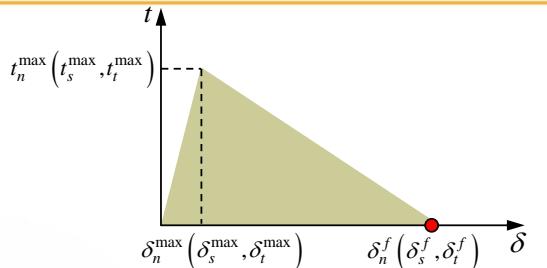
K_n , K_s , and K_t ; normal and tangential stiffness components

© DASSAULT SYSTEMES SIMULIA

Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- Damage modeling for cohesive surfaces
 - Damage of the traction-separation response for cohesive surfaces is defined within the same general framework used for cohesive elements.
 - The difference between the two approaches is that for cohesive surfaces damage is specified as part of the contact interaction properties.



t_n^{\max} , t_s^{\max} , and t_t^{\max} :
peak values of the contact stress

δ_n^{\max} , δ_s^{\max} , and δ_t^{\max} :
peak values of the contact separation

δ_n^f , δ_s^f , and δ_t^f :
separations at failure

Surface-based Cohesive Behavior

- User interface

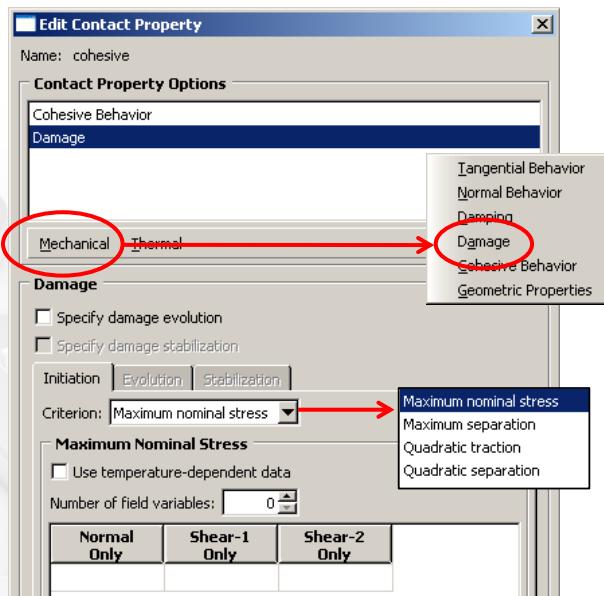
Abaqus/CAE

Abaqus/Standard

*SURFACE INTERACTION, NAME=cohesive
 *COHESIVE BEHAVIOR
 *DAMAGE INITIATION
 *DAMAGE EVOLUTION
 *CONTACT PAIR, INTERACTION=cohesive
 surface1, surface2

Abaqus/Explicit

*SURFACE INTERACTION, NAME=cohesive
 *COHESIVE BEHAVIOR
 *DAMAGE INITIATION
 *DAMAGE EVOLUTION
 *CONTACT
 *CONTACT PROPERTY ASSIGNMENT
 surface1, surface2, cohesive



Surface-based Cohesive Behavior

- Damage initiation criteria

Maximum stress criterion

$$\text{MAX} \left\{ \frac{\langle t_n \rangle}{t_n^{\max}}, \frac{t_s}{t_s^{\max}}, \frac{t_t}{t_t^{\max}} \right\} = 1$$

*DAMAGE INITIATION, CRITERION=MAXS
 $t_n^{\max}, t_s^{\max}, t_t^{\max}$

Quadratic stress criterion

$$\left(\frac{\langle t_n \rangle}{t_n^{\max}} \right)^2 + \left(\frac{t_s}{t_s^{\max}} \right)^2 + \left(\frac{t_t}{t_t^{\max}} \right)^2 = 1$$

*DAMAGE INITIATION, CRITERION=QUADS
 $t_n^{\max}, t_s^{\max}, t_t^{\max}$

t_n : normal contact stress in the pure normal mode
 t_s : shear contact stress along the first shear direction
 t_t : shear contact stress along the second shear direction

Maximum separation criterion

$$\text{MAX} \left\{ \frac{\langle \delta_n \rangle}{\delta_n^{\max}}, \frac{\delta_s}{\delta_s^{\max}}, \frac{\delta_t}{\delta_t^{\max}} \right\} = 1$$

*DAMAGE INITIATION, CRITERION=MAXU
 $\delta_n^{\max}, \delta_s^{\max}, \delta_t^{\max}$

Quadratic separation criterion

$$\left(\frac{\langle \delta_n \rangle}{\delta_n^{\max}} \right)^2 + \left(\frac{\delta_s}{\delta_s^{\max}} \right)^2 + \left(\frac{\delta_t}{\delta_t^{\max}} \right)^2 = 1$$

*DAMAGE INITIATION, CRITERION=QUADU
 $\delta_n^{\max}, \delta_s^{\max}, \delta_t^{\max}$

δ_n : separation in the pure normal mode
 δ_s : separation in the first shear direction
 δ_t : separation in the second shear direction

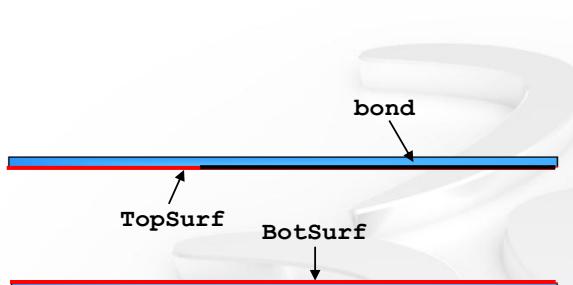
Note: Recall the damage initiation criteria for the cohesive elements: if the initial constitutive thickness $T_o = 1$, then $\varepsilon = \delta T_o = \delta$. In this case, the separation measures for both approaches are exactly the same.

Surface-based Cohesive Behavior

- Example: Double cantilever beam

③ Define the damage initiation criterion

- The quadratic stress criterion is specified for this problem.

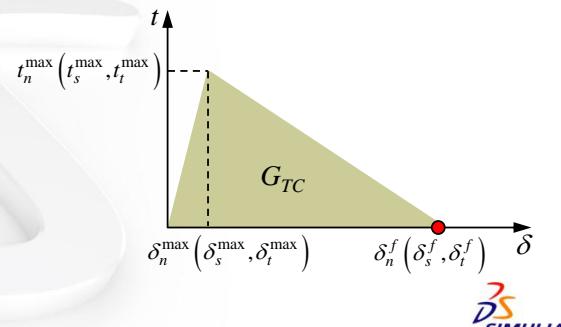


```
...
*CONTACT PAIR, INTER=cohesive
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR,
ELIGIBILITY=SPECIFIED CONTACTS
5.7e14, 5.7e14, 5.7e14
*DAMAGE INITIATION, CRITERION=QUADS
5.7e7, 5.7e7, 5.7e7
 $t_n^{\max}$   $t_s^{\max}$   $t_t^{\max}$ 
```

Surface-based Cohesive Behavior

- **Damage evolution**

- For surface-based cohesive behavior, damage evolution describes the degradation of the **cohesive** stiffness.
- In contrast, for cohesive elements damage evolution describes the degradation of the **material** stiffness.
- Damage evolution can be based on energy or separation (same as for cohesive elements).
 - Specify either the total fracture energy (a property of the cohesive interaction) or the post damage-initiation effective separation at failure.
 - May depend on mode mix
 - Mode mix may be defined in terms of energy or traction



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

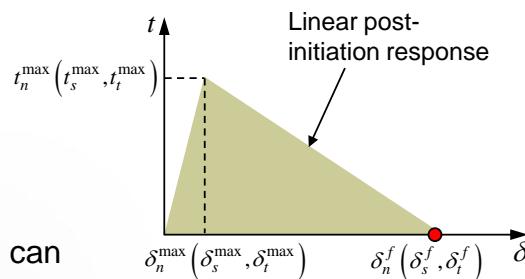
Surface-based Cohesive Behavior

- **Separation-based damage evolution**

- Damage is a function of an effective separation:

$$\delta \equiv \sqrt{(\delta_s)^2 + \delta_s^2 + \delta_t^2}$$

- As with cohesive elements, the post damage-initiation softening response can be either:
 - Linear
 - Exponential
 - Tabular



© DASSAULT SYSTEMES

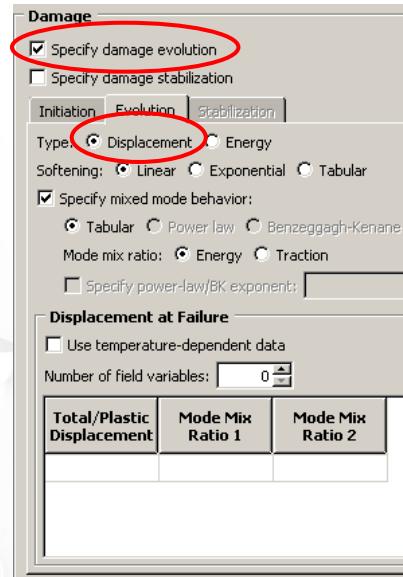
Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- Separation-based damage evolution (cont'd)

- Usage:

*DAMAGE EVOLUTION, TYPE = **DISPLACEMENT**,
SOFTENING = { **LINEAR** | **EXPONENTIAL** | **TABULAR** },
MIXED MODE BEHAVIOR = **TABULAR**



Analysis of Composite Materials with Abaqus

Surface-based Cohesive Behavior

- Energy-based damage evolution

- As with cohesive elements, the energy-based damage evolution criterion can be defined as a function of mode mix using either a tabular form or one of two analytical forms:

Power law

$$\left(\frac{G_I}{G_{IC}}\right)^\alpha + \left(\frac{G_{II}}{G_{IIC}}\right)^\alpha + \left(\frac{G_{III}}{G_{IIIC}}\right)^\alpha = 1$$

Benzeggagh-Kenane (BK)

$$G_{IC} + (G_{IIC} - G_{IC}) \left(\frac{G_{shear}}{G_T} \right)^\eta = G_{TC}$$

where $G_{shear} = G_{II} + G_{III}$

$$G_T = G_I + G_{shear}$$

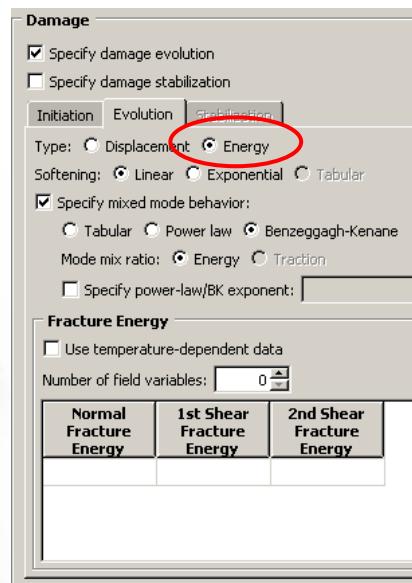


Surface-based Cohesive Behavior

- Energy-based damage evolution (cont'd)

- Usage:

*DAMAGE EVOLUTION, TYPE = ENERGY,
 SOFTENING = { LINEAR | EXPONENTIAL },
 MIXED MODE BEHAVIOR = { TABULAR | POWER LAW | BK },
 POWER = value



Analysis of Composite Materials with Abaqus

© DASSAULT SYSTEMES

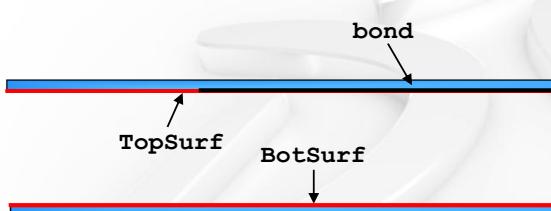
Surface-based Cohesive Behavior

- Example: Double cantilever beam

4 Define damage evolution

- The energy-based damage evolution based on the BK mixed mode behavior is specified.

$$G_{IC} + (G_{IIc} - G_{IC}) \left(\frac{G_{shear}}{G_T} \right)^\eta = G_{TC}$$



```
...
*CONTACT PAIR, INTER=cohesive
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR,
  ELIGIBILITY=SPECIFIED CONTACTS
  5.7e14, 5.7e14, 5.7e14
*DAMAGE INITIATION, CRITERION=QUADS
  5.7e7, 5.7e7, 5.7e7
*DAMAGE EVOLUTION, TYPE=ENERGY,
  MIXED MODE BEHAVIOR=BK, POWER=2.284
  280.0, 280.0, 280.0
    ↑
    η
  GIC   GIIc   GIIIc
```



Analysis of Composite Materials with Abaqus

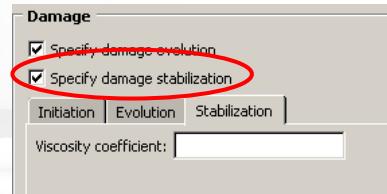
© DASSAULT SYSTEMES

Surface-based Cohesive Behavior

- **Viscous regularization**

- Can be specified to facilitate solution convergence in Abaqus/Standard for surface-based cohesive behavior when stiffness degradation occurs.
- Output:
 - Energy associated with viscous regularization: ALLCD

© DASSAULT SYSTEMES

***DAMAGE STABILIZATION**

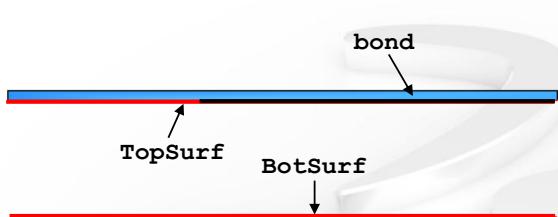
Analysis of Composite Materials with Abaqus



Surface-based Cohesive Behavior

- **Example: Double cantilever beam**

- Specify a viscosity coefficient for the cohesive surface behavior



```

...
*CONTACT PAIR, INTER=cohesive
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR,
ELIGIBILITY=SPECIFIED CONTACTS
5.7e14, 5.7e14, 5.7e14
*DAMAGE INITIATION, CRITERION=QUADS
5.7e7, 5.7e7, 5.7e7
*DAMAGE EVOLUTION, TYPE=ENERGY,
MIXED MODE BEHAVIOR=BK, POWER=2.284
280.0, 280.0, 280.0
*DAMAGE STABILIZATION
1.e-5
viscosity coefficient,  $\mu$ 

```

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Surface-based Cohesive Behavior

- Example: Double cantilever beam
 - Summary of the input for the traction-separation response

Cohesive elements

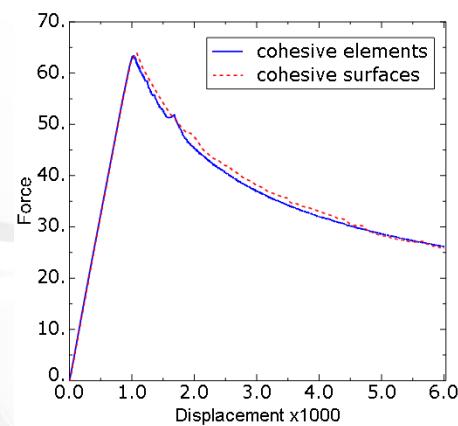
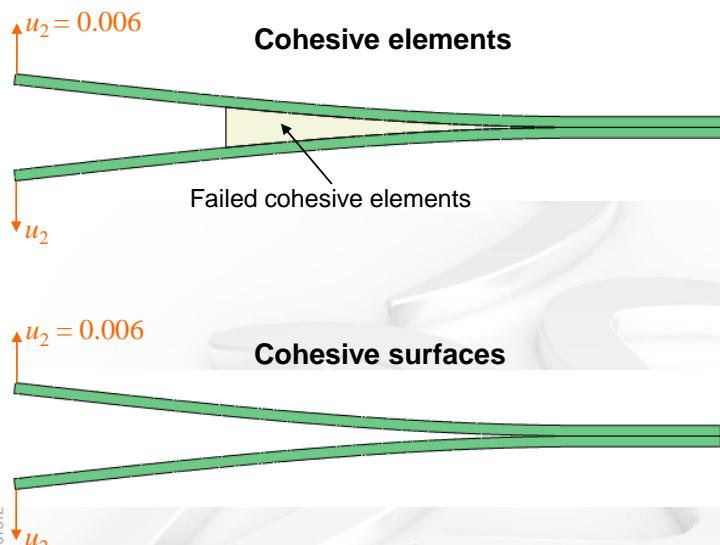
```
*COHESIVE SECTION, MATERIAL=cohesive,
RESPONSE=TRACTION SEPARATION,
ELSET=coh_elems, CONTROLS=visco
, 0.02
*MATERIAL, NAME=cohesive
*ELASTIC, TYPE=TRACTION
5.7e14, 5.7e14, 5.7e14
*DAMAGE INITIATION, CRITERION=QUADS
5.7e7, 5.7e7, 5.7e7
*DAMAGE EVOLUTION, TYPE=ENERGY,
MIXED MODE BEHAVIOR=BK, POWER=2.284
280.0, 280.0, 280.0
*SECTION CONTROLS, NAME=visco,
VISCOSITY=1.e-5
```

Cohesive surfaces

```
*SURFACE INTERACTION, NAME=cohesive
*COHESIVE BEHAVIOR,
ELIGIBILITY=SPECIFIED CONTACTS
5.7e14, 5.7e14, 5.7e14
*DAMAGE INITIATION, CRITERION=QUADS
5.7e7, 5.7e7, 5.7e7
*DAMAGE EVOLUTION, TYPE=ENERGY,
MIXED MODE BEHAVIOR=BK, POWER=2.284
280.0, 280.0, 280.0
*DAMAGE STABILIZATION
1.e-5
```

Surface-based Cohesive Behavior

- Results



Element- vs. Surface-based Cohesive Behavior

© DASSAULT SYSTEMES



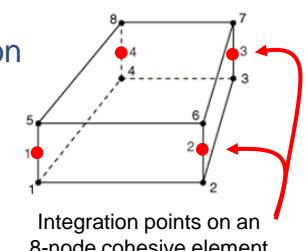
L10.74

Element- vs. Surface-based Cohesive Behavior

Preprocessing

- **Cohesive elements**

- Gives you direct control over the cohesive element mesh density and stiffness properties.
- Constraints are enforced at the element integration points.
 - Refining the cohesive elements relative to the connected structures will likely lead to improved constraint satisfaction and more accurate results.



- **Cohesive surfaces**

- Are easily defined using contact interactions and cohesive interaction properties.
- A pure master-slave in formulation is used.
- Constraints are enforced at the slave nodes.
 - Refining the slave surface relative to the master surface will likely lead to improved constraint satisfaction and more accurate results.

© DASSAULT SYSTEMES

Element- vs. Surface-based Cohesive Behavior

Initial configuration:

- **Cohesive elements**

- Must be bonded at the start of the analysis.
- Once the interface has failed, the surfaces do not re-bond.

- **Cohesive surfaces**

- Can bond anytime contact is established (i.e., “sticky” contact behavior).
- Cohesive interface need not be bonded at the start of the analysis.
- You can control whether debonded surfaces will stick or not stick if contact occurs again.
 - By default, they do not stick.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Element- vs. Surface-based Cohesive Behavior

Constitutive behavior:

- **Cohesive elements**

- Allow for several constitutive behavior types:
 - Traction-separation constitutive model
 - Including multiple failure mechanisms
 - Continuum-based constitutive model
 - For adhesive layers with finite thickness
 - Uses conventional material models
 - Uniaxial stress-based constitutive model
 - Useful in modeling gaskets and/or single adhesive patches

- **Cohesive surfaces**

- Must use the traction-separation interface behavior.
 - Intended for bonded interfaces where the interface thickness is negligibly small.
 - Only one failure mechanism is allowed.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Element- vs. Surface-based Cohesive Behavior

Influence on stable time increment (Abaqus/Explicit only): $\Delta t = \left(\frac{L^e}{c_d} \right)$

- **Cohesive elements**

- Often require a small stable time increment.
- Cohesive elements are generally thin and sometimes quite stiff.
- Consequently, they often have a stable time increment that is significantly less than that of the other elements in the model.

- **Cohesive surfaces**

- Cohesive surface behavior with the default cohesive stiffness properties is formulated to minimally affect the stable time increment.
- Abaqus uses default contact penalties to model the cohesive stiffness behavior in this case.
- You can specify a non-default cohesive stiffness values.
 - However, high stiffnesses may reduce the stable time increment.

Element- vs. Surface-based Cohesive Behavior

Mass:

- **Cohesive elements**

- The element material definitions include mass.

- **Cohesive surfaces**

- Do not add mass to the model.
- Indented for thin adhesive interfaces; thus, neglecting adhesive mass is appropriate for most applications.
 - However, nonstructural mass can be added to the contacting elements if necessary.

Element- vs. Surface-based Cohesive Behavior

Summary:

- **Cohesive elements**
 - Are recommended for more detailed adhesive connection modeling.
 - Additional preprocessing effort (and often increased computational cost) is compensated for by gaining:
 - Direct control over the connection mesh
 - Additional constitutive response options
 - E.g., model adhesives of finite thickness
- **Cohesive surfaces**
 - Provides a quick and easy way to model adhesive connections.
 - Negligible interface thicknesses only
 - Surfaces can bond anytime contact is established (“sticky” contact)
 - Model contact adhesives, Velcro, tape, and other bonding agents that can stick after separation.

Notes

Notes

Virtual Crack Closure Technique (VCCT)

Lecture 11

© DASSAULT SYSTEMES



L11.2

Overview

- Introduction
- VCCT Criterion
- Output
- VCCT Plug-in
- Comparison with Cohesive Behavior
- Examples

© DASSAULT SYSTEMES



Introduction

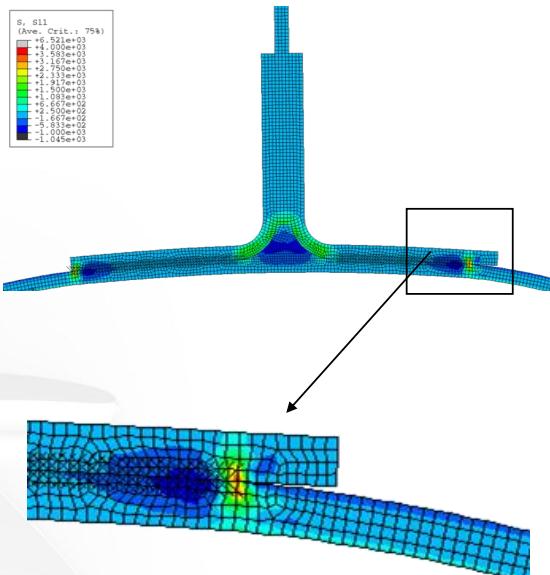
© DASSAULT SYSTEMES



L11.4

Introduction

- Motivation is aircraft composite structural analysis
 - To reduce the cost of laminated composite structures, large integrated bonded structures are being considered.
 - In primary structures, bondlines and interfaces between plies are required to carry interlaminar loads.
 - Damage tolerance requirements dictate that bondlines and interfaces carry required loads with damage.



Modeling debonding along skin-stringer interface



Introduction

- Analysis requirements for composite damage
 - Apply Linear Elastic Fracture Mechanics (LEFM) to bondlines and interfaces
 - 2D and 3D delaminations
 - Propagation
 - Mode separation
 - Multiple cracks
 - Non-linear behavior (e.g., postbuckling)
 - Composite structure
 - Practical (CPU time, minimum set of models)

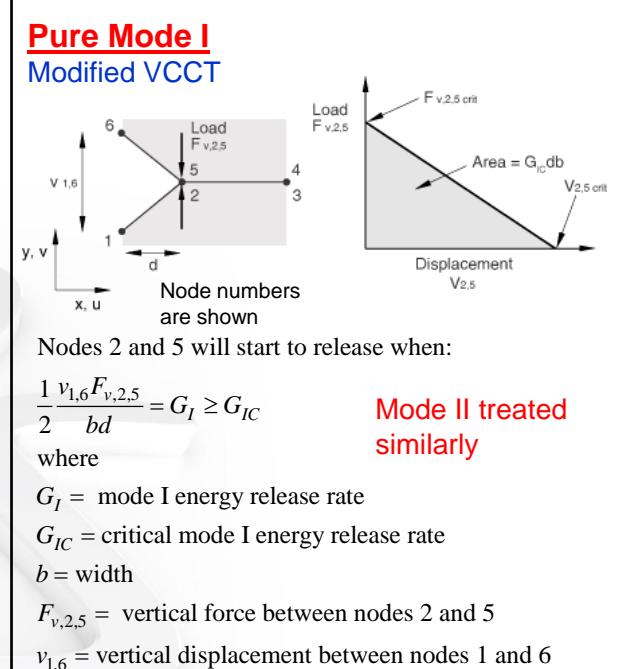
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Introduction

- VCCT uses LEFM concepts
 - Based on computing the energy release rates for normal and shear crack-tip deformation modes.
 - Compare energy release rates to interlaminar fracture toughness.
- See Rybicki, E. F., and Kanninen, M. F., "A Finite Element Calculation of Stress Intensity Factors by a Modified Crack Closure Integral," *Engineering Fracture Mechanics*, Vol. 9, pp. 931-938, 1977.



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

VCCT Criterion

© DASSAULT SYSTEMES



L11.8

VCCT Criterion

- The debond capability is used to perform the crack propagation analysis for initially bonded crack surfaces.
- The crack propagation analysis allows for five types of fracture criteria:
 - 1 Critical stress criterion
 - 2 Crack opening displacement criterion
 - 3 Crack length vs. time criterion
 - 4 VCCT criterion
 - 5 Low-cycle fatigue criterion
- Defining case 4, “VCCT criterion,” is the subject of this lecture.
 - The details of cases 1, 2, and 3 are discussed in Appendix 1 “Additional Crack Propagation Analysis using the Debond Capability.”
 - The details of case 5 will be discussed later in Lecture 12 “Low-cycle Fatigue.”

© DASSAULT SYSTEMES

VCCT Criterion

- When using VCCT to model crack propagation,
 - you must:
 - 1 define contact pairs for potential crack surfaces;
 - 2 define initially bonded crack surfaces;
 - 3 activate the crack propagation capability; and
 - 4 specify the VCCT criterion.
 - you also may:
 - define spatially varying critical energy release rates;
 - use viscous regularization, contact stabilization, and/or automatic stabilization to overcome convergence difficulties for unstable propagating cracks;
 - use a linear scaling technique to accelerate convergence for VCCT.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



VCCT Criterion

- Defining the VCCT criterion is not currently supported in Abaqus/CAE.
 - However, the VCCT plug-in is available and allows you to interactively define the debond interface(s).
 - The details of the VCCT plug-in will be discussed later in this lecture.
 - Downloaded from “VCCT plug-in utility,” SIMULIA Answer 3235.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



VCCT Criterion

- **Example: Double cantilever beam (DCB)**
 - Analyze debonding of a DCB model using the VCCT criterion.
 - Steps required for setting up the model include:
 - Define slave (**TopSurf**) and master (**BotSurf**) surfaces along the debond interface.
 - Define a set (**bond**) containing the initially bonded region (part of **TopSurf** in this example).
 - The Keywords interface is illustrated in this example.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

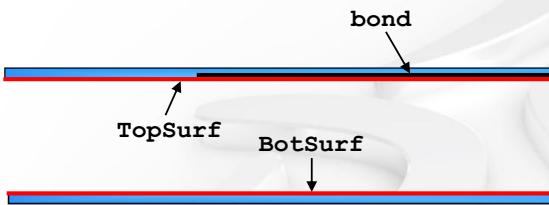
**bond****TopSurf****BotSurf**

VCCT Criterion

1 Define contact pairs for potential crack surfaces

- Potential crack surfaces are modeled as slave and master contact surfaces.
 - Any contact formulation except the finite-sliding, surface-to-surface formulation can be used.
 - Cannot be used with self-contact.

© DASSAULT SYSTEMES



Note: The frictionless interaction property is assumed.

```
*NSET, NSET=bond, GENERATE
1, 121, 1
*SURFACE, NAME=TopSurf
_TopBeam_S1, S1
*SURFACE, NAME=BotSurf
_BotBeam_S1, S1
*CONTACT PAIR, INTER=...
TopSurf, BotSurf
slave surface    master surface
```

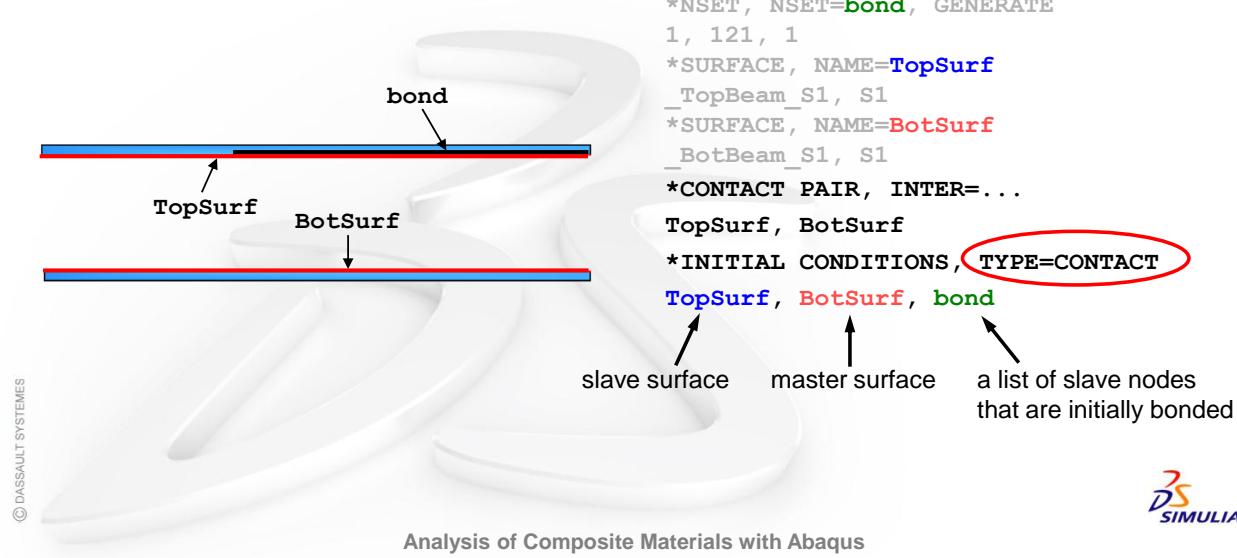


Analysis of Composite Materials with Abaqus

VCCT Criterion

2 Define initially bonded crack surfaces

- The initially bonded contact pair is identified with the *INITIAL CONDITIONS, TYPE=CONTACT option.



VCCT Criterion

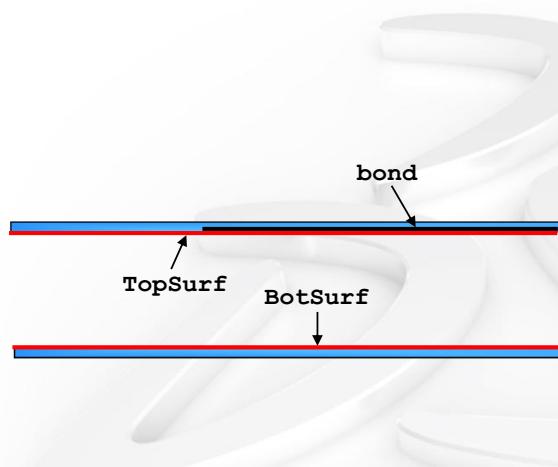
- The unbonded portion of the slave surface will behave as a regular contact surface.
- If the node set that includes the initially bonded slave nodes is not specified, the initial contact condition will apply to the entire contact pair.
 - In this case, no crack tips can be identified, and the bonded surfaces cannot separate.
- For the VCCT criterion, the initially bonded nodes are bonded in all directions.

VCCT Criterion

3 Activate the crack propagation capability

- The *DEBOND option is used to activate crack propagation in a given step.
- The SLAVE and MASTER parameters identify the surfaces to be debonded.

© DASSAULT SYSTEMES



```

*NSET, NSET=bond, GENERATE
1, 121, 1
*SURFACE, NAME=TopSurf
_TopBeam_S1, S1
*SURFACE, NAME=BotSurf
_BotBeam_S1, S1
*CONTACT PAIR, INTER=...
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf

```



Analysis of Composite Materials with Abaqus

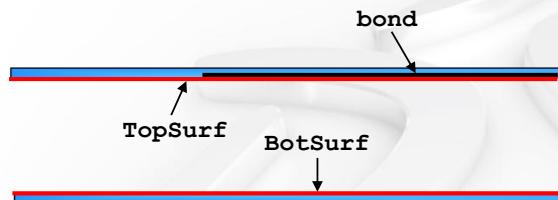
VCCT Criterion

4 Specify the VCCT criterion

- The **BK law** model is used in this example.

BK law:

$$G_{equivC} = G_{IC} + G_{IIC} - G_{IC} \left(\frac{G_H + G_{III}}{G_I + G_H + G_{III}} \right)^\eta$$



```

*NSET, NSET=bond, GENERATE
1, 121, 1
*SURFACE, NAME=TopSurf
_TopBeam_S1, S1
*SURFACE, NAME=BotSurf
_BotBeam_S1, S1
*CONTACT PAIR, INTER=...
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf
*FRACTURE CRITERION, TYPE=VCCT,
MIXED MODE BEHAVIOR=BK
280.0, 280.0, 0.0, 2.284

```



Analysis of Composite Materials with Abaqus

VCCT Criterion

- The crack-tip node debonds when the fracture criterion, f ,

$$f = \frac{G_{equiv}}{G_{equivC}},$$

reaches the value 1.0 within a given tolerance, f_{tol} :

$$1 \leq f \leq 1 + f_{tol}.$$

where

G_{equiv} is the equivalent strain energy release rate, and

G_{equivC} is the critical equivalent strain energy release rate calculated based on the user-specified mode-mix criterion and the bond strength of the interface.

- For the VCCT criterion, the default value of f_{tol} is 0.2.
 - Use following option to control f_{tol} :

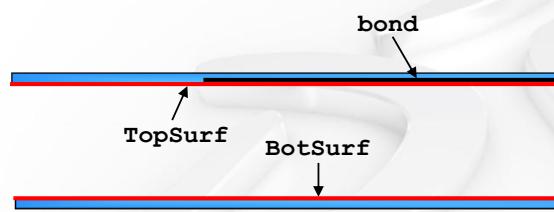
*FRACTURE CRITERION, TYPE=VCCT, TOLERANCE= f_{tol}



Analysis of Composite Materials with Abaqus

VCCT Criterion

- In the DCB model, the tolerance is set to 0.1.



```

*NSET, NSET=bond, GENERATE
1, 121, 1
*SURFACE, NAME=TopSurf
_TopBeam_S1, S1
*SURFACE, NAME=BotSurf
_BotBeam_S1, S1
*CONTACT PAIR, INTER=...
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf
*FRACTURE CRITERION, TYPE=VCCT,
MIXED MODE BEHAVIOR=BK, TOLERANCE=0.1
280.0, 280.0, 0.0, 2.284

```



Analysis of Composite Materials with Abaqus

VCCT Criterion

- In addition to the BK law model, Abaqus/Standard also provides two other commonly used mode-mix criteria for computing G_{equivC} : the Power law and the Reeder law models.
 - An appropriate model is best selected empirically.
- Power law**

$$\frac{G_{equiv}}{G_{equivC}} = \left(\frac{G_I}{G_{IC}} \right)^{am} + \left(\frac{G_{II}}{G_{IIC}} \right)^{an} + \left(\frac{G_{III}}{G_{IIIC}} \right)^{ao}$$

*FRACTURE CRITERION, TYPE=VCCT, MIXED MODE BEHAVIOR=POWER
 $G_{IC}, G_{IIC}, G_{IIIC}, am, an, ao$

- Reeder law**
 - Applies only to three-dimensional problems

$$G_{equivC} = G_{IC} + \left(G_{IIC} - G_{IC} + G_{IIIC} - G_{IIC} \left(\frac{G_{III}}{G_{II} + G_{III}} \right) \right) \left(\frac{G_{II} + G_{III}}{\sum G_i} \right)^\eta$$

*FRACTURE CRITERION, TYPE=VCCT, MIXED MODE BEHAVIOR=REEDER
 $G_{IC}, G_{IIC}, G_{IIIC}, \eta$

VCCT Criterion

- Spatially varying critical energy release rates**
 - The VCCT criterion can be defined with varying energy release rates by specifying the critical energy release rates at all nodes on the slave surface.
 - In this case, the critical energy release rates should be interpolated from the critical energy release rates specified at the nodes with the *NODAL ENERGY RATE option.
 - However, the exponents (e.g., η) are still read from the data lines under the *FRACTURE CRITERION option.

```
*NODAL ENERGY RATE
node ID1, GIC, GIIC, GIIIC
node ID2, GIC, GIIC, GIIIC
...

```

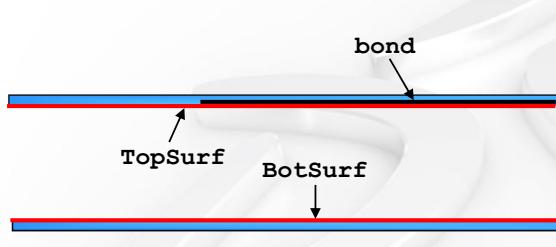
model data

```
*STEP
*STATIC
...
```

```
*FRACTURE CRITERION, TYPE=VCCT, MIXED MODE BEHAVIOR=BK, NODAL ENERGY RATE
GIC, GIIC, GIIIC, η
```

VCCT Criterion

- Viscous regularization for VCCT
 - Can be used to overcome some convergence difficulties for unstable propagating cracks.
 - Example: DCB
 - Set the value of the viscosity coefficient to 0.1.



© DASSAULT SYSTEMES

```

*NSET, NSET=bond, GENERATE
1, 121, 1
*SURFACE, NAME=TopSurf
_TopBeam_S1, S1
*SURFACE, NAME=BotSurf
_BotBeam_S1, S1
*CONTACT PAIR, INTER=...
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf,
MASTER=BotSurf, VISCOSITY=0.1
*FRACTURE CRITERION, TYPE=VCCT, MIXED
MODE BEHAVIOR=BK, TOLERANCE=0.1
280.0, 280.0, 0.0, 2.284

```



Analysis of Composite Materials with Abaqus

VCCT Criterion

- In addition, contact and automatic stabilization that are not specific to VCCT can be also used to aid convergence.
 - They are built into Abaqus/Standard and are compatible with VCCT.
- Note that the crack propagation behavior may be modified by the damping forces.
 - Therefore, monitor the damping energy (ALLVD or ALLSD) and compare it with the total strain energy in the model (ALLSE) to ensure that the results are reasonable in the presence of damping.
 - ALLVD stores the damping energy generated from viscous regularization.
 - ALLSD stores the damping energy generated from contact stabilization and automatic stabilization.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



VCCT Criterion

- **Linear scaling to accelerate convergence for VCCT**
 - Abaqus provides a linear scaling technique to quickly converge to the critical load state. This reduces the solution time required to reach the onset of crack growth.
 - This technique works best for models in which the deformation is nearly linear before the onset of crack growth.
 - Once the first crack-tip node releases, the linear scaling calculations will no longer be valid and the time increment will be set to the default value.
 - Usage:
 $\star\text{CONTROLS, LINEAR SCALING}$
 β
 where β is the coefficient of linear scaling.
- For details of linear scaling to accelerate convergence for VCCT, see “Crack propagation analysis,” Section 11.4.3 of the Abaqus Analysis User’s Manual.

VCCT Criterion

- **Tips for using the VCCT criterion**
 - Crack propagation problems using the VCCT criterion are numerically challenging.
 - To help you create a successful model, several tips for using the VCCT criterion are provided:
 - The master debonding surfaces must be continuous.
 - The tie MPCs should NOT be used for the slave debonding surface to avoid overconstraints.
 - A small clearance between the debonding surfaces can be specified to eliminate unnecessary severe discontinuity iterations during incrementation as the crack begins to progress.
 - Note: More tips are provided in “Crack propagation analysis,” Section 11.4.3 of the Abaqus Analysis User’s Manual.

Output



L11.26

Output

- The following output options are provided to support the VCCT criterion:
 - Abaqus/CAE supports the surface output requests for VCCT.

OUTPUT, FIELD, FREQUENCY=*freq
CONTACT OUTPUT, MASTER=*master*, SLAVE=*slave

OUTPUT, HISTORY, FREQUENCY=*freq
***CONTACT OUTPUT, [(MASTER=*master*, SLAVE=*slave*) | (NSET=*nset*)]**

Edit Field Output Request

Name: F-Output-1
Step: Step-1
Procedure: Static, General

Domain: Interaction : Int-1

Frequency: Every n increments n: 1

Timing: Output at exact times

Output Variables

Select from list below Predefined variables

DBS,DBT,DBSF,OPENBC,CRSTS

Contact

Failure/Fracture

DBS, Remaining stress

DBT, Time at bond failure

DBSF, Fraction of remaining failed bond stress

OPENBC, Opening behind crack tip at bond failure

CRSTS, Critical stress at bond failure

ENRRT, Strain energy release rates

EFENRRTR, Effective energy release rate ratio

BDSTAT, Bond state

Edit History Output Request

Name: H-Output-1
Step: Step-1
Procedure: Static, General

Domain: Interaction : Int-1

Frequency: Every n increments n: 1

Timing: Output at exact times

Output Variables

Select from list below Preselected defaults All Enabled

DBS,DBT,DBSF,OPENBC,CRSTS,ENRRT,EFENRRTR,BDSTAT,

Contact

Failure/Fracture

DBS, Remaining stress in failed bond

DBT, Time at bond failure

DBSF, Fraction of remaining failed bond stress

OPENBC, Opening behind crack tip at bond failure

CRSTS, Critical stress at bond failure

ENRRT, Strain energy release rates

EFENRRTR, Effective energy release rate ratio

BDSTAT, Bond state

Output

- The following bond failure quantities can be requested as surface output:

DBT	The time when bond failure occurred
DBSF	Fraction of stress at bond failure that still remains
DBS	Stress in the failed bond that remains
OPENBC	Relative displacement behind crack.
CRSTS	Critical stress at failure.
ENRRT	Strain energy release rate.
EFENRRTR	Effective energy release rate ratio.
BDSTAT	Bond state (=1.0 if bonded, 0.0 if unbonded)
- All of the above variables can be visualized in Abaqus/Viewer.
- The initial contact status of all of the slave nodes is printed in the data (.dat) file.

Output

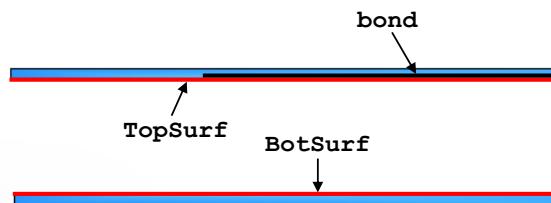
- Example: DCB**

- Request surface output:

```

...
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf, VISCOSITY=0.1
*FRACTURE CRITERION, TYPE=VCCT, MIXED MODE BEHAVIOR=BK, TOLERANCE=0.1
280, 280, 280, 2.284
...

```



```

*OUTPUT, FIELD, VAR=PRESELECT
*CONTACT OUTPUT, SLAVE=TopSurf, MASTER=BotSurf
DBT, DBS, OPENBC, CRSTS, ENRRT, BDSTAT
*OUTPUT, HISTORY
*CONTACT OUTPUT, SLAVE=TopSurf, MASTER=BotSurf, NSET=bond
DBT, DBS, OPENBC, CRSTS, ENRRT, BDSTAT
*NODE OUTPUT, NSET=tip
U2, RF2
*END STEP

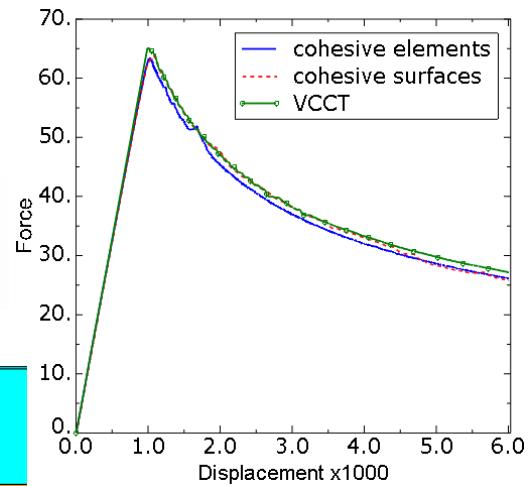
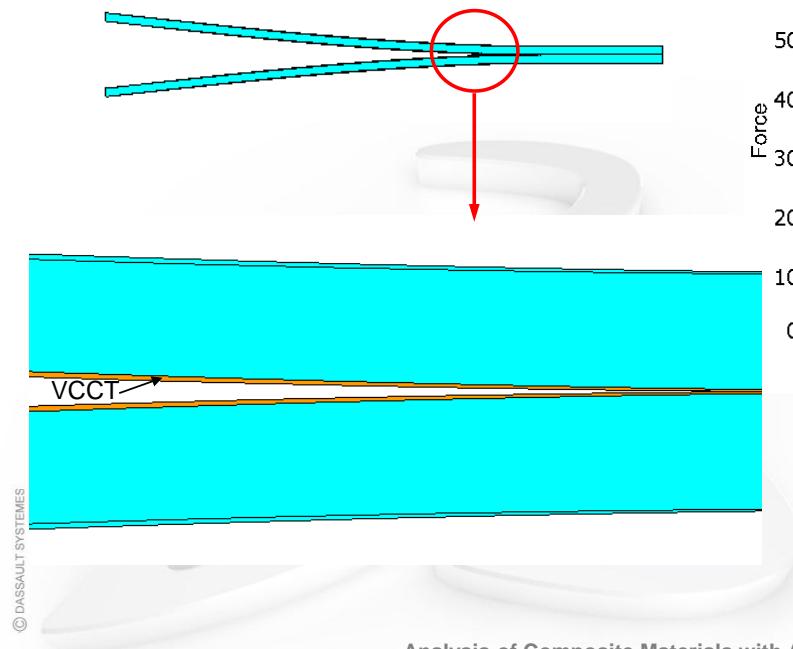
```

field output

history output

Output

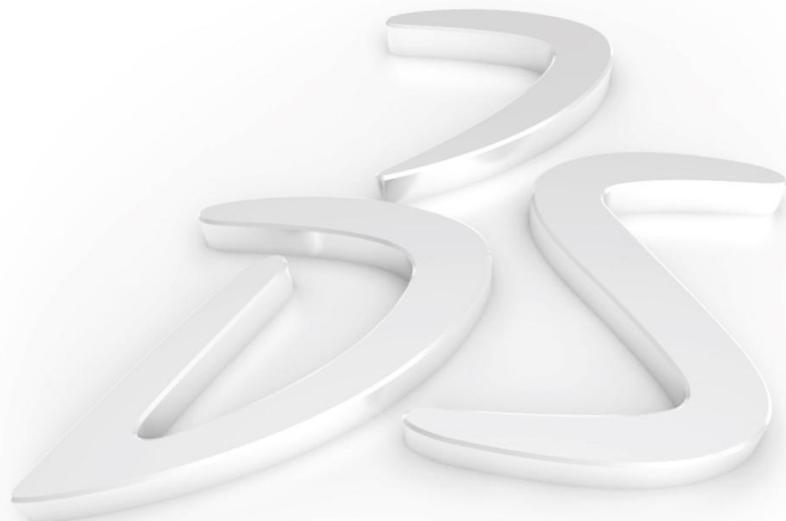
- Results



Analysis of Composite Materials with Abaqus



VCCT Plug-in



VCCT Plug-in

- **VCCT plug-in**

- provides an interactive interface to define the debond interface(s).
- supports the following keyword options required for VCCT analysis:

```
*INITIAL CONDITIONS, TYPE=CONTACT

*DEBOND, SLAVE=slave, MASTER=master, OUTPUT=[fil|dat|both], VISCOSITY= $\mu$ 

*FRACTURE CRITERION, TYPE=VCCT,
MIXED MODE BEHAVIOR=[BK|POWER|REEDER], TOLERANCE= $f_{tol}$ ,
NODAL ENERGY RATE

*NODAL ENERGY RATE

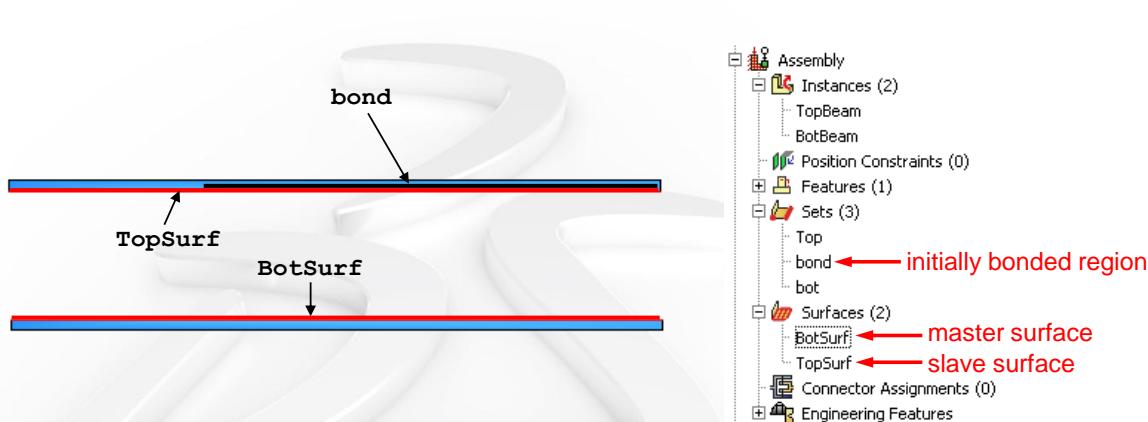
*CONTROLS, LINEAR SCALING
```

- For details please refer to “VCCT plug-in utility,” SIMULIA Answer 3235.

VCCT Plug-in

- **Example: Double Cantilever Beam (DCB)**

- The VCCT plug-in is discussed in the context of the Keywords interface presented earlier.



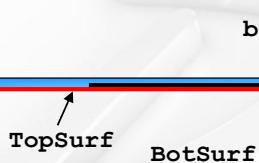
VCCT Plug-in

1 Define contact pairs for potential crack surfaces

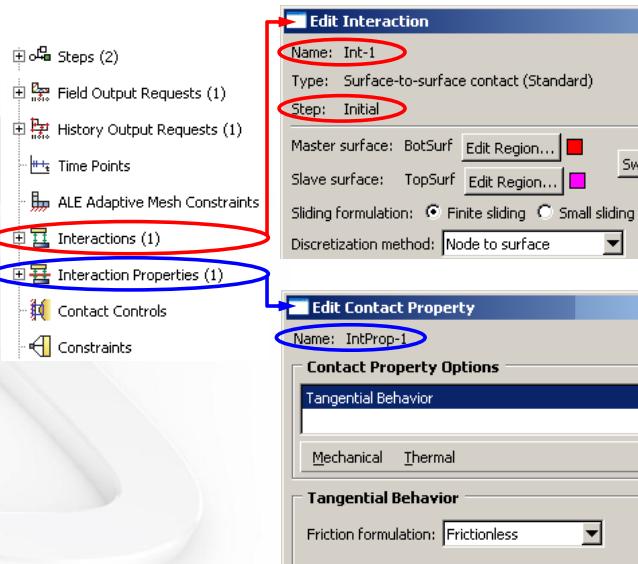
- Frictionless contact is assumed.

```
*SURFACE INTERACTION, NAME=IntProp-1
1.
*FRICTION
0.0
*CONTACT PAIR, INTERACTION=IntProp-1
TopSurf, BotSurf
```

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus



SIMULIA

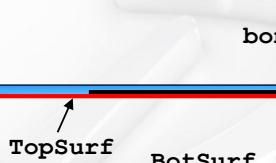
VCCT Plug-in

2 Define the VCCT criterion

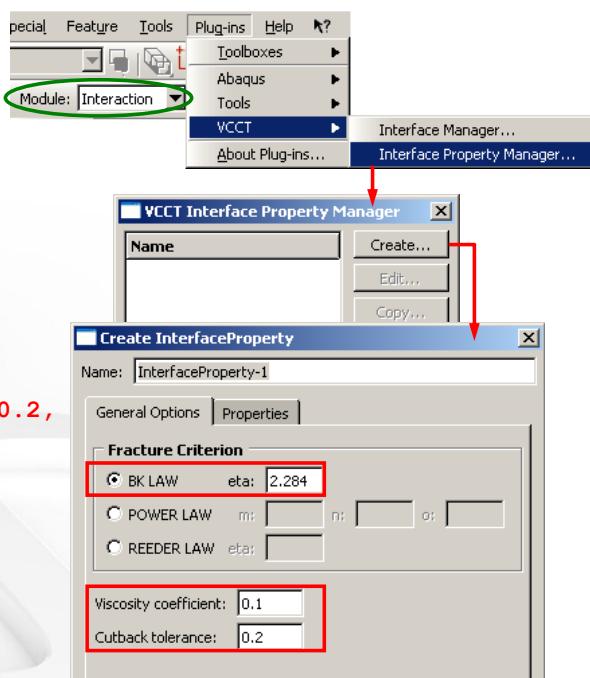
- 2a** Select the fracture criterion, viscosity coefficient, and cutback tolerance.

```
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf,
VICOSITY=0.1
*FRACTURE CRITERION, TYPE=VCCT, TOLERANCE=0.2,
MIXED MODE BEHAVIOR=BK
280, 280, 280, 2.284
```

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

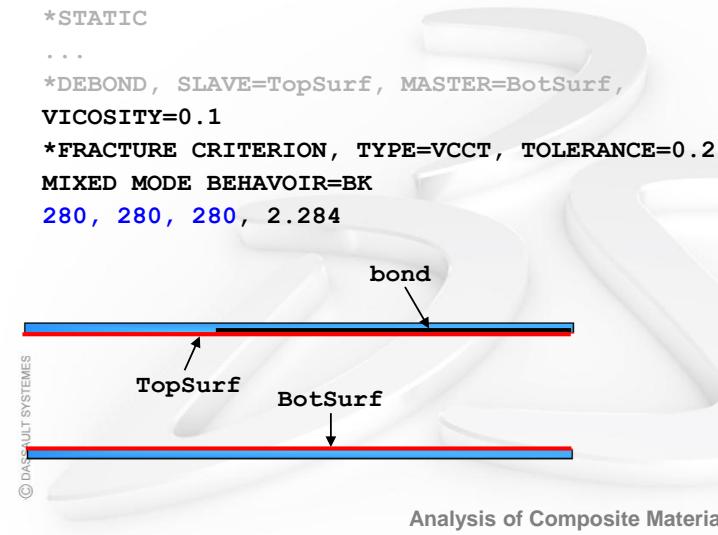


SIMULIA

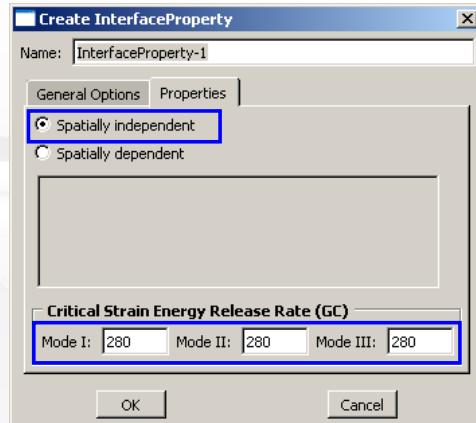
VCCT Plug-in

2b Specify critical strain energy release rates

...
`*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf,
VICOSITY=0.1
*FRACTURE CRITERION, TYPE=VCCT, TOLERANCE=0.2,
MIXED MODE BEHAVIOR=BK
280, 280, 280, 2.284`



© DASSAULT SYSTEMES





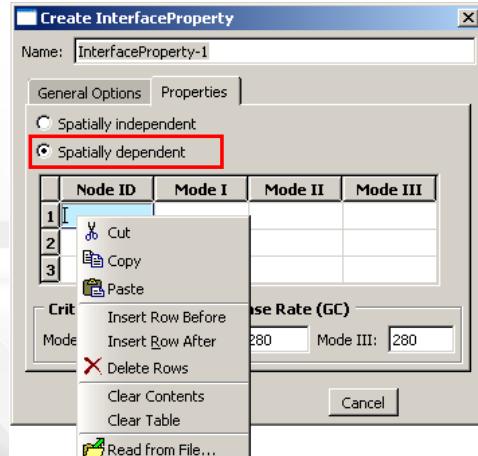
Analysis of Composite Materials with Abaqus

VCCT Plug-in

- The VCCT plug-in also supports defining spatially varying critical energy release rates.
 - Click mouse button 3 to manage the table.

`*NODAL ENERGY RATE
node ID1, GIC, GIIC, GIIIIC
node ID2, GIC, GIIC, GIIIIC
...
*STEP
*STATIC
...
*FRACTURE CRITERION, TYPE=VCCT,
MIXED MODE BEHAVIOR=BK, NODAL ENERGY RATE
GIC, GIIC, GIIIIC, η`

© DASSAULT SYSTEMES





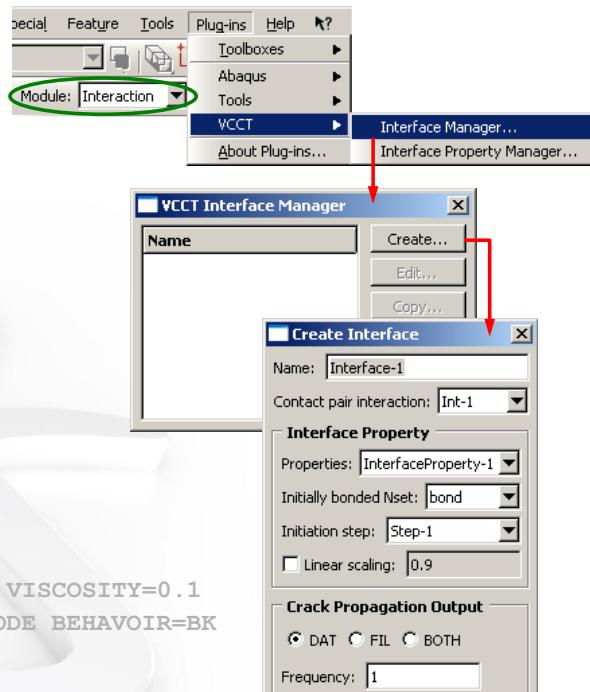
Analysis of Composite Materials with Abaqus

VCCT Plug-in

3 Define the VCCT bonded interface

- Select the initially bonded region, the crack propagation output file and frequency, and the debond initiation step.
- Note: The VCCT plug-in allows specification of linear scaling.

```
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NAME=Step-1
*STATIC, NLGEOM
...
*DEBOND, SLAVE=TopSurf, MASTER=BotSurf, VISCOSITY=0.1
*FRACTURE CRITERION, TYPE=VCCT, MIXED MODE BEHAVIOR=BK
280, 280, 280, 2.284
```



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

VCCT Plug-in

- The relevant keywords will be generated when Abaqus/CAE writes the input file.

surface interaction

initial contact conditions

debond

fracture criterion

field output

history output

```
** Interaction: Int-1
*Contact Pair, interaction=IntProp-1
TopSurf, BotSurf
*Initial conditions, type=contact
TopSurf, BotSurf, bond
**
** -----
**
** STEP: Step-1
**
*Step, name=Step-1, nlgeom=YES
*Static
1., 1., 1e-05, 1.
**
** OUTPUT REQUESTS
**
*Debond, slave=TopSurf, master=BotSurf, FREQUENCY=1, viscosity=0.1
*Fracture Criterion, tolerance=0.2, type=vcct, MIXED MODE BEHAVIOR=BK
228, 228, 228, 2.285
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
  *Element Output, directions=YES
    E, S
*Contact Output
  BDSTAT, CRSTS, DBS, DBSF, DBT, EFENRRTR, ENRRT, OPENBC
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Contact Output, master=BotSurf, slave=TopSurf
  BDSTAT, CRSTS, DBS, DBSF, DBT, EFENRRTR, ENRRT, OPENBC
```

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Comparison with Cohesive Behavior

© DASSAULT SYSTEMES



L11.40

Comparison with Cohesive Behavior

- VCCT and cohesive behavior are very similar in their application and formulation.
 - Both theories
 - are used to model interfacial shearing and delamination crack propagation and failure,
 - use an elastic damage constitutive theory to model the material's response once damage has initiated, and
 - dissipate the same amount of fracture energy between damage initiation and complete failure.

© DASSAULT SYSTEMES



Comparison with Cohesive Behavior

- The fundamental difference between VCCT and cohesive behavior is in the way crack propagation is predicted.
 - In VCCT an existing flaw is assumed.
 - VCCT is appropriate for brittle crack propagation problems.
 - However, cohesive behavior can model damage initiation.
 - Damage initiation in cohesive behavior is based strictly on the predefined ultimate (normal and/or shear) stress/strain limit.
 - Cohesive behavior can be used for both brittle and ductile crack propagation problems.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Comparison with Cohesive Behavior

- VCCT may be viewed as more fundamentally based on fracture mechanics.
 - The damage initiation and damage evolution are both based on fracture energy, whereas cohesive behavior use the fracture energy only during damage evolution.
- Applicability of VCCT is limited to “self-similar” crack propagation analyses.
 - This implies a steady-state running crack.
 - Difficult to reproduce in practice.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Comparison with Cohesive Behavior

- Summary: Complementary techniques for modeling of debonding

VCCT	Cohesive behavior
Use the debond framework (surface based)	Interface elements (element based) or contact (surface based)
Assumes an existing flaw	Can model crack initiation
Brittle fracture using LEFM occurring along a well defined crack front	Ductile fracture occurring over a smeared crack front modeled with spanning cohesive elements or cohesive contact
Requires GI, GII, and GIII	Requires E, σ_{max} , GI, GII, and GIII
Crack propagates when strain energy release rate exceeds fracture toughness	Crack initiates when cohesive traction exceeds critical value and releases critical strain energy when fully open
Crack surfaces are rigidly bonded when uncracked.	Crack surfaces are joined elastically when uncracked.
Available only in Abaqus/Standard	Available in Abaqus/Standard and Abaqus/Explicit

- Both are needed to satisfy general fracture requirements

Examples

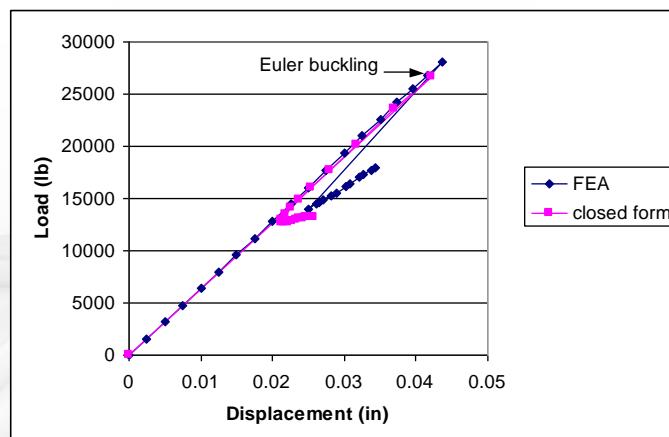
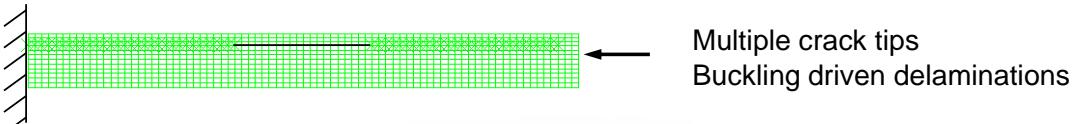
Examples

- Verification problems

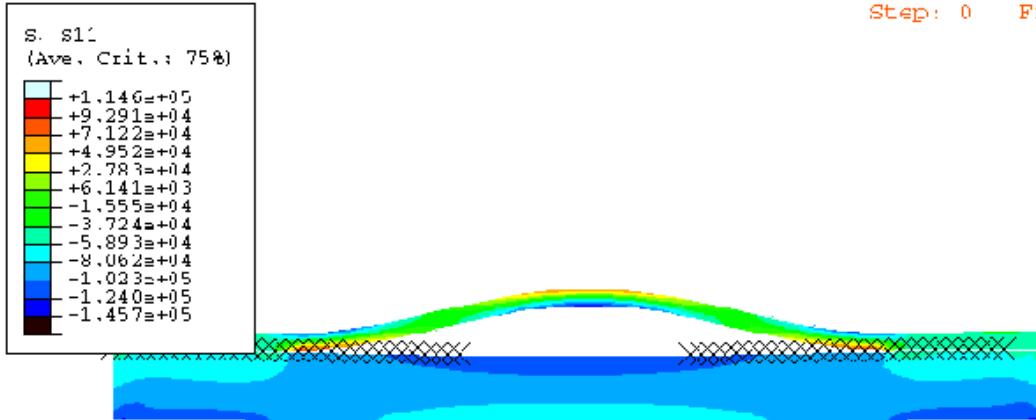
- DCB
- SLB
- ENF
- Alfano-Crisfield
 - Alfano, G., and M. A. Crisfield, "Finite Element Interface Models for the Delamination Analysis of Laminated Composites: Mechanical and Computational Issues," *International Journal for Numerical Methods in Engineering*, vol. 50, pp. 1701–1736, 2001.
 - Also available as Abaqus Benchmark Problem 2.7.1 with cohesive elements
- NASA Panel
 - Reeder, J.R., Song, K., Chunchu, P.B., and Ambur, D.R., "Postbuckling and Growth of Delaminations in Composite Plates Subjected to Axial Compression," AIAA 2002-1746.

Examples

- Compression Buckling/Delamination Single Disbond (Unreinforced)



Examples



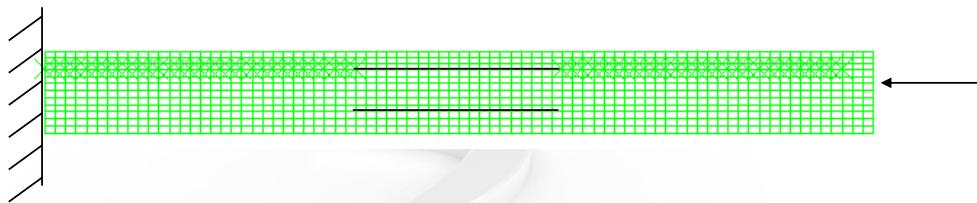
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Examples

- **Compression Buckling/Delamination Multiple Disbonds (Unreinforced)**



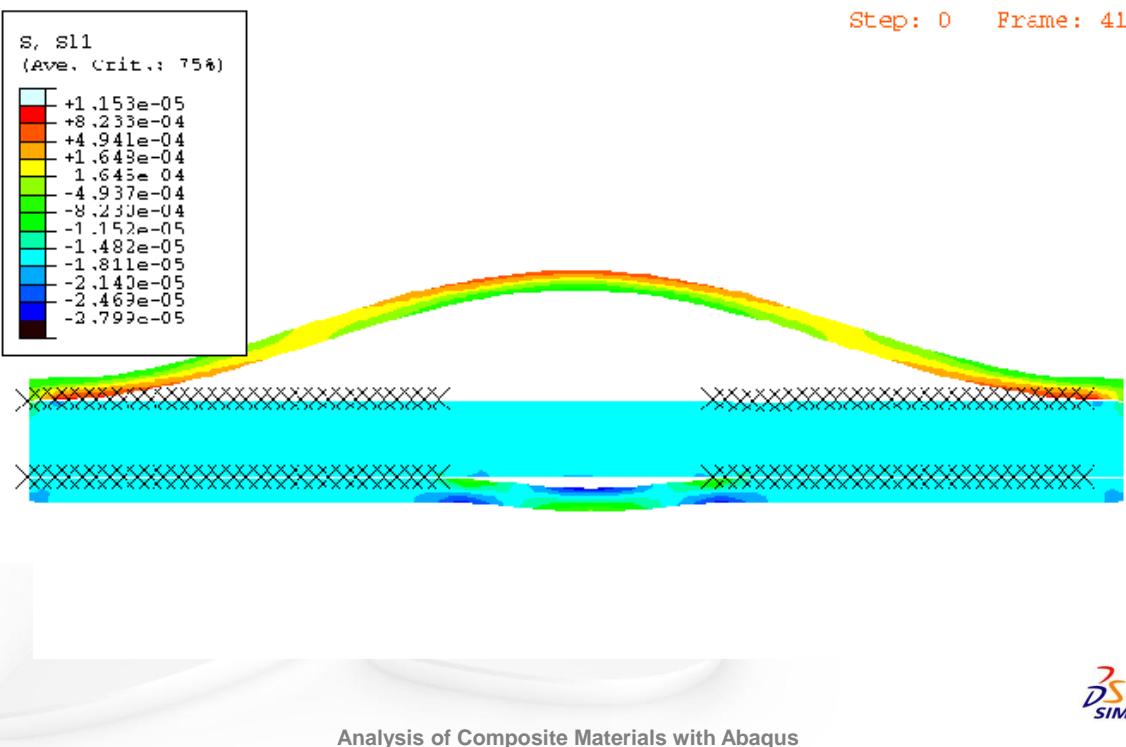
Multiple cracks can also be addressed

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Examples

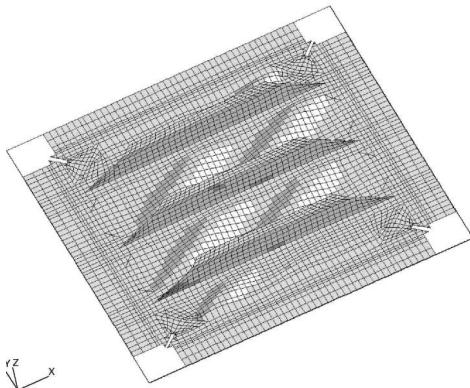


Examples

- T-Joint Pull-off Model

Examples

- Postbuckling Behavior of Skin-Stringer Panels



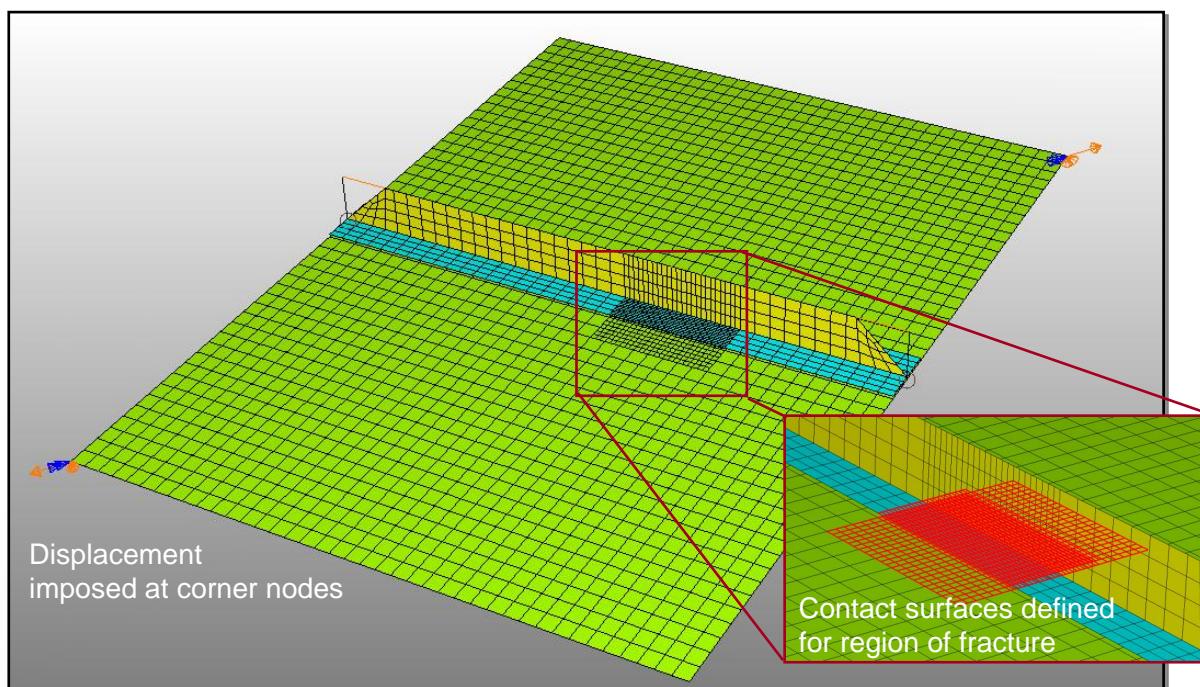
Courtesy Boeing



Analysis of Composite Materials with Abaqus

© DASSAULT SYSTEMES

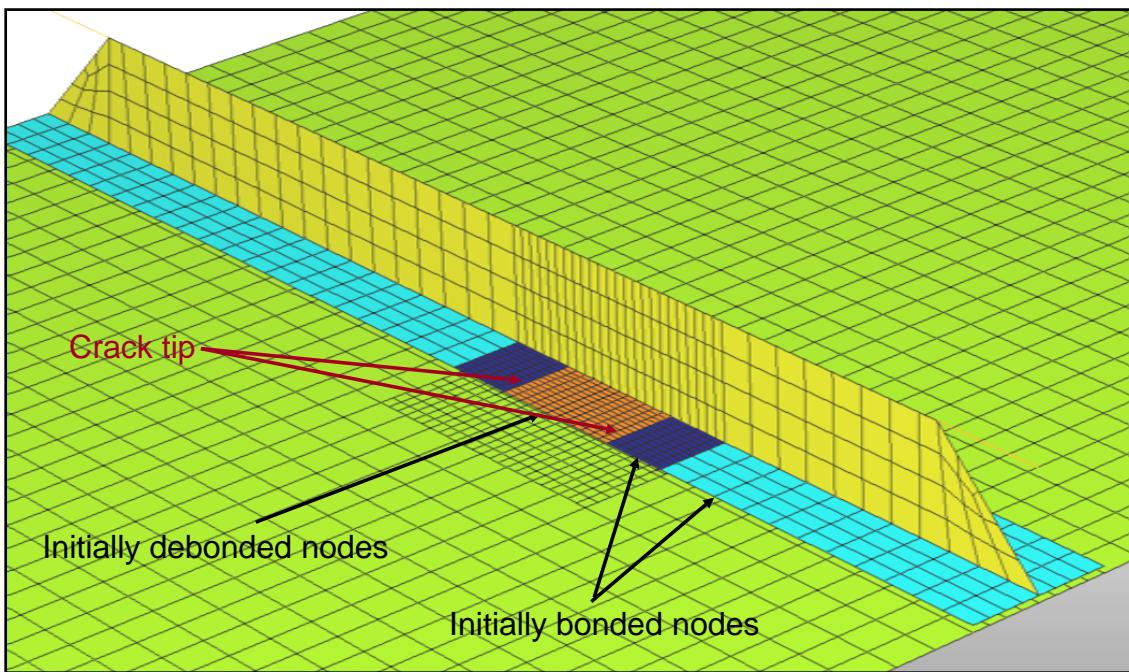
Examples



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Examples

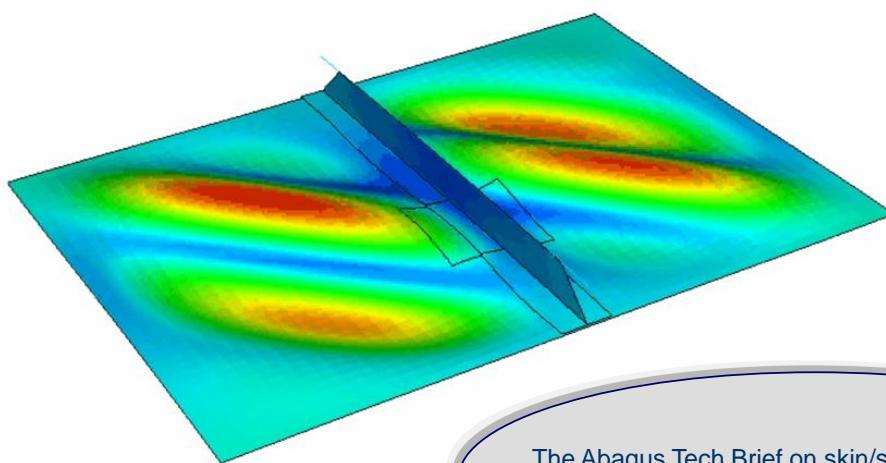


© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Examples



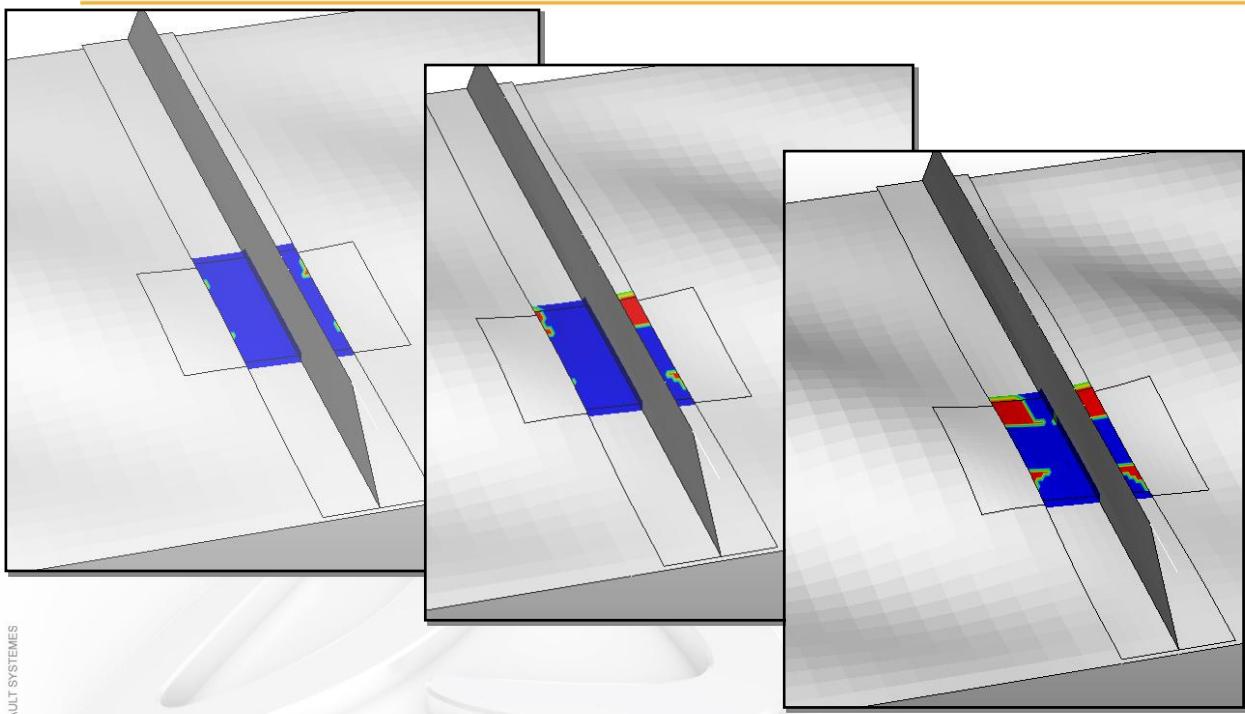
© DASSAULT SYSTEMES

The Abaqus Tech Brief on skin/stringer bonded joint analysis can be downloaded from www.simulia.com



Analysis of Composite Materials with Abaqus

Examples



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Notes

Notes

Low-cycle Fatigue Criterion

Lecture 12

© DASSAULT SYSTEMES



L12.2

Overview

- **Introduction**
- **Direct Cyclic Low-cycle Fatigue Analysis**
- **Low-cycle Fatigue Criterion**

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES



L12.4

Introduction

- Delamination growth in composites due to sub-critical cyclic loadings is a widespread concern for the aerospace industry.
- The low-cycle fatigue criterion available in Abaqus models progressive delamination growth at interfaces in laminated composites subjected to sub-critical cyclic loadings.
 - This criterion can be used only in a low-cycle fatigue analysis using the direct cyclic approach (*DIRECT CYCLIC, FATIGUE).
 - The interface along which the delamination (or crack) propagates must be indicated in the model.
 - The fracture energy release rates at the crack tips in the interface elements are calculated based on the VCCT technique.

© DASSAULT SYSTEMES



Direct Cyclic Low-cycle Fatigue Analysis

© DASSAULT SYSTEMES



L12.6

Direct Cyclic Low-cycle Fatigue Analysis

• Overview

- Low-cycle fatigue analysis is a quasi-static analysis on a structure subjected to sub-critical cyclic loading.
- It models progressive delamination growth at the interface in laminated composites and progressive damage and failure in bulk materials.
 - The onset and growth of delamination are characterized by the Paris Law.
 - The details will be discussed later in section “Low-cycle Fatigue Criterion.”
 - Progressive damage and failure in bulk materials will not be covered in this lecture.
 - It can be associated with thermal as well as mechanical loading.

© DASSAULT SYSTEMES

Direct Cyclic Low-cycle Fatigue Analysis

- Low-cycle fatigue analysis uses the direct cyclic procedure to directly obtain the stabilized cyclic response of the structure.
 - The direct cyclic procedure combines a Fourier series approximation with time integration of the nonlinear material behavior to obtain the stabilized cyclic solution iteratively using a modified Newton method.
 - You can control the number of Fourier terms, the number of iterations, and the incrementation during the cyclic time period to improve the accuracy.
- Within each loading cycle, it assumes geometrically linear behavior and fixed contact conditions.
- For more details, please see “Low-cycle fatigue analysis using the direct cyclic approach,” Section 6.2.7 of the Abaqus Analysis User’s Manual.

Direct Cyclic Low-cycle Fatigue Analysis

- Defining low-cycle fatigue analysis

***DIRECT CYCLIC, FATIGUE, [CETOL=tolerance, DELTMX=Δθ_{max}]**

$\Delta t_0, T, \Delta t_{\min}, \Delta t_{\max}, n_0, n_{\max}, \Delta n, i_{\max}$

$\Delta N_{\min}, \Delta N_{\max}, N, \Delta D_{tol}$

where Δt_0 : initial time increment

T : time of a single loading cycle

Δt_{\min} : minimum time increment allowed

Δt_{\max} : maximum time increment allowed

n_0 : initial number of terms in the Fourier series

n_{\max} : maximum number of terms in the Fourier series

Δn : increment in number of terms in the Fourier series

i_{\max} : maximum number of iterations allowed in a step → controls the iteration

N : total number of cycles allowed in a step

} controls the incrementation

} controls the Fourier series representations

Note: $\Delta N_{\min}, \Delta N_{\max}$, and ΔD_{tol} control damage extrapolation in the bulk material and will not be covered in this lecture.

Direct Cyclic Low-cycle Fatigue Analysis

- **Limitations**

- Contact conditions cannot change during a given cycle when direct cyclic analysis is used iteratively to obtain a stabilized solution.
- Geometric nonlinearity can be included only in any general step prior to a direct cyclic step; however, only small displacements and strains will be considered during the cyclic step.

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Low-cycle Fatigue Criterion

© DASSAULT SYSTEMES



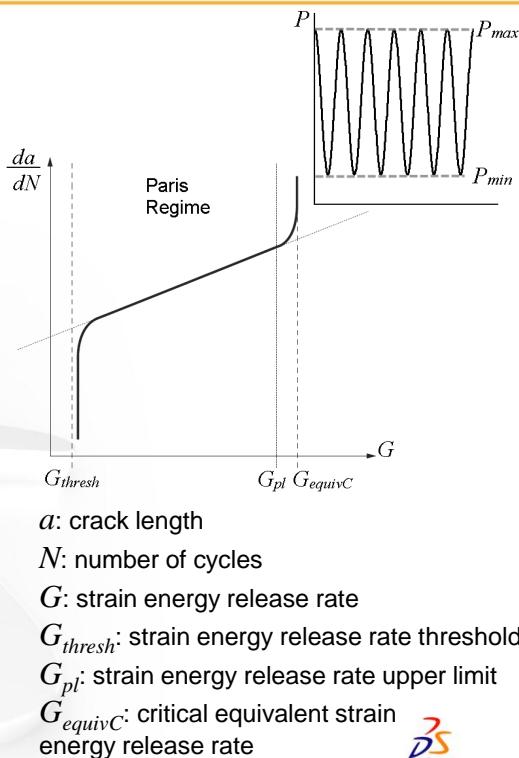
Low-cycle Fatigue Criterion

- The onset and fatigue delamination growth at the interfaces are characterized by using the Paris Law, which relates crack growth rates da/dN to the relative fracture energy release rate ΔG ,

$$\Delta G = G_{max} - G_{min}$$

where G_{max} and G_{min} correspond to the strain energy release rates when the structure is loaded up to P_{max} and P_{min} , respectively.

- The Paris regime is bounded by G_{thresh} and G_{pl} .
 - Below G_{thresh} , there is no fatigue crack initiation or growth.
 - Above G_{pl} , the fatigue crack will grow at an accelerated rate.



Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

- G_{equivC} is calculated based on the user-specified mode-mix criterion and the bond strength of the interface.

- This was discussed in Lecture 11 “VCCT.”

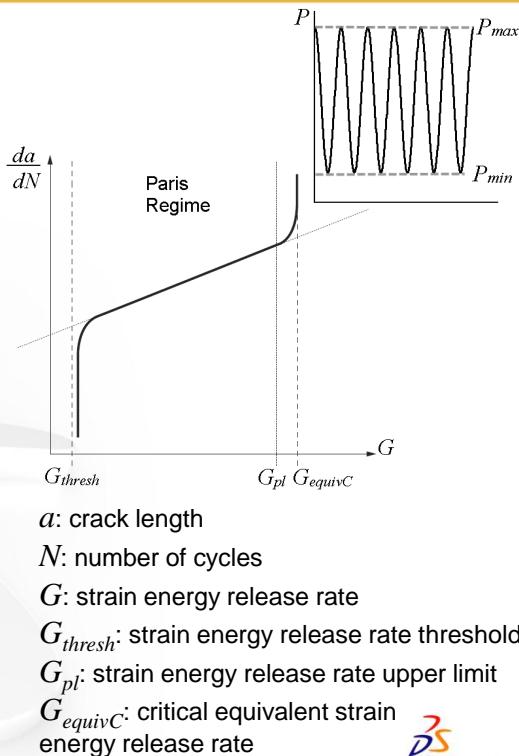
- Onset of fatigue delamination

- The fatigue crack growth initiation criterion is defined as:

$$f = \frac{N}{c_1 \Delta G^{c_2}} \geq 1.0,$$

where c_1 and c_2 are material constants.

- The interface elements at the crack tips will not be released unless the above equation is satisfied and $G_{max} > G_{thresh}$.



Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

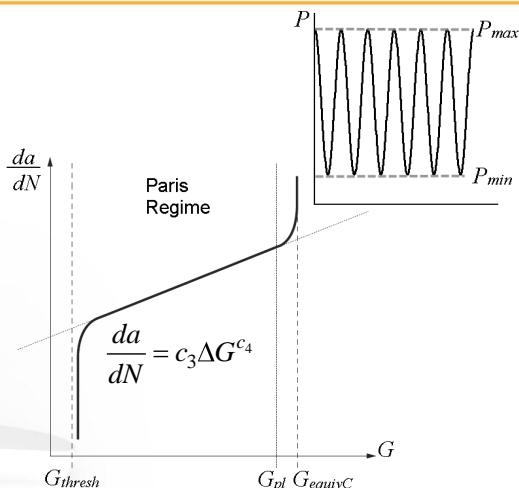
- Fatigue delamination growth**

- Once the delamination growth criterion is satisfied at the interface, the crack growth rate da/dN can be calculated based on ΔG .

- da/dN is given by the Paris Law if $G_{thresh} < G_{max} < G_{pl}$

$$\frac{da}{dN} = c_3 \Delta G^{c_4}$$

where c_3 and c_4 are material constants.



a : crack length

N : number of cycles

G : strain energy release rate

G_{thresh} : strain energy release rate threshold

G_{pl} : strain energy release rate upper limit

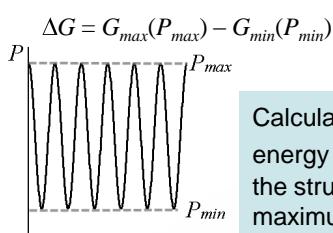
G_{equivC} : critical equivalent strain energy release rate



Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

- Fatigue crack growth governed by the Paris Law**



If $G_{thresh} < G_{max} < G_{pl}$

1

Calculate the relative fracture energy release rate, ΔG , when the structure is loaded between its maximum and minimum values.

a : crack length

N : number of cycles

ΔN : incremental number of cycles

c_1, c_2, c_3, c_4 : material constants

2

Crack initiation: $N_o = c_1 \Delta G^{c_2}$

Crack evolution: $\frac{da}{dN} = c_3 \Delta G^{c_4}$

$$a_{N+\Delta N} = a_N + \Delta N c_3 \Delta G^{c_4}$$

$N + \Delta N$



Release the most critical element



If $N + \Delta N > N_o$

3

Damage extrapolation: Calculate the incremental number of cycles, ΔN , for each crack tip and find minimum cycles to fail, ΔN_{min}

- Repeat the above process until the maximum number of cycles is reached or until the ultimate load carrying capability is reached.

Analysis of Composite Materials with Abaqus



Low-cycle Fatigue Criterion

- The syntax used to define the low-cycle fatigue criterion and the corresponding output requests is similar to those used for the VCCT criterion except the following:
 - For the low-cycle fatigue criterion, set TYPE=FATIGUE on the *FRACTURE CRITERION option:

***FRACTURE CRITERION, TYPE=FATIGUE, MIXED MODE BEHAVIOR=[BK|REEDER]**

$c_1, c_2, c_3, c_4, G_{thresh}/G_{equivC}, G_{pl}/G_{equivC}, G_{IC}, G_{IIC}$
 $G_{IIIC}, \eta, \theta, f_v$

***FRACTURE CRITERION, TYPE=FATIGUE, MIXED MODE BEHAVIOR=POWER**

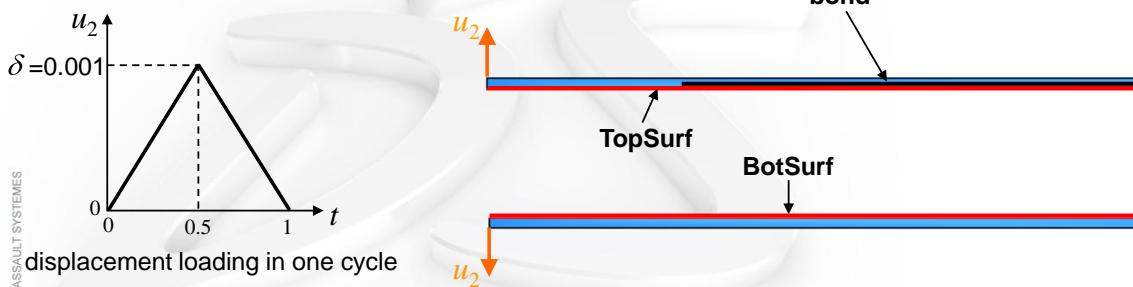
$c_1, c_2, c_3, c_4, G_{thresh}/G_{equivC}, G_{pl}/G_{equivC}, G_{IC}, G_{IIC}$
 $G_{IIIC}, a_m, a_n, a_o, \theta, f_v$

- By default, $G_{thresh}/G_{equivC} = 0.01$ and $G_{pl}/G_{equivC} = 0.85$.

- Note:** Defining the low-cycle criterion is not currently supported in Abaqus/CAE.

Low-cycle Fatigue Criterion

- Example: Low-cycle fatigue prediction for the DCB model**
 - This case consists of the following steps:
 - Step 1: VCCT analysis
 - This step can be used to check whether the peak loading leads to static crack propagation.
 - Step 2: Low-cycle fatigue analysis
 - This step assesses the fatigue life of the DCB model subjected to sub-critical cyclic loading.



Low-cycle Fatigue Criterion

- Partial input:

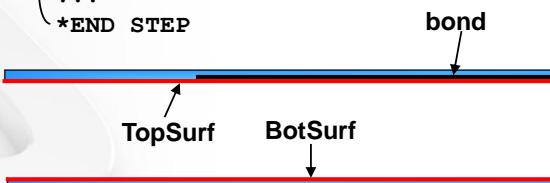
```
...
*CONTACT PAIR, SMALL SLIDING
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf,
MASTER=BotSurf
*FRACTURE CRITERION, TYPE=VCCT,
MIXED MODE BEHAVIOR=BK
280, 280, 280, 2.284
*OUTPUT, FIELD
*CONTACT OUTPUT, SLAVE=TopSurf,
MASTER=BotSurf
BDSTAT, DBT, DBS, OPENBC, CRSTS,
ENRRT
*END STEP
```

Step 1:
VCCT
analysis

© DASSAULT SYSTEMES

```
*STEP, INC=5000
*DIRECT CYCLIC, FATIGUE
0.25,1,,,25,25,,5
,,1000
*DEBOND, SLAVE=TopSurf,
MASTER=BotSurf
*FRACTURE CRITERION, TYPE=FATIGUE,
MIXED MODE BEHAVIOR=BK
0.5,-0.1,4.8768E-6,1.15,,,280,280
280,2.284
*OUTPUT, FIELD
*CONTACT OUTPUT
BDSTAT, DBT, DBS, OPENBC, CRSTS,
ENRRT
...
*END STEP
```

Step 2:
Fatigue
analysis



SIMULIA

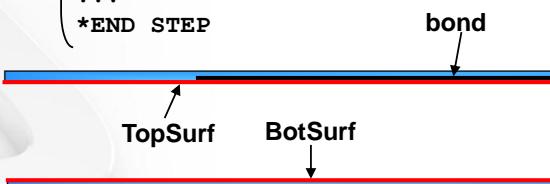
Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

- The procedure to complete the DCB model through the first step (the VCCT analysis) is exactly the same as that discussed in Lecture 7 “VCCT.”
 - Define contact pairs for potential crack surfaces
 - Define initially bonded crack surfaces
 - Activate the crack propagation capability in the first step
 - Specify the VCCT criterion in the first step (a static, general step)
- The details of defining the low-cycle fatigue analysis (the second step) will be discussed next.

```
...
*CONTACT PAIR, SMALL SLIDING
TopSurf, BotSurf
*INITIAL CONDITIONS, TYPE=CONTACT
TopSurf, BotSurf, bond
*STEP, NLGEOM
*STATIC
...
*DEBOND, SLAVE=TopSurf,
MASTER=BotSurf
*FRACTURE CRITERION, TYPE=VCCT,
MIXED MODE BEHAVIOR=BK
280, 280, 280, 2.284
*OUTPUT, FIELD
*CONTACT OUTPUT
BDSTAT, DBT, DBS, OPENBC, CRSTS,
ENRRT
...
*END STEP
```

Step 1:
VCCT
analysis



SIMULIA

Analysis of Composite Materials with Abaqus

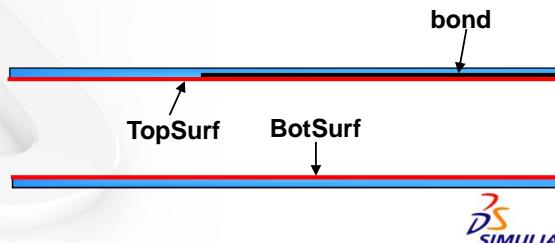
© DASSAULT SYSTEMES

Low-cycle Fatigue Criterion

5 Define the low-cycle fatigue analysis

- The following data are used to define this low-cycle fatigue analysis:
 - Initial time increment: 0.25 sec
 - Time of a single loading cycle: 1 sec
 - Initial number of terms in the Fourier series: 25
 - Maximum number of terms in the Fourier series: 25
 - Maximum number of iterations allowed in the step: 5
 - Total number of cycles allowed in the step: 1000
 - Default values are used for all other entries.

```
...
*STEP, INC=5000
Low-cycle Fatigue Analysis
*DIRECT CYCLIC, FATIGUE
0.25,1,,,25,25,,5
,,1000
```



SIMULIA

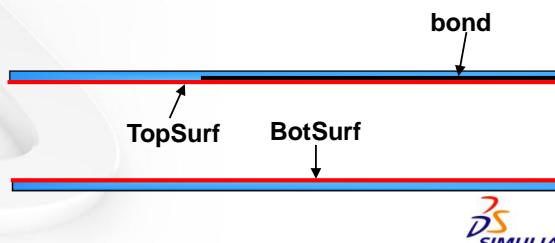
Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

6 Activate the crack propagation capability

- Similar to the VCCT analysis, the *DEBOND option is used to activate the crack propagation in the low-cycle fatigue analysis step.
 - The SLAVE and MASTER parameters identify the surfaces to be debonded.

```
...
*STEP, INC=5000
Low-cycle Fatigue Analysis
*DIRECT CYCLIC, FATIGUE
0.25,1,,,25,25,,5
,,1000
*DEBOND, SLAVE=TopSurf,
MASTER=BotSurf
```



SIMULIA

Analysis of Composite Materials with Abaqus

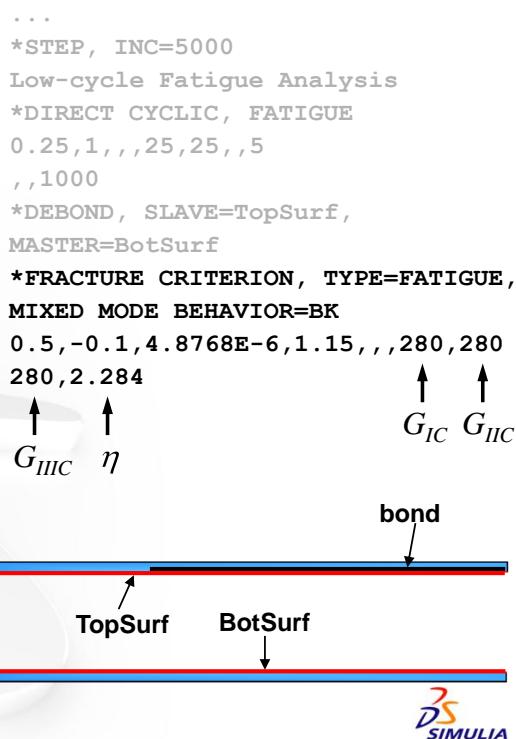
Low-cycle Fatigue Criterion

7 Specify the low-cycle fatigue criterion

- In this model, the material constants are assumed to be the following:
 - $c_1 = 0.5$,
 - $c_2 = -0.1$
 - $c_3 = 4.8768E-6$ $\frac{da}{dn} = c_3 \Delta G^{c_4}$
 - $c_4 = 1.15$
- Note:** The values of these material constants should be determined experimentally.
- The BK model (default) is used.

$$f = \frac{N}{c_1 \Delta G^{c_2}} \geq 1.0$$

$$\frac{da}{dn} = c_3 \Delta G^{c_4}$$

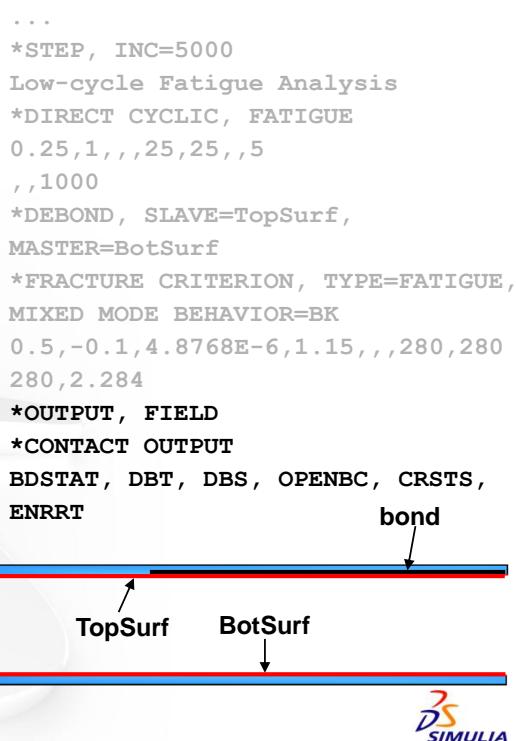


Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

8 Request output

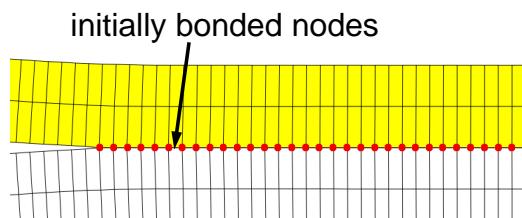
- The output options for the low-cycle fatigue criterion are same as those for the VCCT criterion.



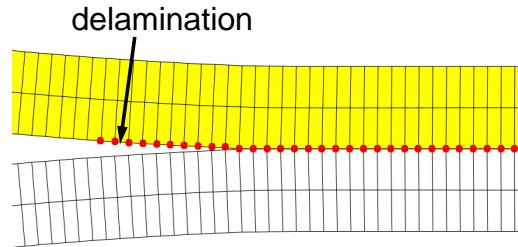
Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

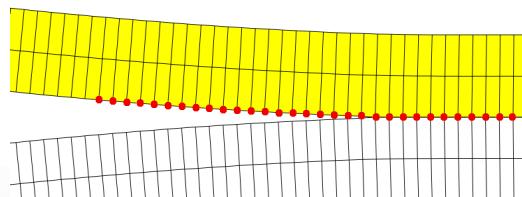
- Results



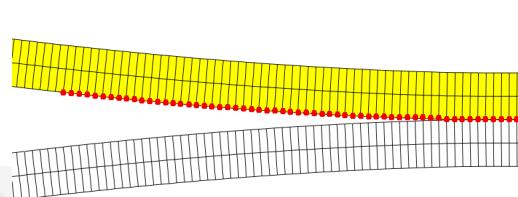
N=1



N=11



N=21



N=51

N is the number of cycles

© DASSAULT SYSTEMES

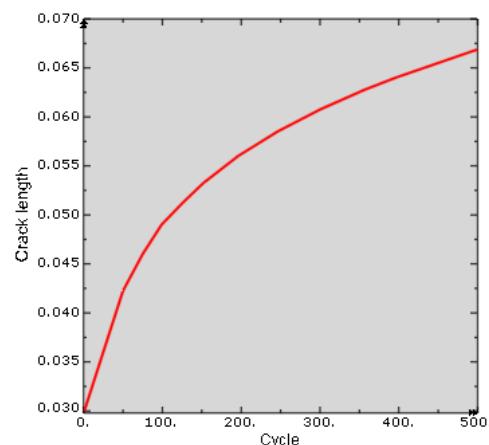
Analysis of Composite Materials with Abaqus

Low-cycle Fatigue Criterion

- More results



delamination growth after 100
loading cycles



crack length vs. cycle
number

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Notes

Notes

Modeling Composite Material Impact with Abaqus/Explicit

Lecture 13

© DASSAULT SYSTEMES

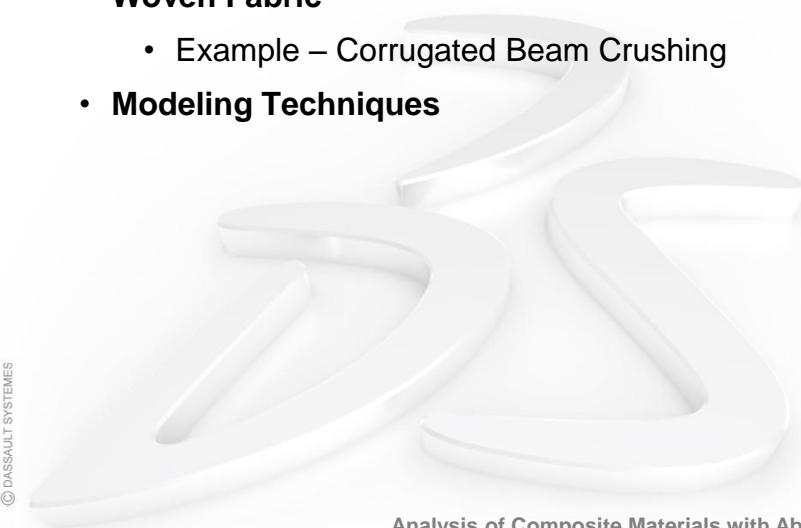


L13.2

Overview

- **Introduction**
- **Composite Damage Models in Abaqus/Explicit**
- **Unidirectional Fiber**
 - Example – Composite Plate Impact
- **Woven Fabric**
 - Example – Corrugated Beam Crushing
- **Modeling Techniques**

© DASSAULT SYSTEMES



Introduction

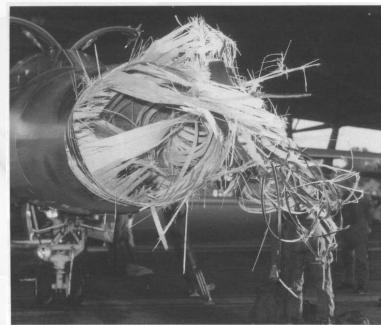
© DASSAULT SYSTEMES



L13.4

Introduction

- Impact resistance of composite materials is of primary importance in many industries
 - Automotive
 - Road debris
 - Vehicle crashworthiness
 - Aerospace
 - Aircraft crashworthiness
 - Bird strike
 - Defense
 - Ballistics



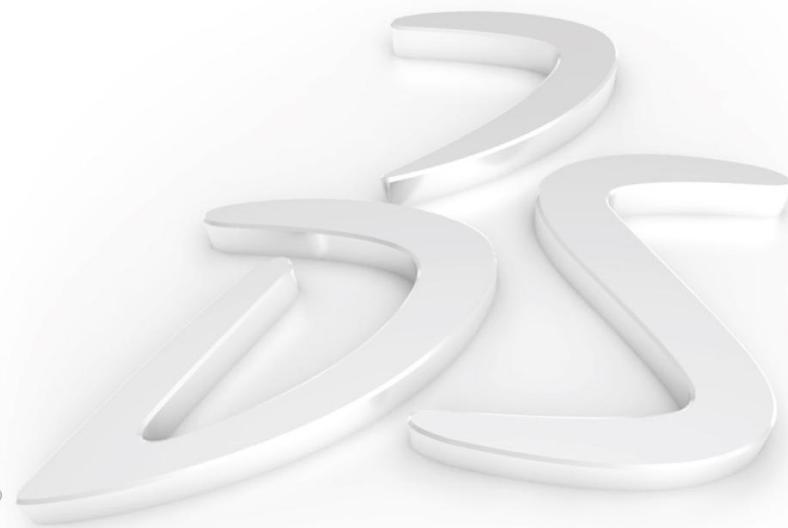
A great danger in F-111 operations at high speed and close to the ground is that of birdstrike.
The remains of A8-138's radome bear testimony to the forces involved. (RAAF Historical)

© DASSAULT SYSTEMES



Composite Damage Models in Abaqus/Explicit

© DASSAULT SYSTEMES



L13.6

Composite Damage Models in Abaqus/Explicit

- Abaqus/Explicit provides additional damage models for fiber-reinforced composite materials.
 - The built-in damage model discussed in Lecture 9 can only be used with elements that have a plane stress formulation.
 - Plane stress, membrane, and shell elements.
 - User-defined material subroutines (VUMATs) are available to extend this capability to elements with other stress states (3D, for example).
- Composite damage VUMATs
 - Two routines are available:
 - Unidirectional fiber VUMAT (extension of built-in capability to include 3D)
 - Available via SIMULIA Answer 3123
 - Woven fabric VUMAT
 - Available as a built-in user subroutine
 - Both of these routines require user input to specify how the composite materials will behave under load.

© DASSAULT SYSTEMES

Unidirectional Fiber VUMAT

© DASSAULT SYSTEMES



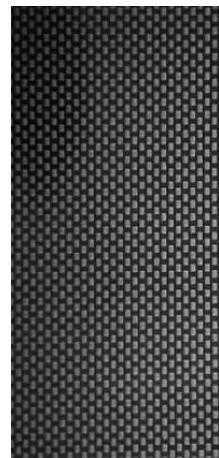
L13.8

Unidirectional Fiber VUMAT

- The primary assumption is that elastic stress/strain relations are given by orthotropic damaged elasticity

$$\begin{pmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{31} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & 2G_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & 2G_{23} & 0 \\ 0 & 0 & 0 & 0 & 0 & 2G_{31} \end{pmatrix} \begin{pmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \varepsilon_{12} \\ \varepsilon_{23} \\ \varepsilon_{31} \end{pmatrix}$$

- Four damage variables are introduced
 - Two associated with fiber tension and compression d_{ft}, d_{fc}
 - Two associated with matrix tension and compression d_{mt}, d_{mc}



© DASSAULT SYSTEMES

Unidirectional Fiber VUMAT

- These four damage variables are used to define global fiber and matrix damage variables.

$$d_f = 1 - (1 - d_{f_f})(1 - d_{f_c})$$

$$d_m = 1 - (1 - d_{m_f})(1 - d_{m_c})$$

- The damaged elastic constants, C_{ij} , are defined in terms of the undamaged elastic constants and the damage variables.

$$C_{11} = (1 - d_f)C_{11}^0$$

$$C_{22} = (1 - d_f)(1 - d_m)C_{22}^0$$

$$C_{33} = (1 - d_f)(1 - d_m)C_{33}^0$$

$$C_{12} = (1 - d_f)(1 - d_m)C_{12}^0$$

$$C_{23} = (1 - d_f)(1 - d_m)C_{23}^0$$

$$C_{13} = (1 - d_f)(1 - d_m)C_{13}^0$$

$$G_{12} = (1 - d_f)(1 - s_{m_f}d_{m_f})(1 - s_{m_c}d_{m_c})G_{12}^0$$

$$G_{23} = (1 - d_f)(1 - s_{m_f}d_{m_f})(1 - s_{m_c}d_{m_c})G_{23}^0$$

$$G_{31} = (1 - d_f)(1 - s_{m_f}d_{m_f})(1 - s_{m_c}d_{m_c})G_{31}^0$$

The factors s_{m_f} and s_{m_c} control the loss of shear stiffness by matrix tensile and compressive failure, respectively.



Unidirectional Fiber VUMAT

- The undamaged elastic constants are functions of the undamaged Young's moduli and Poisson's ratios

$$C_{11}^0 = E_{11}^0(1 - \nu_{23}\nu_{32})\Gamma$$

$$C_{22}^0 = E_{22}^0(1 - \nu_{13}\nu_{31})\Gamma$$

$$C_{33}^0 = E_{33}^0(1 - \nu_{12}\nu_{21})\Gamma$$

$$C_{12}^0 = E_{11}^0(\nu_{21} + \nu_{31}\nu_{23})\Gamma$$

$$C_{23}^0 = E_{22}^0(\nu_{32} + \nu_{12}\nu_{31})\Gamma$$

$$C_{13}^0 = E_{11}^0(\nu_{31} + \nu_{21}\nu_{32})\Gamma$$

$$\Gamma = 1 / (1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{31}\nu_{13} - 2\nu_{21}\nu_{32}\nu_{13})$$



Unidirectional Fiber VUMAT

- 19 user material constants must be specified for this subroutine
 - Young's moduli in the three primary axes
 - $E_{11}^0, E_{22}^0, E_{33}^0$
 - Poisson's ratios
 - $\nu_{12}, \nu_{13}, \nu_{23}$
 - Shear moduli
 - $G_{12}^0, G_{13}^0, G_{23}^0$
 - Shear strengths
 - S_{12}, S_{13}, S_{23}
 - Tensile and compressive failure stress in each primary direction
 - $X_{1t}, X_{1c}, X_{2t}, X_{2c}, X_{3t}, X_{3c}$
 - Damping (optional)
 - β

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Unidirectional Fiber VUMAT

- Data input

```
*MATERIAL, NAME=matName
*DENSITY
ρ
*USER MATERIAL, CONSTANTS=27
**Line 1:
E110,E220,E330,ν12,ν13,ν23,G120,G130
**Line 2:
G230,β,0,0,0,0,0,0
**Line 3:
X1t,X1c,X2t,X2c,X3t,X3c,0,0
**Line 4:
S12,S13,S23
*DEPVAR, DELETE=5
```

Why did the previous slide indicate 19 constants are required?

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Unidirectional Fiber VUMAT

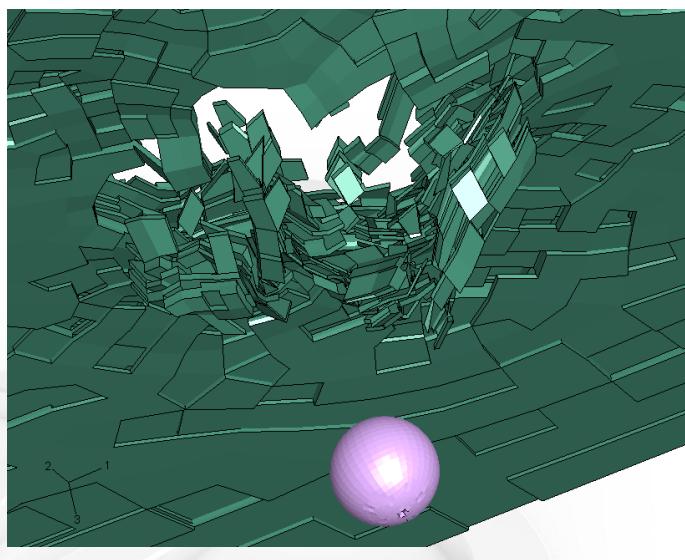
- Output
 - In addition to the standard output variables for stress-displacement elements, the following output variables have a special meaning for this VUMAT:
 - SDV1 Tensile damage along direction 1 (fiber direction)
 - SDV2 Compressive damage along direction 1
 - SDV3 Tensile damage along direction 2 (transverse direction)
 - SDV4 Compressive damage along direction 2 (transverse direction)
 - SDV5 Material point status; 1 if active, 0 if failed
 - SDV6-11 Components of viscous stresses if beta-damping is active
 - SDV12-17 Components of elastic strain tensor

Unidirectional Fiber VUMAT

- Example: Composite laminate plate ballistic impact

Unidirectional Fiber VUMAT

- Results:



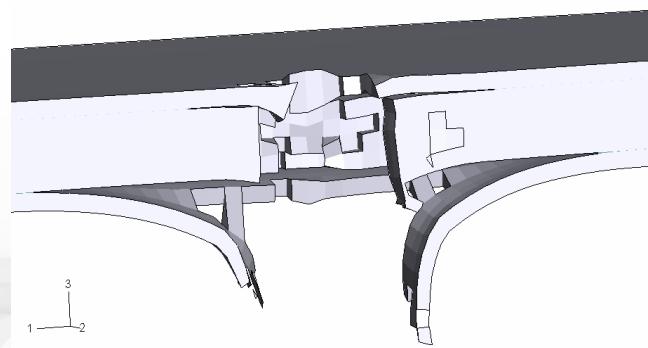
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Unidirectional Fiber VUMAT

- Using cohesive elements for delamination prediction



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

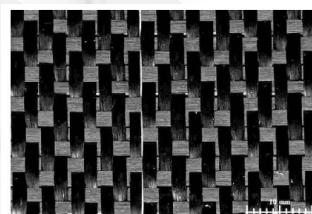
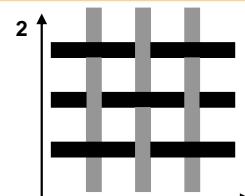
© DASSAULT SYSTEMES



L13.18

Woven Fabric VUMAT

- A schematic of the assumed woven material is shown to the right.
- The fiber directions are assumed to be, *and to remain*, orthogonal (no wrinkling due to shear).
- The constitutive stress-strain relations are formulated in a local Cartesian coordinate system with base vectors aligned with the fiber directions.
- The fabric reinforced ply is modeled as a homogeneous orthotropic elastic material with the potential to sustain progressive stiffness degradation due to fiber/matrix cracking, and plastic deformation under shear loading.



© DASSAULT SYSTEMES



Woven Fabric VUMAT

- It is assumed that the elastic stress-strain relations are given by orthotropic damaged elasticity.

$$\begin{pmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{12} \end{pmatrix} = \begin{pmatrix} \frac{1}{(1-d_1)E_1} & \frac{-\nu_{12}}{E_1} & 0 \\ \frac{-\nu_{21}}{E_2} & \frac{1}{(1-d_2)E_2} & 0 \\ 0 & 0 & \frac{1}{(1-d_{12})2G_{12}} \end{pmatrix} \begin{pmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{pmatrix}$$

- Three global damage variables are used and are associated with fiber fracture along the 1- and 2-directions and matrix micro-cracking due to shear deformation.
- The model differentiates between tensile and compressive fiber failure modes by activating the corresponding damage variable depending on the stress state in the fiber directions

$$d_1 = d_{1+} \frac{\langle \sigma_{11} \rangle}{|\sigma_{11}|} + d_{1-} \frac{\langle -\sigma_{11} \rangle}{|\sigma_{11}|}; \quad d_2 = d_{2+} \frac{\langle \sigma_{22} \rangle}{|\sigma_{22}|} + d_{2-} \frac{\langle -\sigma_{22} \rangle}{|\sigma_{22}|}$$



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- Fiber response
 - The material response along the fiber directions is characterized with damaged elasticity. It is assumed that the fiber damage variables are a function of the corresponding effective stress
 - The criterion for initiation of fiber failure is assumed to take the form
- Shear response
 - The shear response is dominated by the non-linear behavior of the matrix, which includes both plasticity and stiffness degradation due to matrix microcracking
 - The criterion for initiation of shear failure is assumed to take the form

$$\phi_{12} = \frac{\tilde{\sigma}_{12}}{S}$$



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- Element deletion
 - The VUMAT provides an option to delete elements when any one tensile/compressive damage variable along the fiber directions reaches a maximum specified value, or when the plastic strain due to shear deformation reaches a maximum specified value.
- Calibration
 - The elastic constants and the fiber tension/compression strengths are easily measured from standard coupon tests in uniaxial tension/compression loading of 0/90 laminates.
 - The calibration of damage evolution in the fiber failure modes is based on the fracture energy per unit area of the material, which can be measured experimentally.
 - The shear response is usually calibrated with a cyclic tensile test on a $\pm 45^\circ$ laminate, where the strains along the fiber directions can be neglected.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- 26 user material constants must be specified for this subroutine
 - Young's moduli in fiber 1- and 2-directions
 - $E_{1+/-}, E_{2+/-}$
 - Poisson's ratio
 - ν_{12+}, ν_{12-}
 - Shear modulus
 - G_{12}
 - Shear stress at the onset of shear damage
 - S
 - Tensile and compressive strength along fiber directions
 - $X_{1+/-}, X_{2+/-}$
 - Shear equation parameters
 - $\alpha^{12}, d^{12}_{\max}$

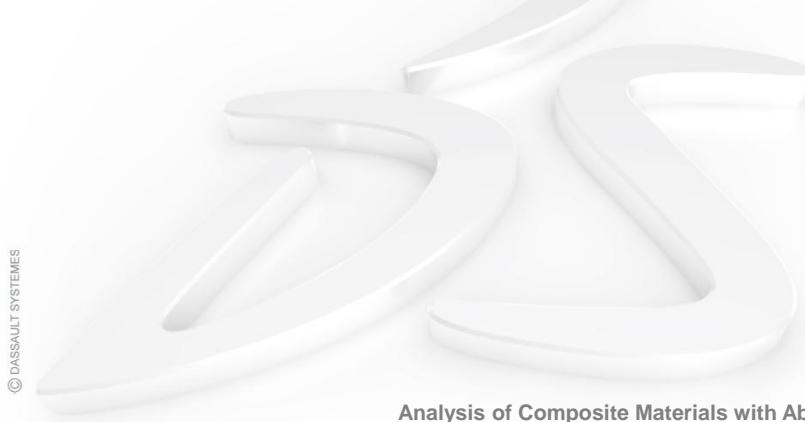
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- Energy per unit area for tensile and compressive fracture along fiber directions
 - $G_f^{1+/-}, G_f^{2+/-}$
- Shear plasticity coefficients
 σ_{y0}, C, p
- Controls for material point failure
 - lDelFlag, d_{\max} , ε_{\max}^{pl} , ε_{\min}

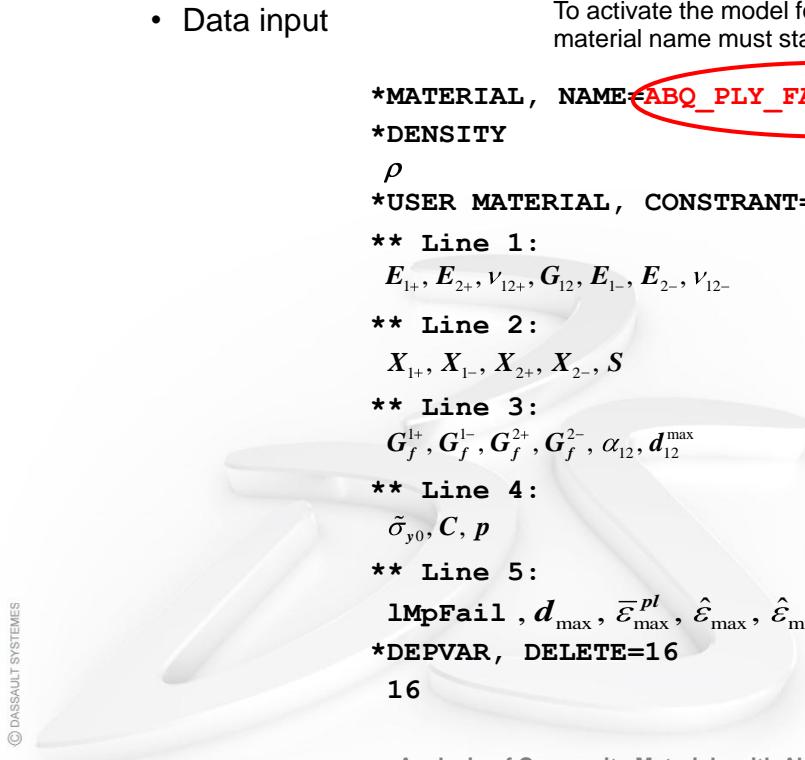


Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- Data input To activate the model for fabric reinforced composites the material name must start with the string **ABQ_PLY_FABRIC**

```
*MATERIAL, NAME=ABQ_PLY_FABRIC_matName
*DENSITY
  ρ
*USER MATERIAL, CONSTRAINT=40
** Line 1:
  E1+, E2+, ν12+, G12, E1-, E2-, ν12-
** Line 2:
  X1+, X1-, X2+, X2-, S
** Line 3:
  Gf1+, Gf1-, Gf2+, Gf2-, α12, d12max
** Line 4:
  σy0, C, p
** Line 5:
  lMpFail, dmax, εmaxpl, ε̂max, ε̂min
*DEPVAR, DELETE=16
```



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- Output
 - In addition to the standard output variables for stress-displacement elements, the following output variables have a special meaning for this VUMAT:

• SDV1	Tensile damage along fiber direction 1
• SDV2	Compressive damage along fiber direction 1
• SDV3	Tensile damage along fiber direction 2
• SDV4	Compressive damage along fiber direction 2
• SDV5	Shear damage
• SDV6	Tensile damage threshold along fiber direction 1
• SDV7	Compressive damage threshold along fiber direction 1
• SDV8	Tensile damage threshold along fiber direction 2
• SDV9	Compressive damage threshold along fiber direction 2
• SDV10	Shear damage threshold

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Woven Fabric VUMAT

- Output (cont'd)
 - SDV11 Equivalent plastic strain
 - SDV12 Elastic strain component 11
 - SDV13 Elastic strain component 22
 - SDV14 Not used
 - SDV15 Elastic strain component 12
 - SDV16 Material point status: 1 if active, 0 if failed

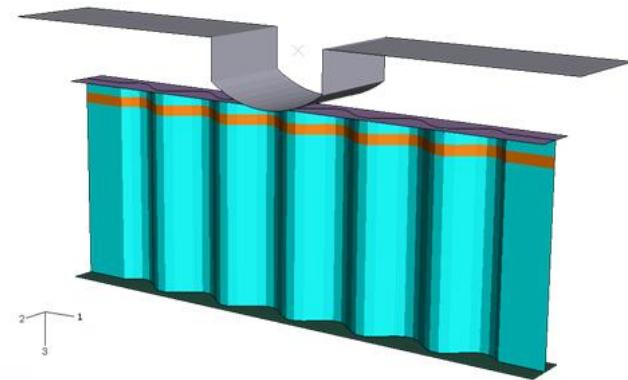
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Woven Fabric VUMAT

- Example: Composite woven fabric beam crush



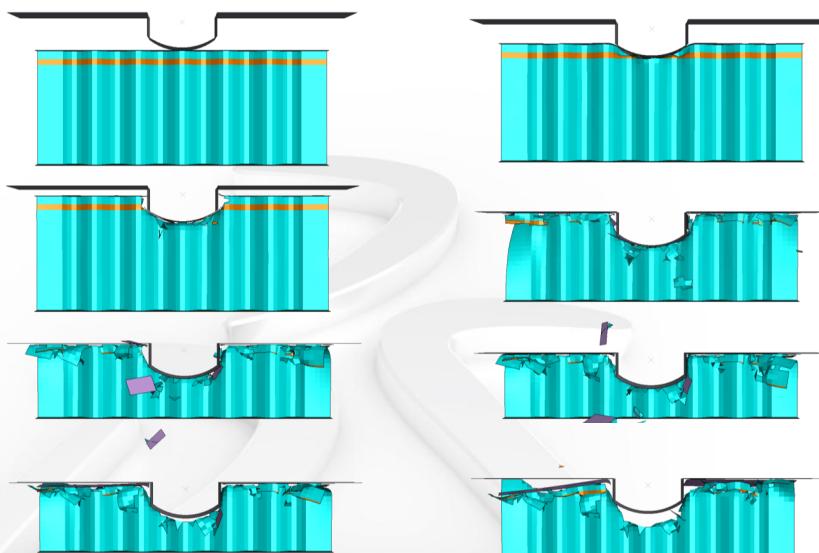
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Woven Fabric VUMAT

- Results:



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Techniques

© DASSAULT SYSTEMES



L13.30

Modeling Techniques

• Stable time increment

- For Abaqus/Explicit, the concept of the stable time increment is important to understand

$$\Delta t \leq \min\left(L_e \sqrt{\frac{\rho}{\hat{\lambda} + 2\hat{\mu}}}\right); \quad \hat{\lambda} = \frac{E\nu}{(1+\nu)(1-2\nu)}; \quad \hat{\mu} = \frac{E}{2(1+\nu)}$$

- When modeling composites at the laminate level, very small element dimensions can be encountered
- This can lead to a very small stable time increment, requiring a large number of increments to complete an analysis
 - For a typical composite where $L_e=0.2$ mm, $\rho=1.5e-9$, $E=65$ GPa, and $\nu=0.3$, Δt is on the order of 1e-10 seconds.
 - To simulate 1 millisecond will require 10 million increments!
- Some ideas for dealing with this follow.

© DASSAULT SYSTEMES



Modeling Techniques

- **Stable time increment**

- Use double precision if the number of increments will exceed 300,000
- To attempt a more time economical solution, the following may be helpful
 - Mass scaling can be used to speed up the simulation for the purposes of checking model setup
 - Dynamic response will be affected
 - Group layers of common material orientation together and model as one layer
 - Use continuum macroscopic (i.e., anisotropic global) properties to model the composite instead of using a mixed modeling technique

© DASSAULT SYSTEMES

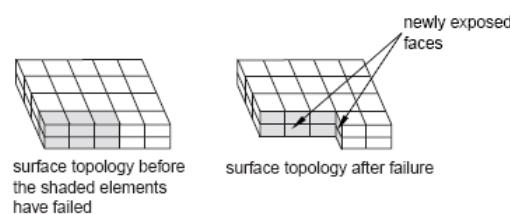


Analysis of Composite Materials with Abaqus

Modeling Techniques

- **Interior surfaces for erosion**

- If erosion will be considered in an impact analysis, care must be taken when defining contact
- Interior mesh surfaces must be included in the contact definitions



- The inclusion of interior faces is not currently supported in Abaqus/CAE, and requires a manual edit to the input file. For example:

```
*SURFACE, NAME=interior_elems, TYPE=ELEMENT
all_elems, interior
```

- This creates a surface named `interior_elems` consisting of the interior faces of the element set `all_elems`, which can now be used in the contact domain

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Notes

Notes

Crack Propagation Analysis using the Debond Capability

Appendix 1

© DASSAULT SYSTEMES



A1.2

Overview

- Introduction
- Modeling Interface Behavior

© DASSAULT SYSTEMES



Introduction

© DASSAULT SYSTEMES



A1.4

Introduction

- In this technique the matrix and reinforcement material are both modeled as deformable continua; sometimes the reinforcement is modeled as rigid.
- The material properties of the constituent material are elastic or inelastic.
- The interface can be fully bonded or (partially) detached.
- The deformation is usually inhomogeneous.
- Nonlinear interactions can be modeled at the interface.
 - Microscopic analysis is used to:
 - simulate fundamental interactions in the material,
 - study composite failure mechanisms, and/or
 - obtain macroscopic linear and nonlinear characteristics of the composite via “unit cell” calculations.

© DASSAULT SYSTEMES

Modeling Interface Behavior

© DASSAULT SYSTEMES

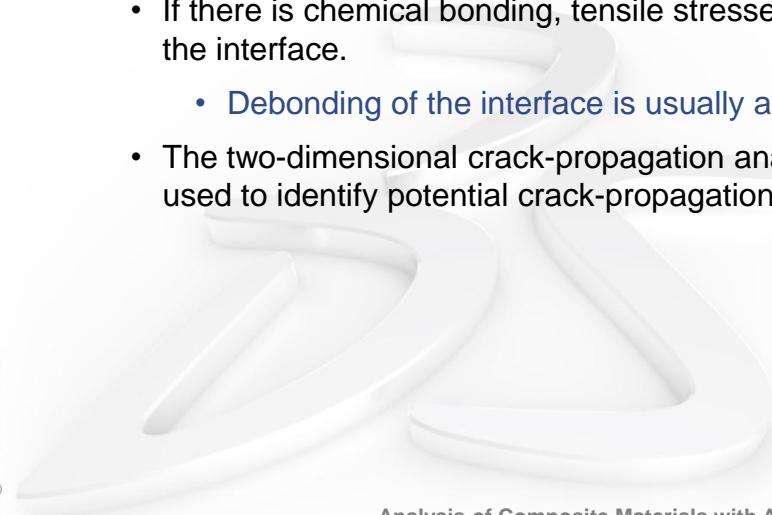


A1.6

Modeling Interface Behavior

- In the nonlinear microscopic modeling of composites, proper modeling of the interface behavior is essential.
- In the absence of chemical bonding between matrix material and reinforcement, no tensile stresses are transferred across the interface.
 - Contact pairs are then used, often with Coulomb friction.
- If there is chemical bonding, tensile stresses can be transferred across the interface.
 - Debonding of the interface is usually a primary concern.
- The two-dimensional crack-propagation analysis capability in Abaqus is used to identify potential crack-propagation surfaces.

© DASSAULT SYSTEMES



Modeling Interface Behavior

- **Identification of partially bonded surfaces**
 - The *INITIAL CONDITIONS, TYPE=CONTACT option is used to identify surfaces that may debond.
 - Constraint forces are applied at the slave nodes to keep them fully bonded to their master surface.
 - The *DEBOND option is then used to specify that crack propagation may occur between these surfaces.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Interface Behavior

- **Debonding of the crack-tip node**
 - The crack-tip node debonds when the failure criterion, f , reaches the value 1.0 within a certain tolerance, f_{tol} : $1 - f_{tol} \leq f \leq 1 + f_{tol}$.
 - When debonding is detected at a node, the nodal traction initially carried between the surfaces is ramped down to zero.
 - At that point the node is fully debonded.
 - Once the surfaces have debonded, they can still interact: they may close again under certain loadings.
 - Frictional effects between them may be included in the modeling.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Interface Behavior

- Crack-propagation criteria

- The user specifies which criterion to use by means of the *FRACTURE CRITERION option, which is a required suboption of the *DEBOND option.
- There are three available criteria that are mutually exclusive for a given contact pair.
 - Critical stress criterion
 - Crack opening displacement criterion
 - Crack length vs. time criterion

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Modeling Interface Behavior

- Critical stress criterion

- The failure index is defined as

$$f = \sqrt{\left(\frac{\max(\sigma_n, 0)}{\sigma^f}\right)^2 + \left(\frac{\tau_1}{\tau_1^f}\right)^2 + \left(\frac{\tau_2}{\tau_2^f}\right)^2},$$

where σ_n is the stress normal to the interface surface and τ_1 and τ_2 are the shear stresses on the interface. σ^f , τ_1^f , and τ_2^f are failure values.

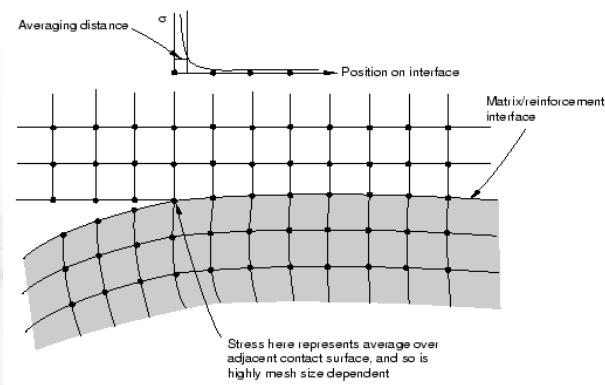
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Modeling Interface Behavior

- The critical stress criterion must be used with care.
 - Since a crack tip gives rise to a singularity in the solution, the stress at the crack tip in a finite element model is highly dependent on the mesh size: it represents some average of the stress in the first fully bonded surface segment behind the crack tip.



- As long as this effect is understood, the approach can provide useful modeling.

© DASSAULT SYSTEMES

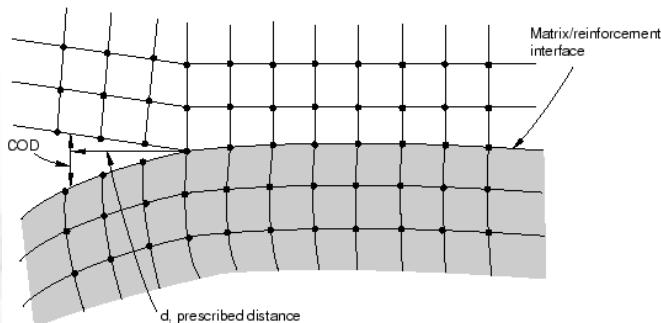


Analysis of Composite Materials with Abaqus

Modeling Interface Behavior

• Crack opening displacement criterion

- In this case Abaqus will measure the opening between the faces of the debonded part of the surface at a fixed distance behind the current crack tip and allow the crack to propagate when this opening reaches a given value.



- This criterion generally is considered suitable for certain debonding problems.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Interface Behavior

- **Crack length vs. time criterion**

- Debonding is specified as a known function of time: $l = l(t)$, where l is the crack length and t is time.
- This criterion is mainly used for comparison to experimental results (which define the crack length–load–time relationship).

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Interface Behavior

- **Other criteria**

- It is also possible to create very detailed debond models via user elements or by using thin continuum elements with a user material.
- The following example uses this technique to study the debonding of a soft matrix material from a rigid inclusion.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Interface Behavior

- Example of microscopic composite analysis

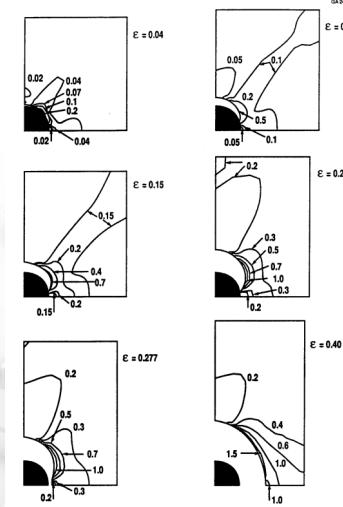


Fig. 11. Plastic strain contours during void nucleation for $l = 0.01r$.

From *Abaqus User Subroutines for Material Modeling*,
R. E. Smelser and R. Becker, ABAQUS Users' Conference, 1989

Notes

Notes

Cohesive Element Modeling Techniques

Appendix 2

© DASSAULT SYSTEMES



A2.2

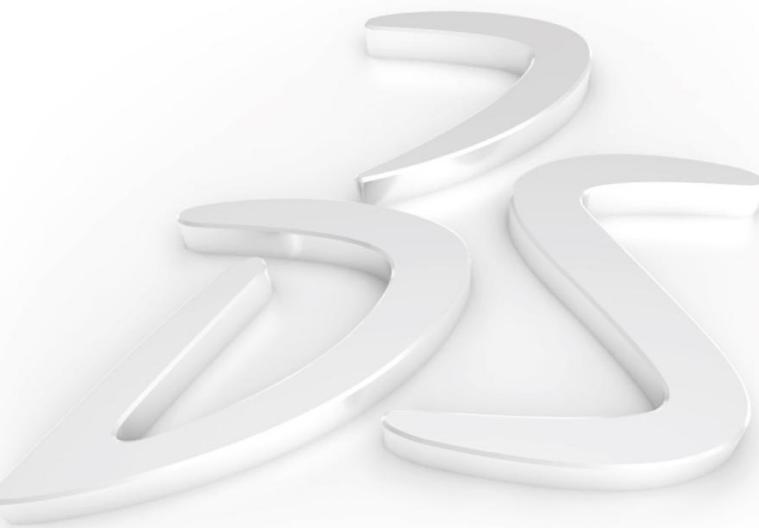
Overview

- Viscous Regularization
- Modeling Techniques

© DASSAULT SYSTEMES



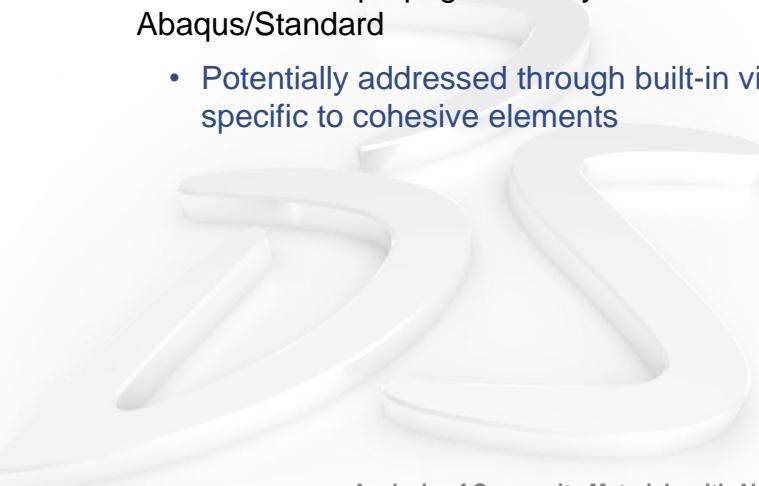
Viscous Regularization



A2.4

Viscous Regularization

- **Cohesive elements have the potential to cause numerical difficulties in the following cases**
 - Stiff cohesive behavior may lead to reduced maximum stable time increment in Abaqus/Explicit
 - Potentially addressed through selective mass scaling
 - Unstable crack propagation may lead to convergence difficulties in Abaqus/Standard
 - Potentially addressed through built-in viscous regularization option specific to cohesive elements



Viscous Regularization

- **Viscous regularization**

- Material models with damage often lead to severe convergence difficulties in Abaqus/Standard
- Viscous regularization helps in such cases
 - Helps make the consistent tangent stiffness of softening material positive for sufficiently small time increments
- Similar approach used in the concrete damaged plasticity model in Abaqus/Standard

$$\sigma = (1 - d_v) \bar{\sigma}$$

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

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Viscous Regularization

- **Consistent material tangent stiffness**

$$D = (1 - d) K_0 - f \frac{\partial d}{\partial \epsilon} \bar{\sigma} \otimes \bar{\sigma}$$

K_0 is the undamaged elastic stiffness

f is a factor that depends on the details of the damage model

- **Viscous regularization ensures that when $\frac{\Delta t}{\mu} \rightarrow 0$, $D = (1 - d) K_0$**

- “Offending” second term is eliminated when the analysis cuts back drastically

© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Viscous Regularization

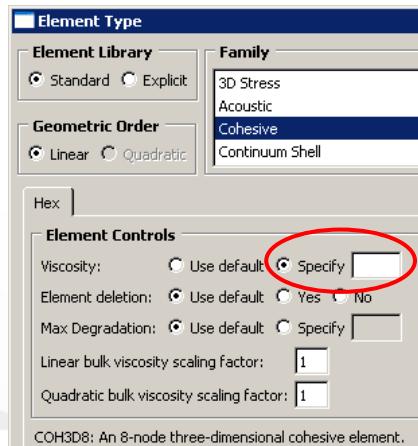
- User interface for viscous regularization

*COHESIVE SECTION, CONTROLS = control1
 *SECTION CONTROLS, NAME = control1,
 VISCOSITY = factor

- Add-on transverse shear stiffness may provide additional stability

*COHESIVE SECTION

*TRANSVERSE SHEAR STIFFNESS



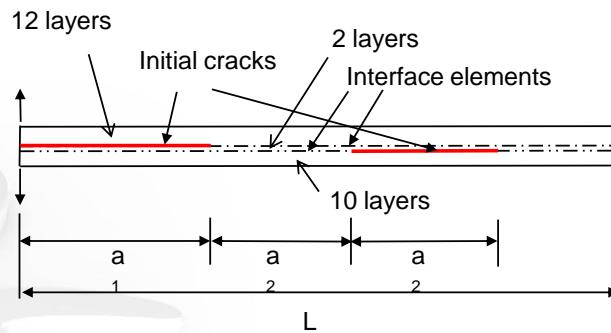
- Output

- Energy associated with viscous regularization: ALLCD

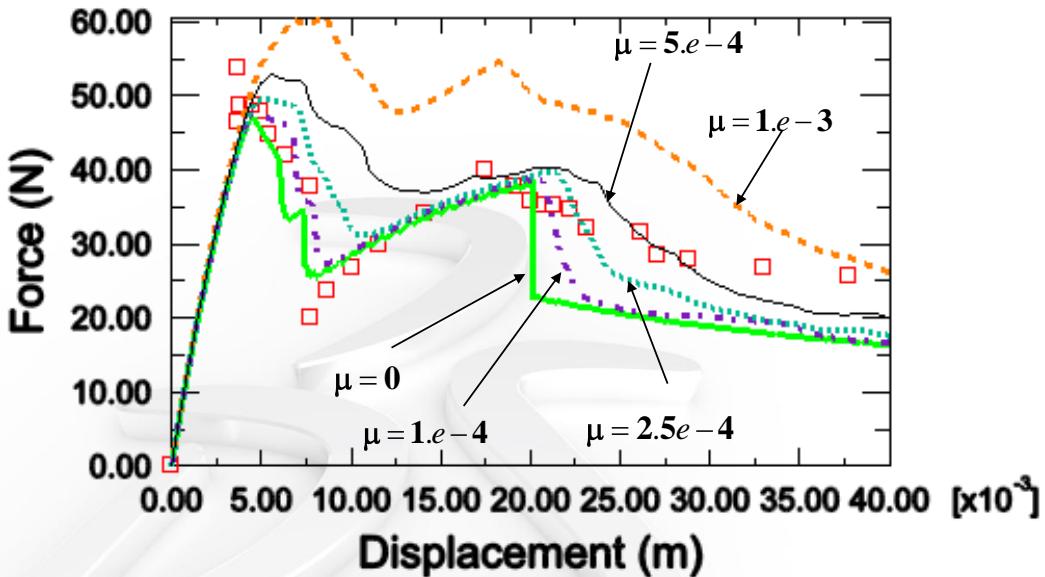
Viscous Regularization

- Example: Multiple delamination problem (Alfano & Crisfield, 2001)

- Industry standard Alfano-Crisfield nonsymmetric delamination examples
- Plies are initially bonded with predefined cracks, then peeled apart in a complex sequence
- Example done in Abaqus/Standard and Abaqus/Explicit
- Effect of viscous regularization is investigated



Viscous Regularization



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Viscous Regularization

- Effect of viscous regularization on convergence of multiple delamination problem:
 - Significant improvements with small regularization factor

Viscous regularization factor	Total number of increments
0.	375
$1.0e-4$	171
$2.5e-4$	153
$1.0e-3$	164

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Techniques

© DASSAULT SYSTEMES



A2.12

Modeling Techniques

- **Model problem: double-cantilever beam**
 - Alfano and Crisfield (2001)
 - Pure Mode I
 - Displacement control
 - Analyzed using
 - 1D (B21),
 - 2D (CPE4I), and
 - 3D (C3D8I) elements
 - Delamination assumed to occur along a straight line
 - Beams: Orthotropic material
 - Cohesive layer: Traction-separation with damage

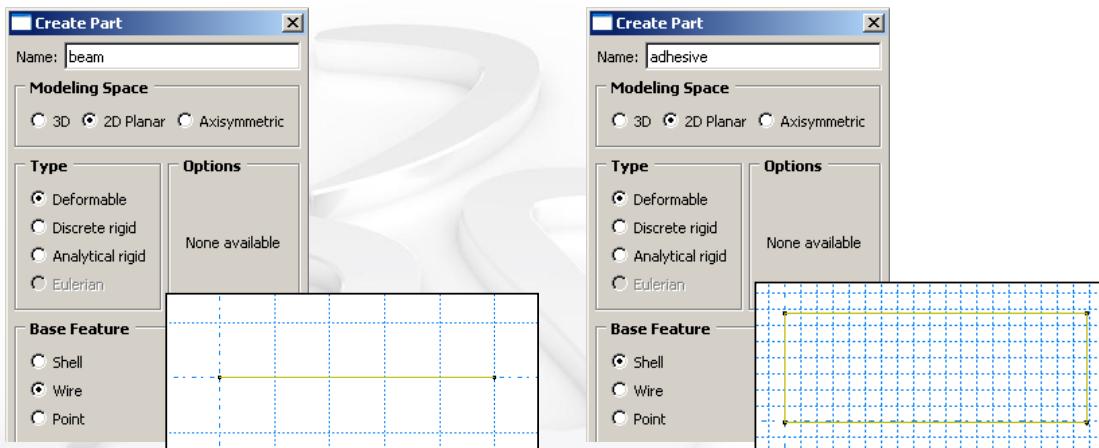


© DASSAULT SYSTEMES



Modeling Techniques

- One-dimensional model
 - Use tie constraints between the cohesive layer and the beams
 - Require distinct parts for the beam and cohesive zone geometry
- Geometry



Analysis of Composite Materials with Abaqus

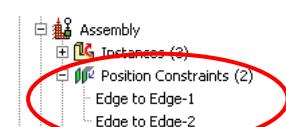
© DASSAULT SYSTEMES

Modeling Techniques

- One-dimensional model (cont'd)
 - Assembly

Create 2 instances of the beam;
one of the cohesive zone

Position the parts to leave gaps
between them; this will later
facilitate picking surfaces



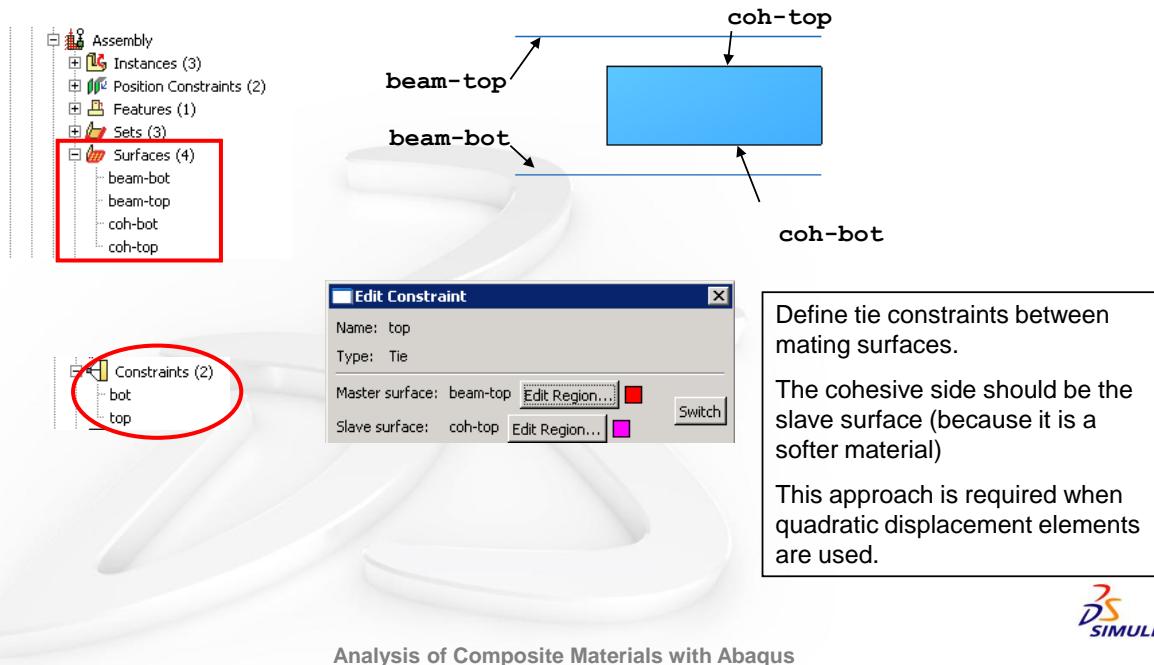
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Modeling Techniques

- One-dimensional model (cont'd)

- Tie constraints

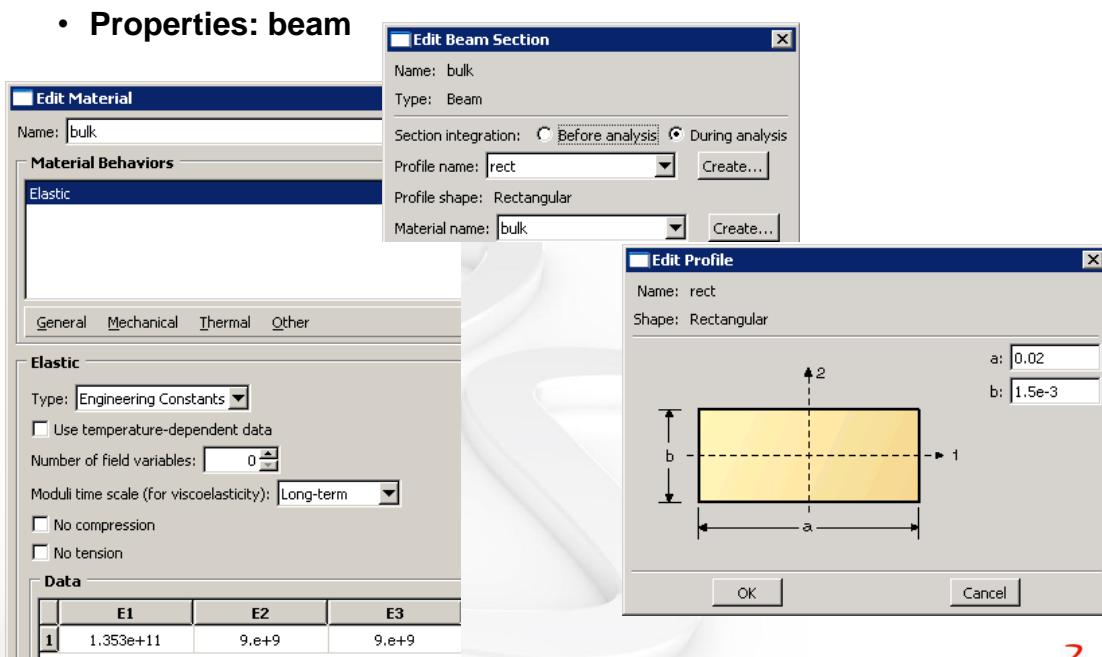


Analysis of Composite Materials with Abaqus

Modeling Techniques

- One-dimensional model (cont'd)

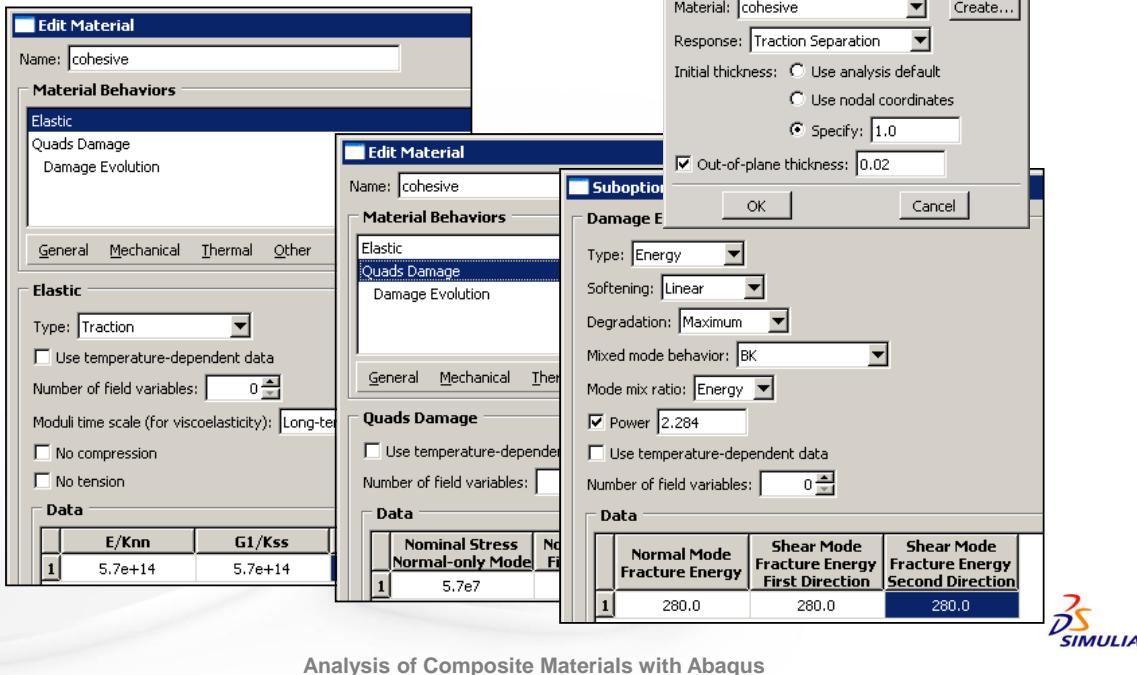
- Properties: beam



Analysis of Composite Materials with Abaqus

Modeling Techniques

- One-dimensional model (cont'd)
 - Properties: adhesive

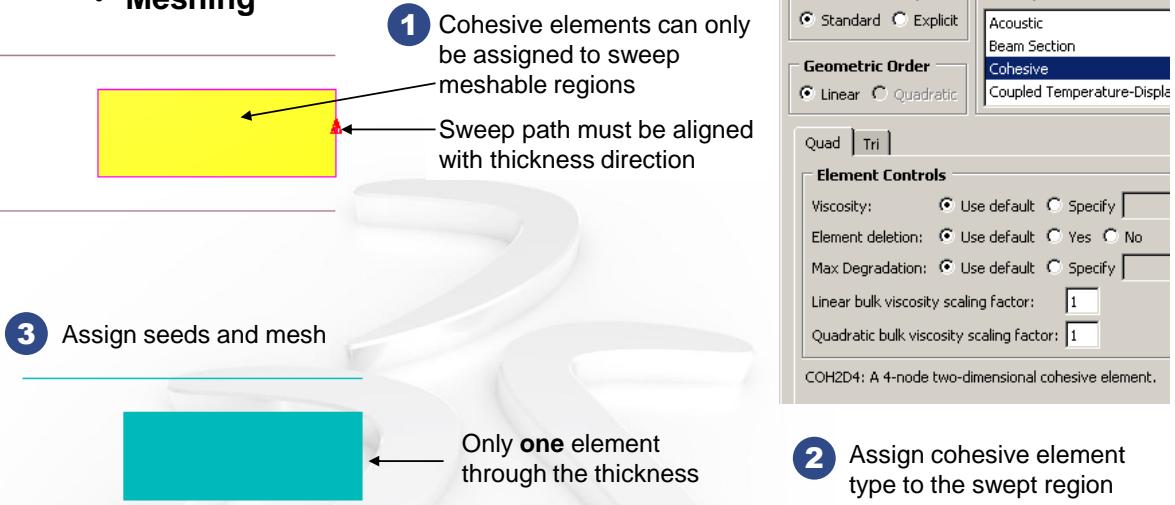


© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

Modeling Techniques

- One-dimensional model (cont'd)
 - Meshing



© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus

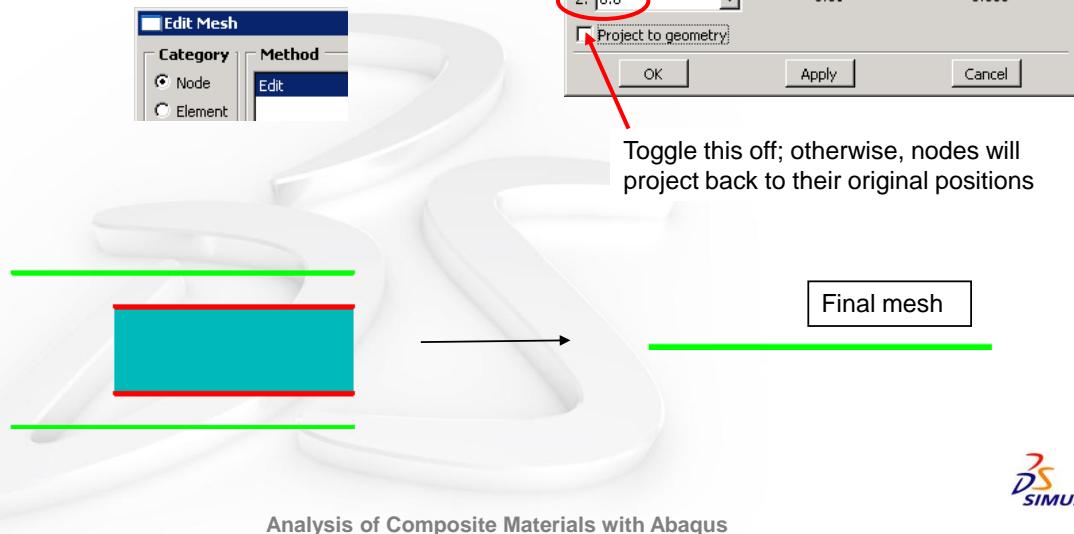
Modeling Techniques

• One-dimensional model (cont'd)

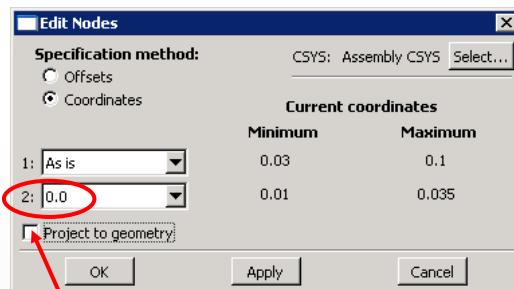
- Meshing (cont'd)

- 4 Edit the nodal coordinates of each part instance so that they all have the same 2-coordinate

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus



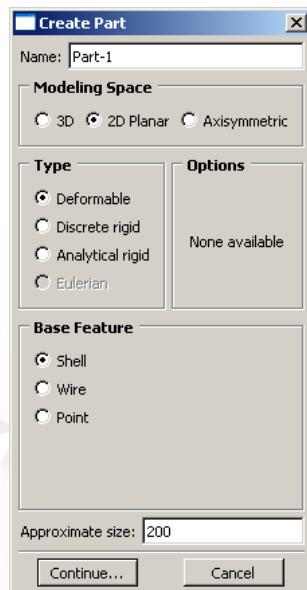
Toggle this off; otherwise, nodes will project back to their original positions

Modeling Techniques

• Two-dimensional model

- All geometry is 2D and planar
- Properties, attributes, etc. treated in a similar manner to the 1D case presented earlier
- Modeling options include:
 - Shared nodes
 - Tie constraints
 - Similar to the 1D model

© DASSAULT SYSTEMES



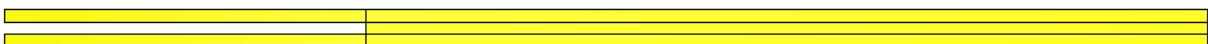
Analysis of Composite Materials with Abaqus

Modeling Techniques

- Two-dimensional model (cont'd)

- Shared nodes

- Define a finite thickness slit in the beam as shown below



- Use the actual overall thickness of the DCB
- The center region represents the cohesive layer

- Mesh the part:



© DASSAULT SYSTEMES



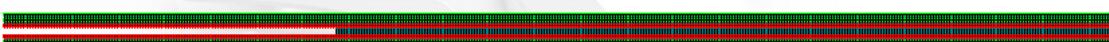
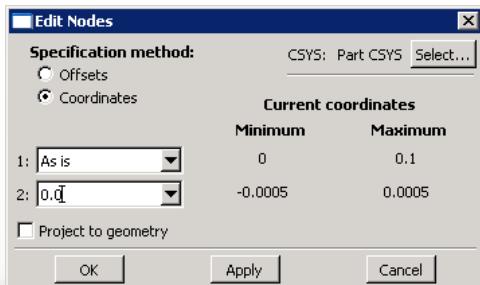
Analysis of Composite Materials with Abaqus

Modeling Techniques

- Two-dimensional model (cont'd)

- Shared nodes (cont'd)

- Edit the coordinates of the nodes along the interface



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Techniques

- Two-dimensional model (cont'd)

- Tie constraints

- 1 Create two instances of the beams and position them as shown below.



- Suppress the visibility of the instances to facilitate picking surfaces, etc.

- 2 Create a finite thickness cohesive layer, position it appropriately in the horizontal direction, define surfaces, etc.

- After meshing, adjust the coordinates of all the nodes in the cohesive layer so that they lie along the interface between the two beams.

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Techniques

- Three-dimensional model

- All geometry is 3D
 - Solid geometry for beams
 - Solid or shell geometry for cohesive layer
 - Modeling options include
 - Shared nodes
 - Tie constraints

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Techniques

- Three-dimensional model (cont'd)
 - Shared nodes

- 1 Partition the geometry and define a mesh seam between these two faces

© DASSAULT SYSTEMES

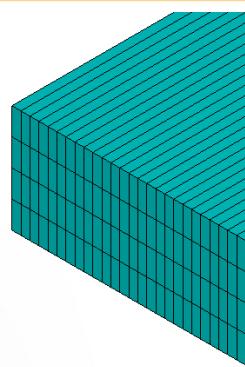


Analysis of Composite Materials with Abaqus

Modeling Techniques

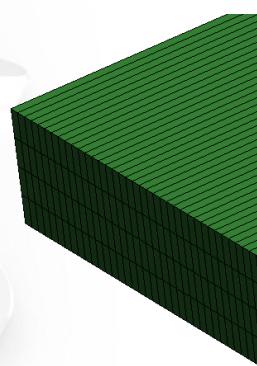
- Three-dimensional model (cont'd)
 - Shared nodes (cont'd)

- 2 Mesh the part with solid (continuum elements).



- 3 Create a orphan mesh

Mesh → Create Mesh Part



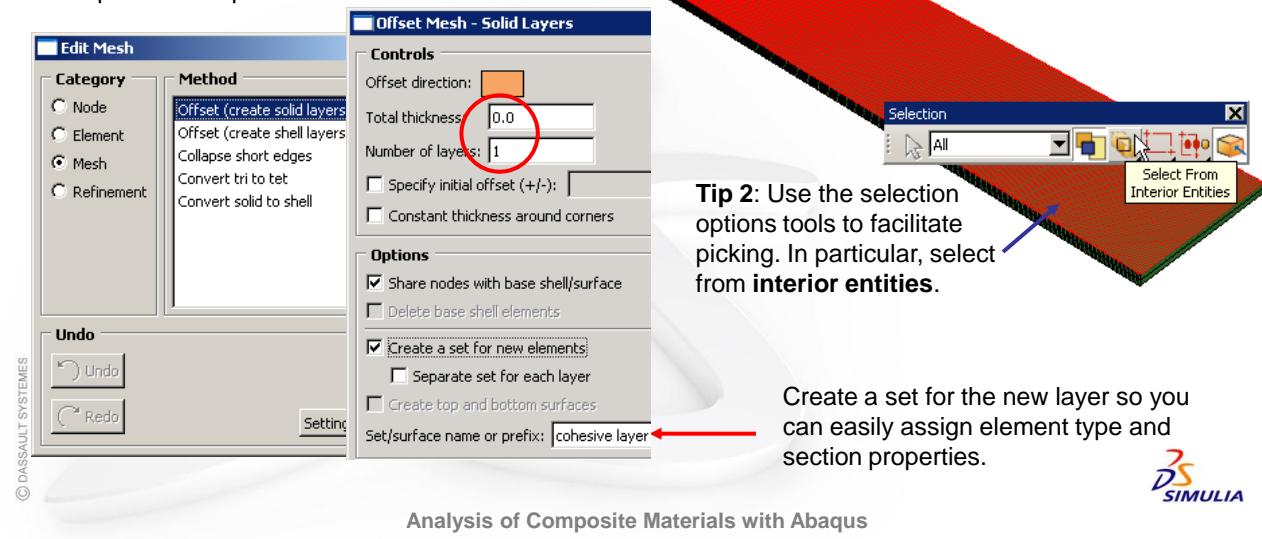
© DASSAULT SYSTEMES

Analysis of Composite Materials with Abaqus



Modeling Techniques

- 4** Create a single **zero-thickness solid layer** by offsetting from the midplane (**selected by angle**) of the orphan mesh created in the previous step



Tip 1: Remove elements from top region with display groups (select by angle)

Tip 2: Use the selection options tools to facilitate picking. In particular, select from **interior entities**.

Create a set for the new layer so you can easily assign element type and section properties.

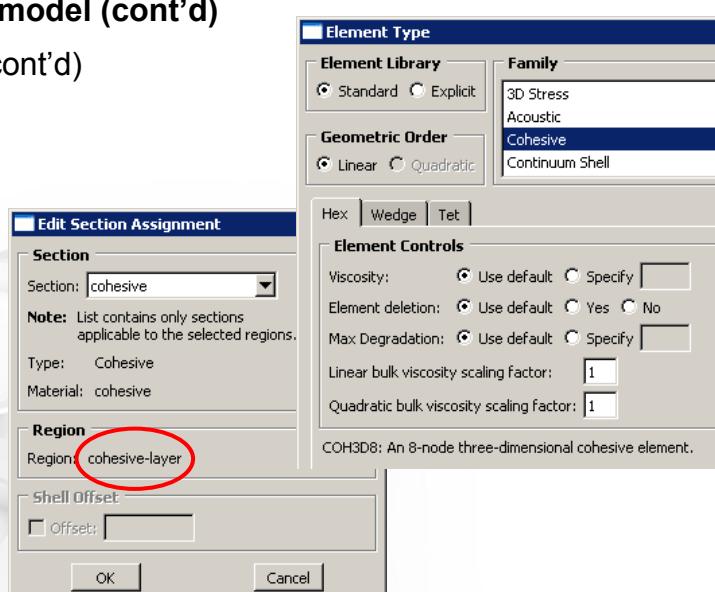
SIMULIA

Analysis of Composite Materials with Abaqus

Modeling Techniques

- Three-dimensional model (cont'd)
 - Shared nodes (cont'd)

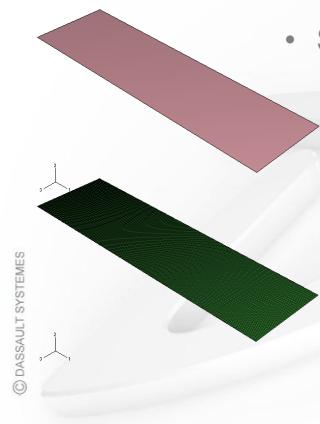
- 5** Assign section properties and the element type to the set created in the previous step



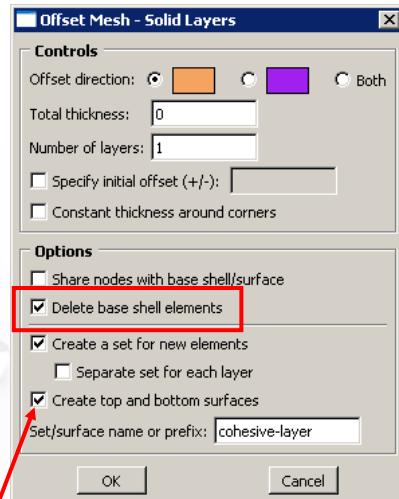
COH3D8: An 8-node three-dimensional cohesive element.

Modeling Techniques

- Three-dimensional model (cont'd)
 - Tie constraints
 - The cohesive region can be defined as
 - Solid (with finite thickness)
 - Edit nodal coordinates of cohesive elements as in previous examples
 - Shell geometry
 - Mesh geometry then create orphan mesh
 - Offset a zero-thickness layer of solid elements from the orphan mesh



Define surfaces automatically to facilitate tie constraints

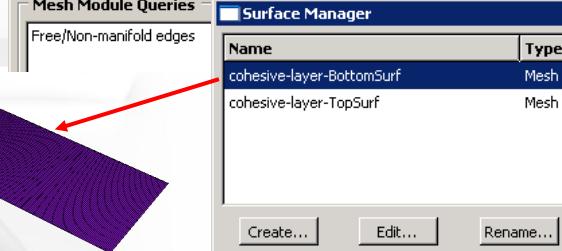
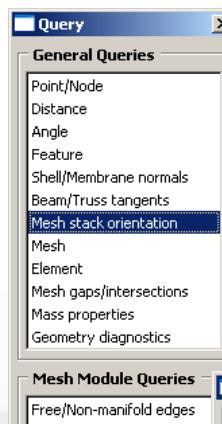
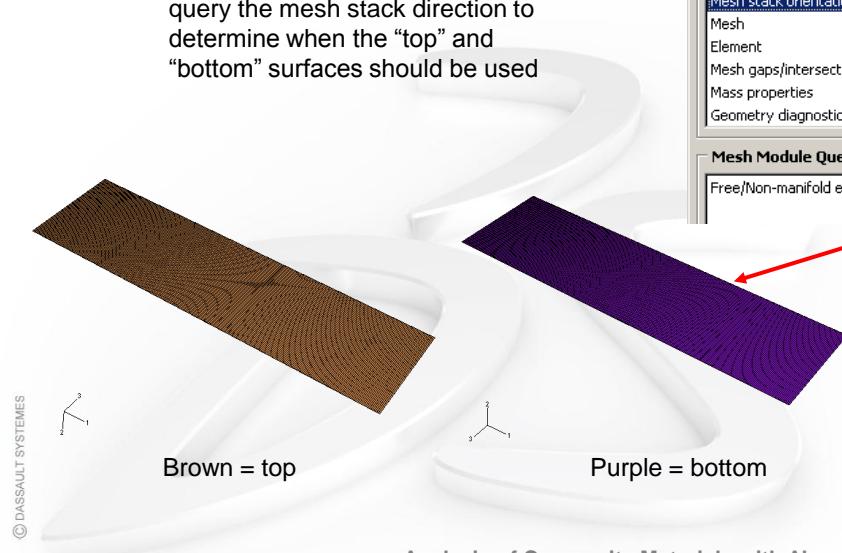


Analysis of Composite Materials with Abaqus

Modeling Techniques

- Three-dimensional model (cont'd)
 - Tie constraints (cont'd)

When defining the tie constraints, query the mesh stack direction to determine when the "top" and "bottom" surfaces should be used

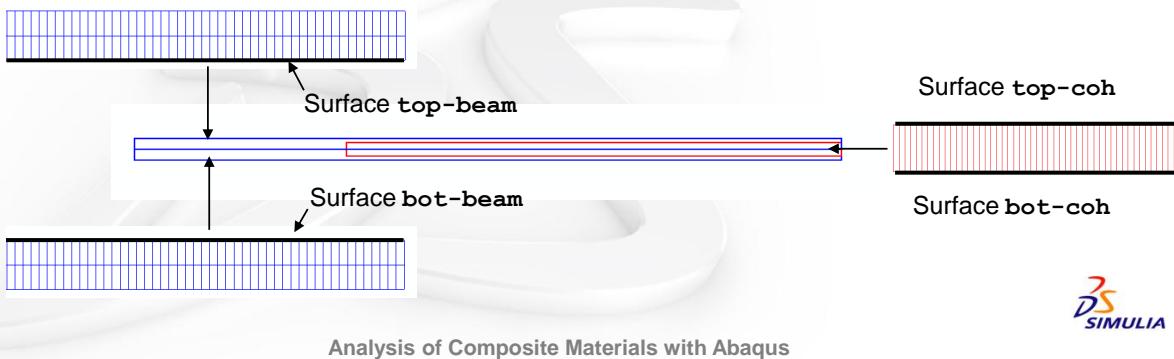


Analysis of Composite Materials with Abaqus

Modeling Techniques

- What if I don't use Abaqus/CAE?

- In this case do the following in the preprocessor of your choice:
 1. Generate the mesh for the structure and cohesive layer (temporarily assigning an arbitrary element type to the cohesive layer)
 2. Position the layer of cohesive elements over the interface
 3. Define surfaces on the structure and cohesive layer
 4. Write the input file



© DASSAULT SYSTEMES

Modeling Techniques

- Edit the input file:

5. Change the element type assigned to the cohesive layer

```
*element, elset=coh, type=coh2d4
```

6. Assign cohesive section properties

```
*cohesive section, elset=coh, material=cohesive,
response=traction separation, stack direction=2, controls=visco
  1.0, 0.02
  :
*material, name=cohesive
*elastic, type=traction
  5.7e+14, 5.7e+14, 5.7e+14
*damage initiation, criterion=quads
  5.7e7, 5.7e7, 5.7e7
*damage evolution, type=energy, mixed mode behavior=bk, power=2.284
  280.0, 280.0, 280.0
```

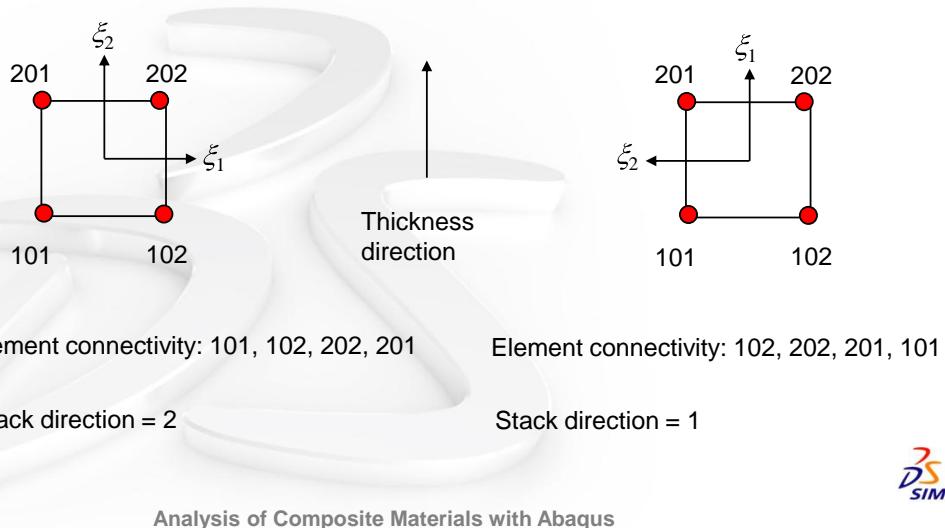
© DASSAULT SYSTEMES



Modeling Techniques

- The stack direction defines the thickness direction based on the element isoparametric directions.
- Set STACK DIRECTION = { 1 | 2 | 3 } to define the element thickness direction along an isoparametric direction.
- 2D example (extends to 3D):

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

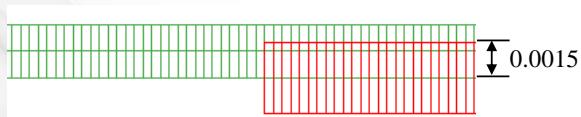
Modeling Techniques

- Edit the input file (cont'd):
- 7. Define tie constraints between the surfaces

```
*tie, name=top, adjust=yes, position tolerance=0.002
Cohesive surface
is the slave
    top-coh, top-beam
*tie, name=bot, adjust=yes, position tolerance=0.002
    bot-coh, bot-beam
```

Setting **adjust=yes** will force Abaqus to move the slave (cohesive element) nodes onto the master surface. By adjusting both the top and bottom cohesive surfaces in this way, a zero-thickness cohesive layer is produced.

The position tolerance should be large enough to contain the slave nodes when measured from the master surface. In this case the overclosure is equal to 0.0015 on either side of the interface so a position tolerance of 0.002 is sufficient to capture all slave nodes.



© DASSAULT SYSTEMES

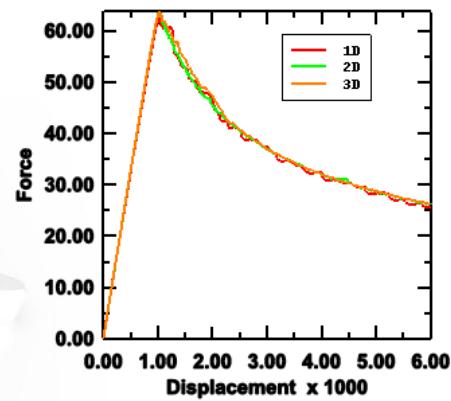
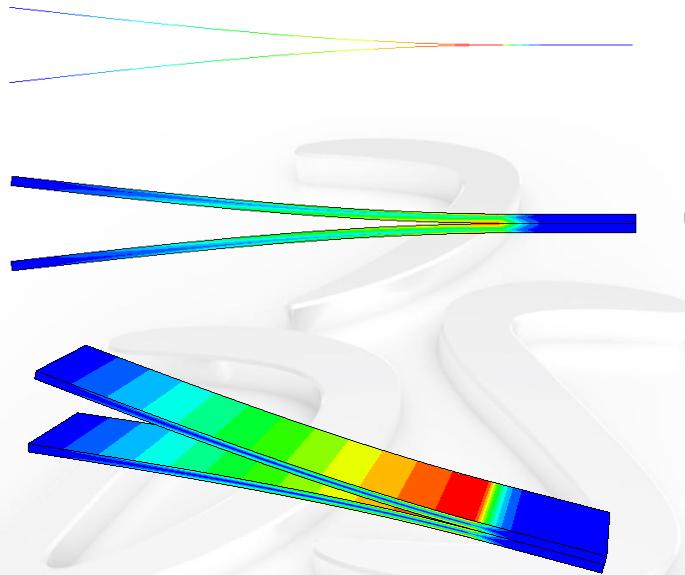


Analysis of Composite Materials with Abaqus

Modeling Techniques

- Results

© DASSAULT SYSTEMES

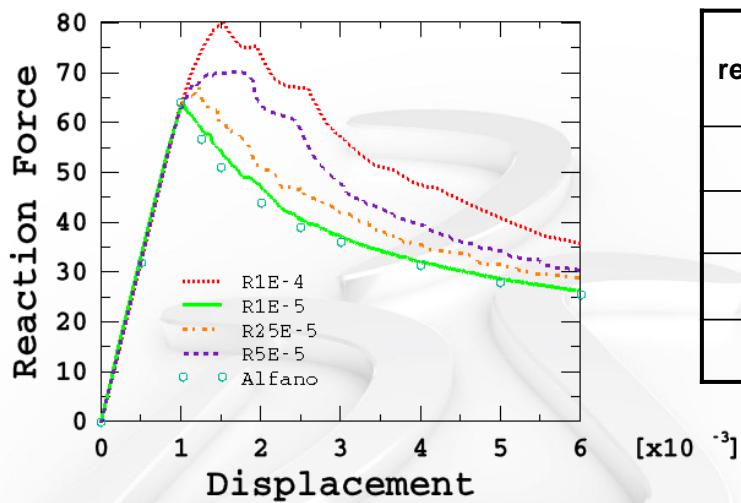


Analysis of Composite Materials with Abaqus

Modeling Techniques

- Effect of viscous regularization

© DASSAULT SYSTEMES



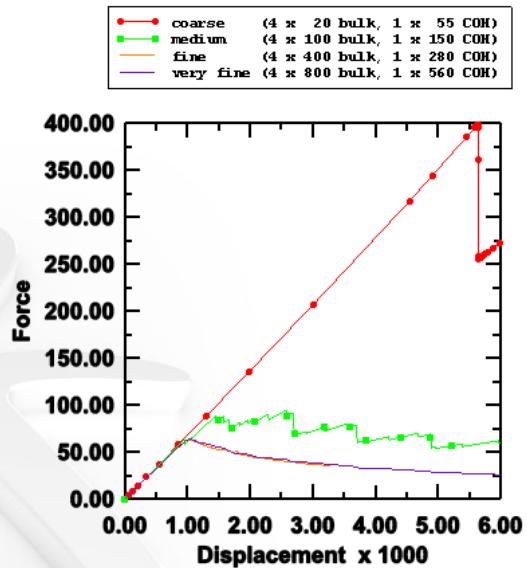
Viscous regularization factor	Total number of increments
1.e-5	636
2.5e-5	163
5.0e-5	129
1.0e-4	90

Analysis of Composite Materials with Abaqus

Modeling Techniques

- Effect of mesh refinement

- Typically, you will need to use a much finer mesh (for both the stress/displacement and cohesive elements) than may be necessary for a problem without cohesive elements.



© DASSAULT SYSTEMES

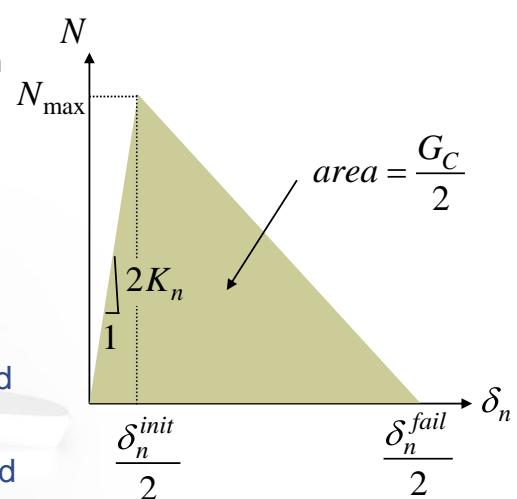


Analysis of Composite Materials with Abaqus

Modeling Techniques

- Cohesive elements on a symmetry plane

- The traction-separation law is based on the separation between the top and bottom faces of the cohesive element.
- On a symmetry plane, however, the separation that is computed is $\frac{1}{2}$ the actual value.
- To account for this, specify:
 - $2\times$ the cohesive stiffness that would be used in a full model.
 - $\frac{1}{2}$ the fracture toughness that would be used in a full model.
 - Linear equations between the nodes on the top and bottom faces in the lateral directions.



$$2K_n = \frac{2E_n}{h_{eff}} = \frac{E_n}{h_{eff}/2}$$

© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Techniques

- Symmetry example

Edit keywords, Model: dcb-alfano-2d-controls-symm

```

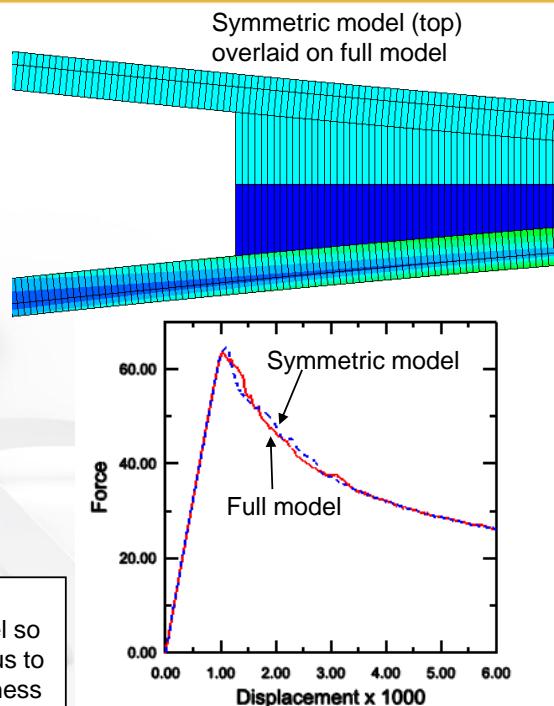
*End Instance
**
*Nset, nset=TOP, instance=PART-1-1
*Nset, nset=coh-top, instance=PART-1-1
*Nset, nset=coh-bot, instance=PART-1-1
** Constraint: Constraint-1
*Equation
2
coh-top, 1, 1.
coh-bot, 1, -1.
*End Assembly
**
** MATERIALS
**
*Material, name=COHESIVE
*Damage Initiation, criterion=QUADS
5.7e+07, 5.7e+07, 5.7e+07
*Damage Evolution, type=ENERGY, mixed mode behavior=BK, power=2.284
140.0, 140.0, 140.0
*Elastic, type=TRACTION
11.4e+14, 11.4e+14, 11.4e+14

```

Block: Add After Remove Discard OK

Constraint on lateral displacements

Constitutive thickness is same as for the full model so double the elastic modulus to double the cohesive stiffness



Analysis of Composite Materials with Abaqus

Notes

Notes

Modeling Issues for Continuum Shell Elements

Appendix 3

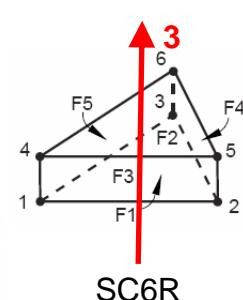
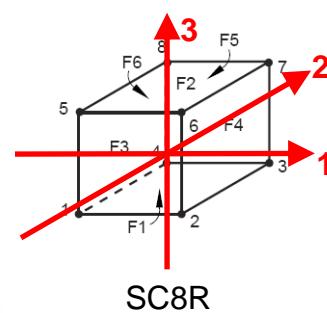
© DASSAULT SYSTEMES



A3.2

Modeling Issues for Continuum Shell Elements

- Defining the thickness direction for continuum shell elements
 - Define the thickness direction based on the element isoparametric directions.
 - Set STACK DIRECTION={1| 2|3} on the *SHELL SECTION or *SHELL GENERAL SECTION option to define the element thickness direction along an isoparametric direction.
 - SC8R has three possible stacking directions; SC6R has only one stack direction (the 3-direction).
 - The default stacking direction is 3 (i.e., based on nodal connectivity).



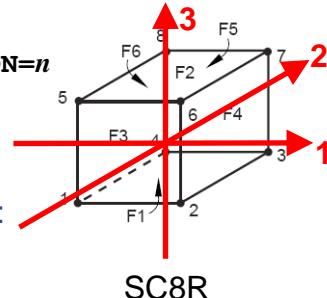
© DASSAULT SYSTEMES

Modeling Issues for Continuum Shell Elements

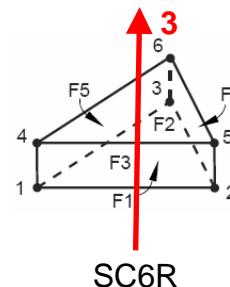
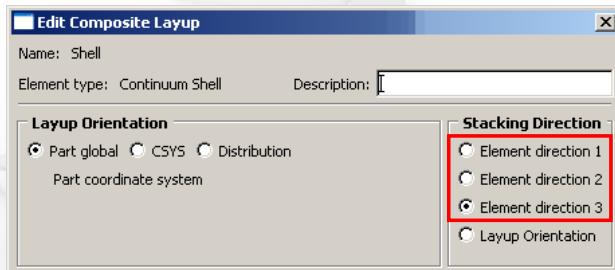
- Keywords interface:

***SHELL SECTION, STACK DIRECTION=n**
***SHELL GENERAL SECTION, STACK DIRECTION=n**

where $n = 1, 2, \text{ or } 3$



- Abaqus/CAE interface (composite layup approach):



SIMULIA

- Note:** The STACK DIRECTION option is not supported in the shell section GUI; use the Keywords Editor to include it in your model.

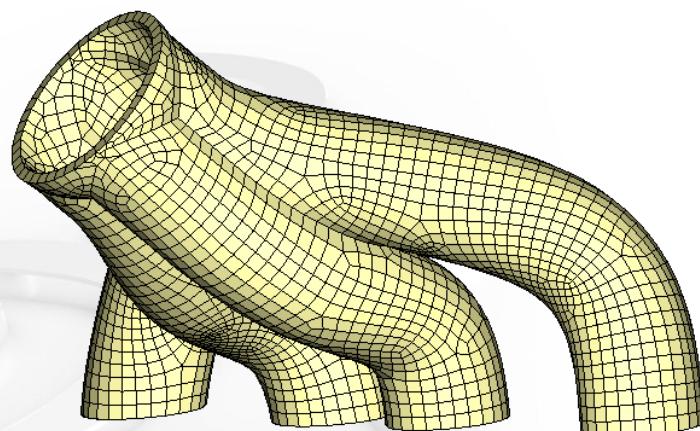
Analysis of Composite Materials with Abaqus

© DASSAULT SYSTEMES

Modeling Issues for Continuum Shell Elements

- Example: Engine manifold

- The element isoparametric direction method is particularly useful in this model given the widespread variations in surface curvature.



SIMULIA

Analysis of Composite Materials with Abaqus

© DASSAULT SYSTEMES

Modeling Issues for Continuum Shell Elements

- ② Define the thickness direction based on material orientations.**
- Set **STACK DIRECTION=ORIENTATION** to define the element thickness direction based on the section orientation definition given by the **ORIENTATION** parameter on the ***SHELL SECTION** or ***SHELL GENERAL SECTION** option.
 - The second data line on the ***ORIENTATION** option defines an **approximate normal**.
 - The element thickness direction is defined by the element faces which project best onto this approximate normal.

© DASSAULT SYSTEMES



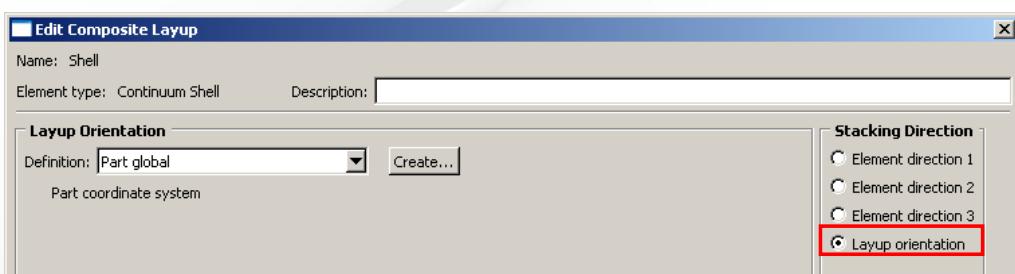
Analysis of Composite Materials with Abaqus

Modeling Issues for Continuum Shell Elements

- Keywords interface:


```
*SHELL SECTION, STACK DIRECTION=ORIENTATION,
          ORIENTATION=name
```

```
*SHELL GENERAL SECTION, STACK DIRECTION=ORIENTATION,
          ORIENTATION=name
```
- Abaqus/CAE interface (composite layup approach):



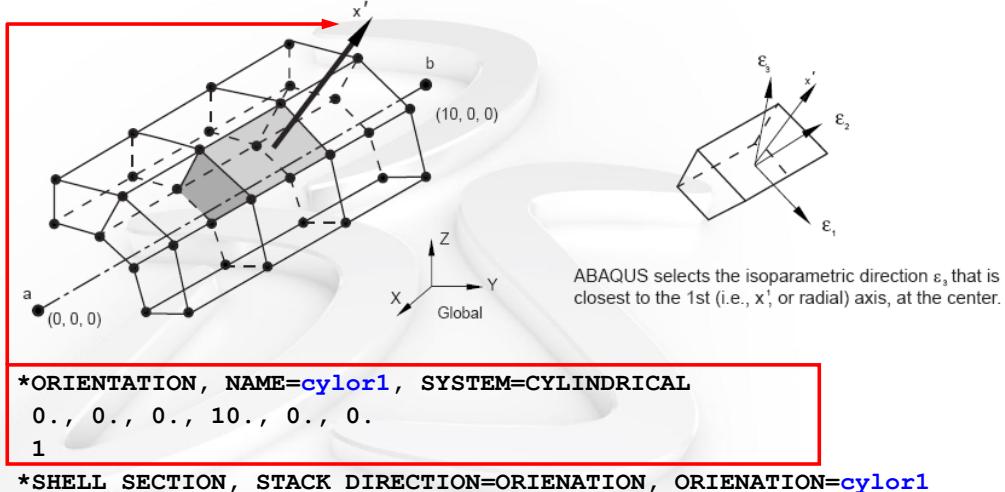
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Issues for Continuum Shell Elements

- The orientation method is most useful when:
 - The thickness directions needs to be independent of nodal connectivity.
 - The modeled structure has a regular shape.
- Example of using a cylindrical coordinate system to define thickness direction



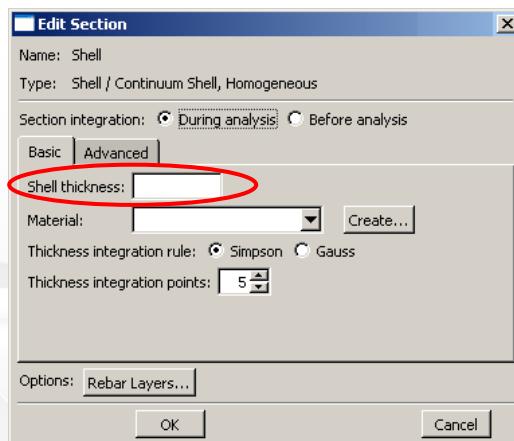
© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

Modeling Issues for Continuum Shell Elements

- Shell thickness
 - The thickness of an element is **always** taken from the nodal coordinates.
 - A thickness value **must** be given on the data line of the *SHELL SECTION or *SHELL GENERAL SECTION option.
 - However, it is only used to calculate some initial properties, such as hourglass stiffness, which are then appropriately scaled to the element thickness.
 - The thickness may be continuously varying (tapered shells) without having to specify the *NODAL THICKNESS option.



© DASSAULT SYSTEMES



Analysis of Composite Materials with Abaqus

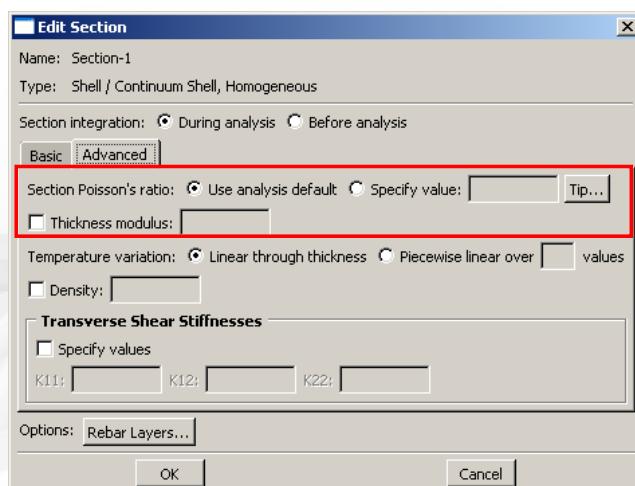
Modeling Issues for Continuum Shell Elements

- Change in thickness and thickness modulus
 - The change in thickness is calculated from the nodal displacements, an effective thickness modulus, and an effective section Poisson's ratio.
 - By default, the thickness modulus and section Poisson's ratio are based on the initial material properties.
 - Alternatively, you may define an effective thickness modulus and effective section Poisson's ratio as part of the shell section definition.
 - In cases where the thickness modulus and section Poisson's ratio cannot be computed, for example when a UMAT or UGENS user subroutine is used, you must specify the effective thickness modulus and effective section Poisson's ratio.
 - The Abaqus usage is given in the next slide.
 - The thickness direction stiffness remains constant and is not recalculated during the analysis.

Modeling Issues for Continuum Shell Elements

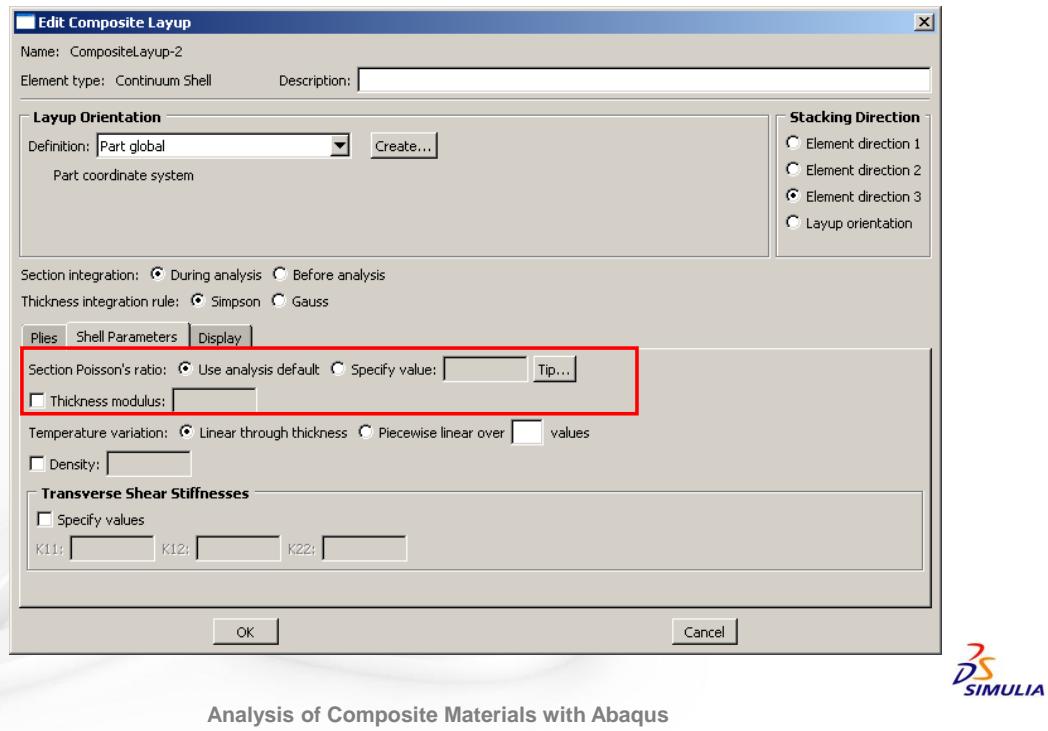
- Keywords interface:


```
*SHELL SECTION, POISSON=v, THICKNESS MODULUS=e
*SHELL GENERAL SECTION, POISSON=v, THICKNESS MODULUS=e
```
- Abaqus/CAE interface (shell section approach):



Modeling Issues for Continuum Shell Elements

- Abaqus/CAE interface (composite layup approach):



Notes

Notes

Optimization of a composite tube

Demo 1

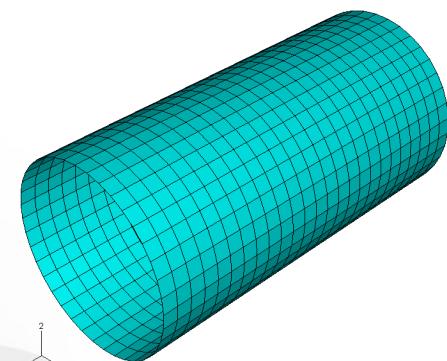
© DASSAULT SYSTEMES



D1.2

Description of the problem

- You have been tasked with the design of a cylindrical structure, and the design challenge is that you must design the structure so that no (or very small) circumferential strain is produced if there is a temperature change of 200 °C
- Weight is an issue; therefore, simply making the structure very stiff in the circumferential direction is not feasible
- The dimensions of the structure must be:
 - Radius: 50 mm
 - Length: 200 mm



© DASSAULT SYSTEMES

Composite design

- Using a composite material for this application seems logical because the thermal expansion characteristics can be controlled based on the constituents we choose
- For example, if we use a unidirectional Graphite-Epoxy sheet pre-impregnated with resin, the material properties would be similar to the following*:
 - $E_1 = 155 \text{ GPa}$, $E_2 = 12.10 \text{ GPa}$, $E_3 = 12.10 \text{ GPa}$, $\nu_{12} = 0.248$, $\nu_{13} = 0.248$, $\nu_{23} = 0.458$, $G_{12} = 4.40 \text{ GPa}$, $G_{13} = 4.40 \text{ GPa}$, $G_{23} = 3.20 \text{ GPa}$
 - $\alpha_1 = -0.01800 \text{ E-6}$, $\alpha_2 = 24.3 \text{ E-6}$, $\alpha_3 = 24.3 \text{ E-6}$
- The important property here is that the coefficient of thermal expansion in the fiber direction is negative, whereas the coefficients in the other directions are positive

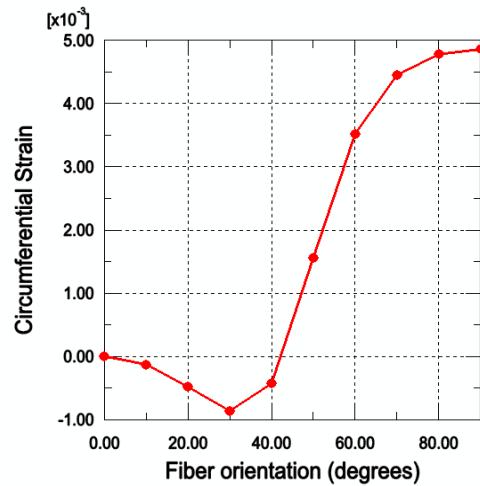
* From: Hyer, M. W., 1998. Stress Analysis of Fiber-Reinforced Composite Materials. McGraw-Hill: Boston.

Composite design

- Some design constraints are introduced to limit the design space:
 - We will consider only a fixed number layers in this laminate (four)
 - We will only use a symmetric, balanced laminate of the form: [angle]_s
- Therefore, the only parameter that is needed is the fiber direction angle
- The parametric study scripting interface will be used to run numerous Abaqus jobs with different fiber orientations to get an idea of what the angle should be
 - The angle will be varied between 0 and 90 degrees (measured from the axis of the cylinder), with ten data points in this range
 - The circumferential strain will be examined to determine the best design

Results

- If we examine the circumferential strain versus fiber direction...



- We see that the circumferential strain is minimized at a fiber angle of approximately 42 degrees!!!

Notes

Notes



Workshop Preliminaries

Setting up the workshop directories and files

If you are taking a public seminar, the steps in the following section have already been done for you: skip to **Basic Operating System Commands, (p. WP.2)**. If everyone in your group is familiar with the operating system, skip directly to the workshops.

The workshop files are included on the Abaqus release CD. If you have problems finding the files or setting up the directories, ask your systems manager for help.

Note for systems managers: If you are setting up these directories and files for someone else, please make sure that there are appropriate privileges on the directories and files so that the user can write to the files and create new files in the directories.

Workshop file setup

(Note: UNIX is case-sensitive. Therefore, lowercase and uppercase letters must be typed as they are shown or listed.)

1. Find out where the Abaqus release is installed by typing

UNIX and Windows NT: **abqxxx whereami**

where **abqxxx** is the name of the Abaqus execution procedure on your system. It can be defined to have a different name. For example, the command for the 6.9-1 release might be aliased to **abq691**.

This command will give the full path to the directory where Abaqus is installed, referred to here as *abaqus_dir*.

2. Extract all the workshop files from the course tar file by typing

UNIX: **abqxxx perl abaqus_dir/samples/course_setup.pl**

Windows NT: **abqxxx perl abaqus_dir\samples\course_setup.pl**

Note that if you have Perl and the compilers already installed on your machine, you may simply type:

UNIX: **abaqus_dir/samples/course_setup.pl**

Windows NT: **abaqus_dir\samples\course_setup.pl**

3. The script will install the files into the current working directory. You will be asked to verify this and to choose which files you wish to install. Choose “**y**” for the appropriate lecture series when prompted. Once you have selected the lecture series, type “**q**” to skip the remaining lectures and to proceed with the installation of the chosen workshops.

Basic operating system commands

(You can skip this section and go directly to the workshops if everyone in your group is familiar with the operating system.)

Note: The following commands are limited to those necessary for doing the workshop exercises.

Working with directories

1. Start in the current working directory. List the directory contents by typing

UNIX: **ls**

Windows NT: **dir**

Both subdirectories and files will be listed. On some systems the file type (directory, executable, etc.) will be indicated by a symbol.

2. Change directories to a workshop subdirectory by typing

Both UNIX and Windows NT: **cd dir_name**

3. To list with a long format showing sizes, dates, and file, type

UNIX: **ls -l**

Windows NT: **dir**

4. Return to your home directory:

UNIX: **cd**

Windows NT: **cd home-dir**

List the directory contents to verify that you are back in your home directory.

5. Change to the workshop subdirectory again.

6. The * is a wildcard character and can be used to do a partial listing. For example, list only Abaqus input files by typing

UNIX: **ls *.inp**

Windows NT: **dir *.inp**

Working with files

Use one of these files, *filename.inp*, to perform the following tasks:

1. Copy *filename.inp* to a file with the name **newcopy.inp** by typing

UNIX: **cp filename.inp newcopy.inp**

Windows NT: **copy filename.inp newcopy.inp**

2. Rename (or move) this new file to **newname.inp** by typing

UNIX: **mv newcopy.inp newname.inp**

Windows NT: **rename newcopy.inp newname.inp**

(Be careful when using **cp** and **mv** since UNIX will overwrite existing files without warning.)

3. Delete this file by typing

UNIX: **rm newname.inp**

Windows NT: **erase newname.inp**

4. View the contents of the files *filename.inp* by typing

UNIX: **more filename.inp**

Windows NT: **type filename.inp | more**

This step will scroll through the file one page at a time.

Now you are ready to start the workshops.

Notes

Notes



Workshop 1

Laminated Composite Panel

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the “Keywords” version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define the material properties of a fiber-matrix layer.
- Define a composite layup using the composite layup editor.
- View a ply stack plot of a composite layup.
- Perform prebuckling, eigenvalue buckling, and postbuckling analyses.
- Use the Visualization module to view material orientations and create contour plots on different plies, and to view an envelope plot.

Introduction

In recent years, fiber-reinforced composite laminated shell structures have been widely used in the aerospace, marine, automobile, and other engineering industries for lightweight applications. The buckling and post-buckling analysis of composite panels is of primary importance to an optimized design for a specific composite light-weight-component as well as the entire structure. The load at which buckling occurs and whether the subsequent deformation is catastrophic or well-behaved are critical design issues and depend heavily on the lay-up of the composite panel material.

In this workshop you will define the material properties of a fiber-matrix layer and composite layup with different ply orientations and study the buckling of a composite structure. Figure W1–1 shows the structure analyzed in this workshop. It is a cylindrical composite panel with a circular hole. The panel is fully clamped on the bottom edge, free to move axially along the top edge, and simply supported along its vertical edges. Three analyses will be performed. The first is a linear (prebuckling) analysis in which the panel is subjected to a uniform end shortening of 0.0316 in. The second analysis consists of an eigenvalue extraction of the first five buckling modes. The total axial force (distributed along the midsection) is used in this analysis, instead of the shortening, since the buckling

load can be easily calculated using the axial force (buckling load = eigenvalue \times axial load). Finally, a nonlinear load-deflection analysis is performed using the modified Riks algorithm. The postbuckling behavior is induced with an initial imperfection based on the extracted buckling modes.

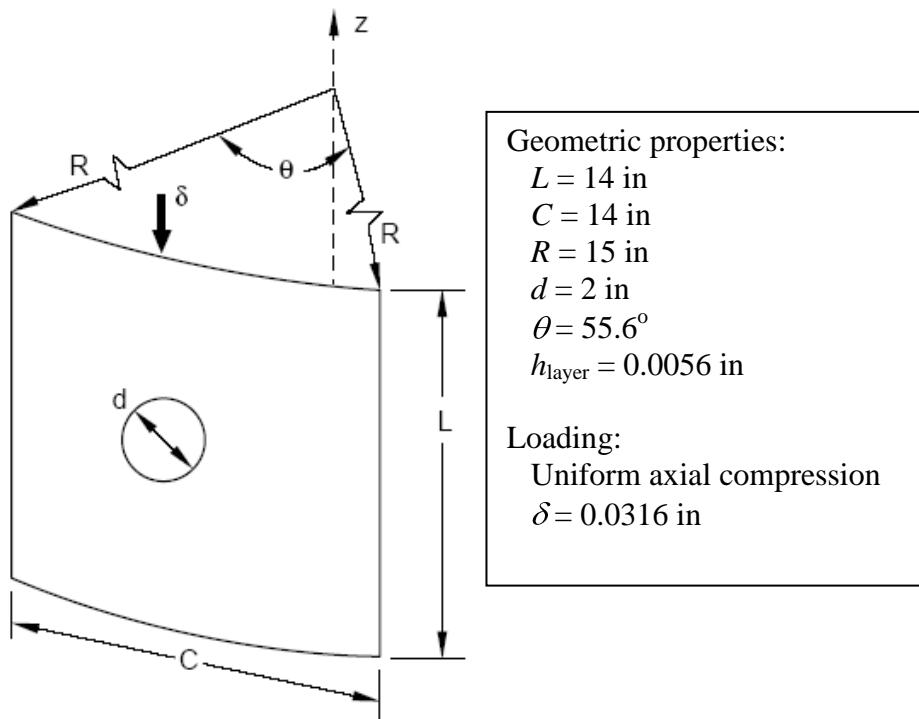


Figure W1–1. Geometry of the cylindrical panel with hole.

Figure W1–2 shows the details of the model, including the mesh and boundary conditions. Note that the symmetry conditions indicated in the figure represent real physical boundary conditions and are not intended to imply a mirrored structure. Thus, the comments regarding symmetry conditions in Lecture 3 do not apply in this particular case.

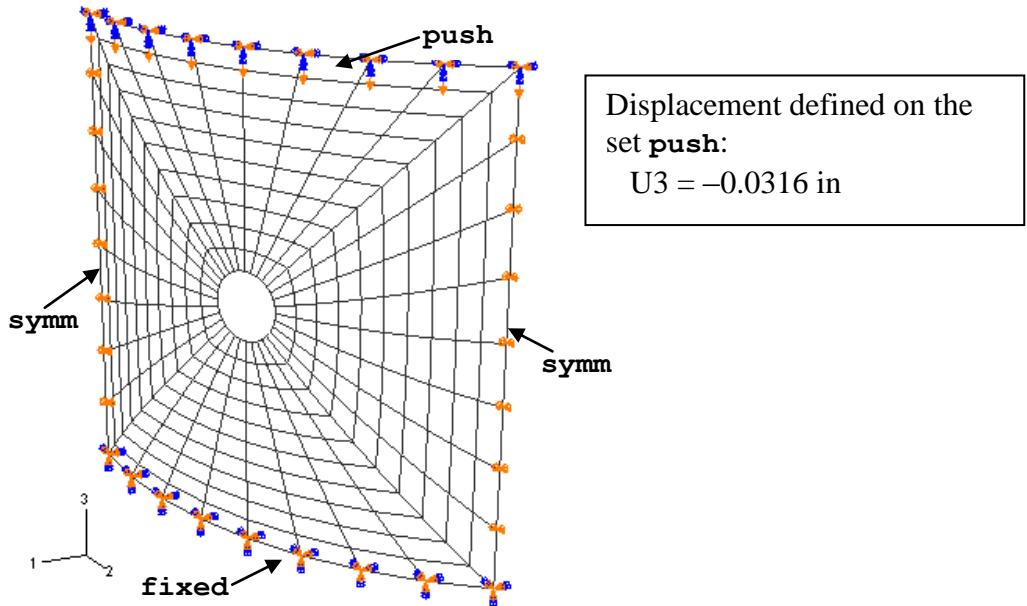


Figure W1–2. Model geometry and boundary conditions

Preliminaries

1. Enter the working directory for this workshop:

```
./composites/interactive/buckle
```

2. Run the script `ws_composites_panel.py` using the following command:

```
abaqus cae startup=ws_composites_panel.py
```

The above command creates an Abaqus/CAE database named `laminated-panel.cae` in the current directory. The geometry, mesh, and boundary condition definitions for the panel are included in the model named `prebuckle`. The shell element type S4R is used in the model. This model will first be used to perform the prebuckling analysis and will later be edited to perform the eigenvalue buckling and postbuckling analyses.

Part 1: Prebuckling analysis

You will use the composite layup editor in the Property module to define a conventional shell composite layup for the composite panel. The layup consists of 16 plies of unidirectional graphite fibers in an epoxy resin. The plies are arranged in the symmetric stacking sequence $[(45, -45, 90, 0)_S]_S$ degrees. Each ply is 0.0056 in thick. The nominal orthotropic elastic material properties of the lamina are

$$E_{11} = 19.6 \times 10^6 \text{ lb/in}^2,$$

$$E_{22} = 1.89 \times 10^6 \text{ lb/in}^2,$$

$$G_{12} = G_{13} = 0.93 \times 10^6 \text{ lb/in}^2,$$

$$G_{23} = 0.63 \times 10^6 \text{ lb/in}^2, \text{ and}$$

$$\nu_{12} = 0.38$$

where the 1-direction is along the fibers, the 2-direction is transverse to the fibers in the surface of the lamina, and the 3-direction is normal to the lamina.

Completing the prebuckling model

To complete the model, do the following:

1. Define the orthotropic elastic behavior of the lamina with the material properties given above.
 - a. In the Model Tree, double-click the **Materials** container. Abaqus/CAE switches to the Property module, and the material editor appears.
 - b. In the **Edit Material** dialog box, name the material **lamina**.
 - c. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic**.
 - d. Under the **Elastic** field, select the type **Lamina**, as shown in Figure W1–3.

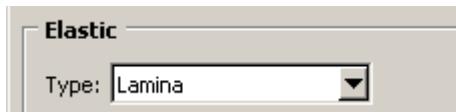


Figure W1–3. Type field in the material editor

- e. Enter the data for the orthotropic elastic material properties, as shown in Figure W1–4:

Data					
	E1	E2	Nu12	G12	G13
1	19.6e6	1.89e6	0.38	0.93e6	0.93e6

Figure W1–4. Data field in the material editor.

- f. Click **OK** to exit the material editor.
2. Define a datum coordinate system (CSYS) that will be used to define the composite layup orientation.
 - a. Click the **Create Datum CSYS: 3 Points** tool  in the toolbox.
 - b. In the **Create Datum CSYS** dialog box, name the datum CSYS **Layup_Orientation**, select **Cylindrical** as the coordinate system type, and click **Continue**.
 - c. Enter **0.0,0.0,0.0** as the origin, **0.0,-1.0,0.0** as the coordinates of a point on the *R*-axis, and **1.0,1.0,0.0** as the coordinates of a point in the *R-Theta* plane to create the datum coordinate system.
3. Define a conventional shell composite layup consisting of the 16 plies noted above. Each ply is assigned the material **lamina** and consists of the entire shell.

- a. In the Model Tree, expand the branch of the part named **panel** underneath the **Parts** container.

- b. Double-click **Composite Layups**.

- c. In the **Create Composite Layup** dialog box, name the composite layup **shell_layup**, choose **Conventional Shell** as the element type, and click **Continue**.

The composite layup editor appears.

- d. Under the **Layup Orientation** field of the composite layup editor, select **Coordinate system** as the definition to define a layup orientation coordinate system.

Note that more options appear underneath **Coordinate system**.

- e. Click **Select** underneath **Coordinate system**. In the prompt area, click **Datum CSYS List** to select the datum CSYS **Layup_Orientation**.

Return to the composite layup editor.

- f. In the **Layup Orientation** field, select **Axis-1** as the direction of the approximate shell normal and accept no additional rotation.

- g. Since the material behavior is linear elastic, choose **Before analysis** as the section integration technique to reduce the computational cost of the job. Accept **No idealization** as the section response.

You will next define the ply data using the ply table of the composite layup editor. Note that three rows are available in the ply table by default.

- h. In the ply table, double-click the column heading **Region**. Select the entire part as the region for all plies. Click mouse button 2 in the viewport or click **Done** in the prompt area to confirm the selection.

- i. In the ply table, double-click the column heading **Material**. In the **Select Material** dialog box that appears, select the material **lamina** and click **OK** to assign the material to all plies.

- j. In the ply table, double-click the column heading **Thickness**. In the **Thickness** dialog box that appears, select **Specify Value** and enter a value of **0.0056** and click **OK** to assign the thickness to all plies.

- k. In the **Rotation Angle** column of the ply table, enter a value of **45** for **Ply-1**, **-45** for **Ply-2**, and **90** for **Ply-3**.

- l. Select any cell of **Ply-3** and then click the **Copy Plies After** icon  above the ply table. A new ply named **Ply-3-Copy1** (a copy of **Ply-3**) appears. Rename the ply **Ply-4** and change the rotation angle of this ply to **0**.

- m. Select all four plies (**Ply-1**, **Ply-2**, **Ply-3**, and **Ply-4**) and the click the **Pattern Plies** icon  above the ply table. In the **Pattern Plies** dialog box that appears, choose **Symmetry** as the pattern and **Last ply in layup**

as the ply about which the pattern will be symmetric. Click **OK** to create four new plies.

Note: To select multiple plies, drag the mouse over the cells containing the data that you want to copy; or use **[Shift]+Click** for multiple selections.

Four new plies appear.

- n. Rename the plies for easy tracking during postprocessing.
The bottom 8 plies in the entire layup are now complete.
- o. Since the composite layup is symmetric, toggle on **Make calculated sections symmetric** to activate simplified modeling of symmetric composites.
The composite layup editor is shown in Figure W1–5.
- p. Click **OK** to exit the composite layup editor.

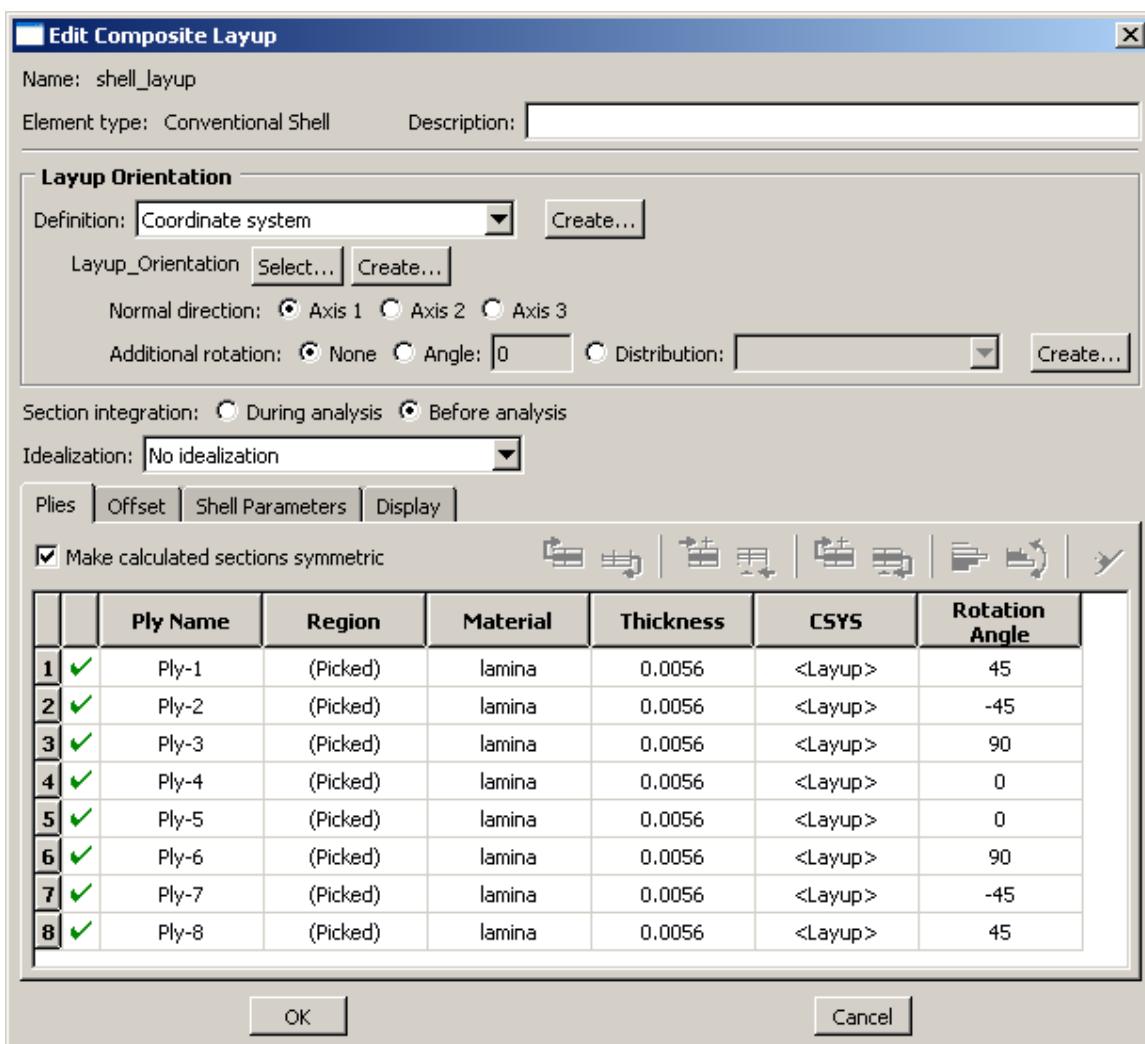


Figure W1–5. The conventional shell composite layup definition

4. You will now create a ply stack plot from a selected region to view the plies defined in the composite layup.

- Click the **Query information** tool  in the tool bar.
- In the **Query** dialog box, select **Ply stack plot** from the list of **Property Module Queries**.

Note that a new viewport is created and tiled vertically with respect to the original viewport.

- In the viewport displaying the part, select the region highlighted in Figure W1–6 (left).

The ply stack plot of this region appears in the new viewport (see Figure W1–6 (right)).

Note: The ply stack plot displays all plies in the composite layup, including those generated using the symmetry option. These plies contain the prefix **Sym_** in their names to indicate that they are repeated plies. The staircase appearance in the ply stack plot has no physical meaning; it is simply a graphical representation that allows you to see the number of plies in the layup and, for example, the relative thickness of a ply and the orientation of its fibers.

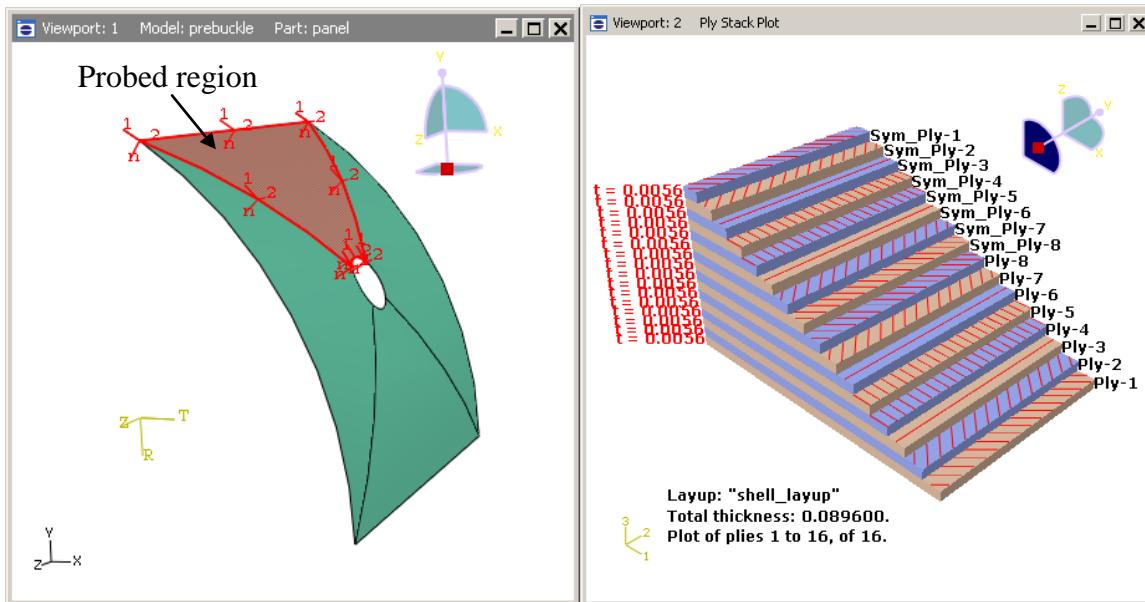


Figure W1–6. Ply stack plot of a probed region

- Close the viewport displaying the ply stack plot and maximize the viewport displaying the part.
- In this workshop, assume the ply at the inner side of the shell is the bottom ply in the layup. Since the first ply in the ply table also represents the bottom ply in the layup, you will define the shell normal so that the outward normal of the shell is positive.

- a. Note the small black triangles at the base of the toolbox icons. These triangles indicate the presence of hidden icons that can be revealed. Click the **Assign Beam Orientation** tool  but do not release the mouse button. When additional icons  appear, release the mouse button.
 - b. Select the **Assign Shell/Membrane Normal** tool .
 - c. Select the entire part as the region whose normals are to be flipped. Click mouse button 2 in the viewport or click **Done** in the prompt area to flip the outward region normals so they are positive.
 - d. Click mouse button 2 again in the viewport or click **Done** in the prompt area to exit the normal assignment interface.
7. Edit the existing field output request for the composite layup. Note that even though you chose the “before analysis” integration scheme, Abaqus uses section points for output purposes. You can view ply orientations and contour a specified ply in the Visualization module.
- a. Rename the default field output request to **shell_layup_output**.
 - b. In the field output request editor, select the **Composite layup** domain and the **panel-1.shell_layup** layup.
 - c. At the bottom of the field output request editor, choose the output request at the (default) middle section point for each ply.
Note that you can also request output at all section points by choosing **All section points in all plies**.
 - d. Click **OK** to exit the field output request editor.
8. Create a job for the prebuckling analysis named **preBuckle** with the following description: **Composite panel -- prebuckling analysis**.
- Tip:** to create a job, double-click **Jobs** in the Model Tree.
9. Save your model database file, and submit the job for analysis (in the Model Tree, click mouse button 3 on the job name and select **Submit** from the menu that appears). From the same menu, you can select **Monitor** to monitor the job’s progress.

Postprocessing the prebuckling analysis

When the analysis is complete, use the following procedure to view the contour plots and material orientations in individual plies in the Visualization module:

1. In the Model Tree, click mouse button 3 on the job **preBuckle** and select **Results** from the menu that appears to open the file **preBuckle.odb** in the Visualization module.
2. From the main menu bar, select **Result→Section Points**. The **Section Points** dialog box appears. Choose the **Plies** selection method. A list of all plies in the composite layup appears in the **Plies** field, as shown in Figure W1–7. You can select any individual ply to display output.

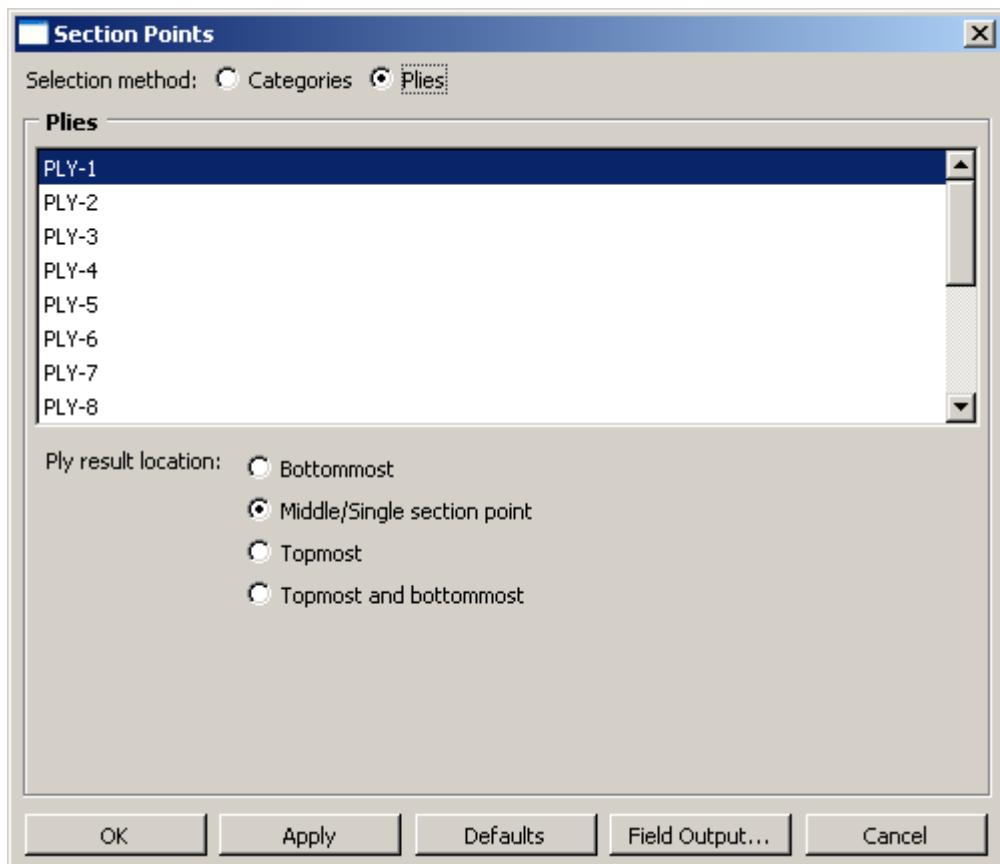


Figure W1–7. **Section Points** dialog box.

3. Accept the default selections (**PLY-1** and the **Middle** result location) and click **Apply**.
4. Click the **Plot Contours on Deformed Shape** tool  in the toolbox. A contour plot of Mises stress at the middle of **PLY-1** appears.
5. Contour the stress component along the fiber direction (variable S11) of **PLY-1**. Use the **Field Output** toolbar to change the displayed output variable:

- a. From the list of variable types on the left side of the **Field Output** toolbar, select **Primary** if it is not already selected.
- b. From the list of available output variables in the center of the toolbar, select output variable **S** if it is not already selected.
- c. From the list of available components and invariants on the right side of the **Field Output** toolbar, select **S11**.

A contour plot of S11 at the middle of **PLY-1** appears.

6. Click the **Plot Material Orientations on Deformed Shape** tool  in the toolbox.

A plot of the material orientations at the middle of **PLY-1** appears.

Note that you can also view the output and material orientations at the bottom, top, or top and bottom locations simultaneously by selecting the corresponding available ply result locations in the **Section Points** dialog box.

7. Create a ply stack plot from a probed composite layup section. The combination of the ply stack plot and the ply-based material orientation plot provides a complete view of the ply orientation.

- a. Click the **Query information** tool  in the tool bar.
- b. In the **Query** dialog box, select **Ply stack plot** from the list of **Visualization Module Queries**.

Note that a new viewport is created and tiled vertically with respect to the original viewport.

- c. The ply stack plot in the Visualization module is section-based. In the viewport displaying the shell, click anywhere on the mesh to select the entire section, as shown in Figure W1–8 (left).
- d. The ply stack from the probed section appears in the new viewport (see Figure W1–8 (right)).

From the ply stack plot you can see that the ply **PLY-1** is the bottom ply in the composite layup, and the fiber in this ply is oriented along 45° with respect to 1-axis of the layup orientation.

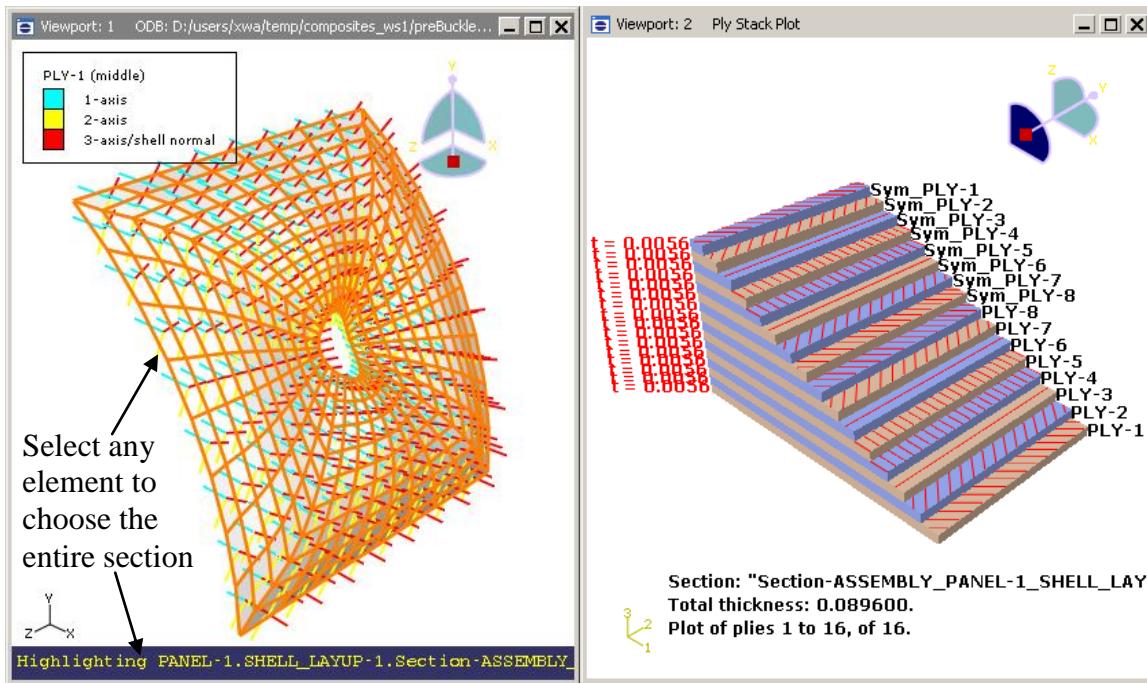


Figure W1-8. Ply stack plot of a probed section

8. Make current the viewport displaying the material orientation plot, if necessary. In the **Section Points** dialog box, select the ply **PLY-2** and click **Apply**.
The material orientation plot now displays the material orientations at the middle of **PLY-2**. Again, from the ply stack plot, you can see that the ply **PLY-2** is the second ply from the bottom in the composite layup, and the fiber in this ply is oriented along -45° with respect to 1-axis of the layup orientation.
9. Click  in the toolbox.
A contour plot of S11 at the middle of **PLY-2** appears.
Note that the S11 contour plot changes with the layer orientation. You can also view the output on other plies.
10. Create an *envelope* plot to identify the maximum values of S11 across all the plies of the layup.
 - a. In the **Section Points** dialog box, choose **Categories** as the selection method and then **Envelope** as the active location.
 - b. Select **Max value** as the criterion and **Integration point** as the position.
 - c. Click **OK** to plot the stress envelope of the shell.
11. Show the critical plies on the envelope plot.
 - a. Delete the viewport displaying ply stack plot.
 - b. Create a new viewport and tile it vertically with the previous one (**Viewport**→**Tile Vertically**).

- c. Change the contour variable to S11 to create the same envelope plot as described in the previous step.



- d. Click the **Contour Options** tool in the toolbox.
- e. In the **Contour Plot Options** dialog box that appears, select the **Other** tabbed page and toggle on **Show labels of plies that match criteria**.
- f. Click **Apply**.

The name of the critical ply at each element appears on the envelope plot. Alternatively, a quilt plot of ply labels can be created.

- g. Toggle off **Show labels of plies that match criteria**; instead toggle on **Color by plies that match criteria, instead of result value**.
- h. Click **OK** to display a quilt plot of critical plies.

Note that, using a combination of these plots created in steps 10 and 11, you can determine both the value of S11 in the critical ply and the location of the critical ply in the layup.

Part 2: Eigenvalue buckling analysis

You will perform an eigenvalue buckling analysis to determine the buckling eigenmodes of the composite structure. These buckling eigenmodes will be used to introduce imperfections into the geometry of the panel to ensure a physically correct deformed shape in the subsequent postbuckling analysis.

To ensure a physically meaningful simulation, the buckling analysis must be performed on a model that is similar to the postbuckling model. Unlike the previous analysis, the total axial force (distributed along the midsection of the top edge) will be used since it will be easier to determine the buckling load (recall the buckling load = eigenvalue \times axial load).

Before you begin the modifications required for the eigenvalue buckling analysis, make a copy of the first analysis model.

Completing the eigenvalue buckling model

To complete the model, do the following:

1. Copy the model named **prebuckle** to a model named **buckle**. Make current the model **buckle** and switch to any assembly-level module.
- Tip:** To copy the model, click mouse button 3 on the model name in the Model Tree and select **Copy Model** in the menu that appears.
2. In the viewport, double-click the 3D Compass. In the **Specify View** dialog box that appears, select **Viewpoint** as the specification method, enter **1, 2.5, 1** as the viewpoint and **0, 0, 1** as the up vector, and click **OK**. The model is now oriented so that the global Z-axis is vertical.

3. The eigenvalue buckling analysis procedure will be used instead of the general static one. Thus, replace the general static step with an eigenvalue buckling step.
 - a. In the Model Tree, expand the **Steps** container.
 - b. Click mouse button 3 on the step **Step-1** and select **Replace** from the menu that appears.
 - c. In the **Replace Step** dialog box, replace the general static step with a linear eigenvalue buckling step. Click **Continue**.
The **Edit Step** dialog box appears.
 - d. Enter 5 in the **Number of eigenvalues requested** field, 50 in the **Vectors used per iteration** field, and 20 in the **Maximum number of iterations** field.
 - e. Click **OK** to exit the step editor.
 - f. Open the field output requests editor and set the domain to **Whole model**.
4. Remove the displacement boundary condition defined along degree of freedom 3 for the set **push**.
 - a. In the Model Tree, expand the **BCs** container.
 - b. Double-click on the boundary condition **push**.
 - c. Toggle off **U3**.
 - d. Click **OK** to save the change and to exit the boundary condition editor.
5. Define the axial force at the center of the top edge.
 - a. In the Model Tree, double-click **Loads**. Accept the default category **Mechanical** and type **Concentrated force**, and click **Continue**.
 - b. In the prompt area, click **Sets** to select the set **point-load**.
Note that the set **point-load** only includes the center of the top edge.
 - c. In the **Edit Load** dialog box, enter a magnitude of **-1000** for **CF3**.
 - d. Click **OK** to exit the load editor.
6. The axial force is distributed along the midsection of the top edge through constraint equations between the center and the remainder of the top edge. The equation constraint is along degree of freedom 3.
 - a. In the Model Tree, double-click **Constraints**.
The **Create Constraint** dialog box appears.
 - b. Select **Equation** and click **Continue**.
The **Edit Constraint** dialog box appears.
 - c. Enter a coefficient of **1.0**, the set name **edge-load**, and degree of freedom **3** in the first row. In the second row, enter a coefficient of **-1.0**, the set name **point-load**, and degree of freedom **3**, as shown in Figure W1-9.
Note that the set **edge-load** includes the top edge except its center.

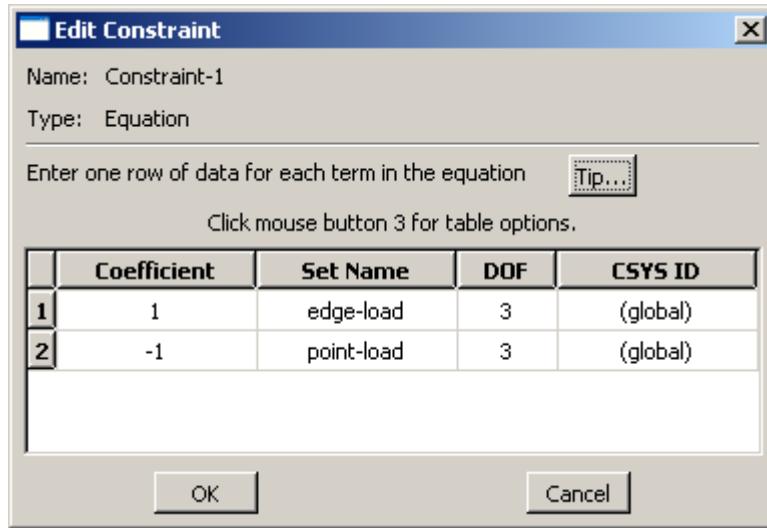


Figure W1–9. **Edit Constraint** dialog box.

- d. Click **OK** to exit the constraint editor.
7. You need to request that additional data are written to the results (.fil) file using the **Keywords Editor**. You will write the normalized nodal displacements corresponding to each linear buckling mode; these will be used to introduce imperfections in the postbuckling analysis.
- a. In the Model Tree, click mouse button 3 on the model named **buckle** and select **Edit Keywords** from the menu that appears.
The **Edit Keywords** dialog box appears, containing the input file that has been generated for your model.
 - b. Only text blocks with a white background can be edited. Use the scroll bar on the right side of the dialog box to find the text block where the ***Restart** option is located (toward the bottom of the file). Select the ***Restart** block, and click **Add After** to add an empty text block.
 - c. In the new text block, enter the following data to specify that displacements should be written to the results file:
- ```
*Node File
U,
```
- d. Click **OK** to save your changes and to exit the **Keywords Editor**.
8. Create a job named **Buckle** for the model **buckle** with the following description: **Composite panel -- buckling analysis**.
9. Save your model database file, submit the job **Buckle** for analysis, and monitor its progress.

### Postprocessing the buckling analysis

When the analysis is complete, use the following procedure to view the eigenmodes from the buckling analysis in the Visualization module:

1. In the Model Tree, click mouse button 3 on the job **Buckle** and select **Results** from the menu that appears to open the file **Buckle.odb** in the Visualization module.
2. Plot the deformed model shape.

The deformed shape for the first eigenmode will be displayed in the viewport. The corresponding eigenvalue will be reported in the state block. Adjust your view, if necessary, to see the deformed configuration more clearly.

3. View the deformed shapes of the other buckling modes using the frame selector  or the frame control buttons  in the context bar above the viewport.

Figure W1–10 shows the first five eigenmodes of the composite panel.

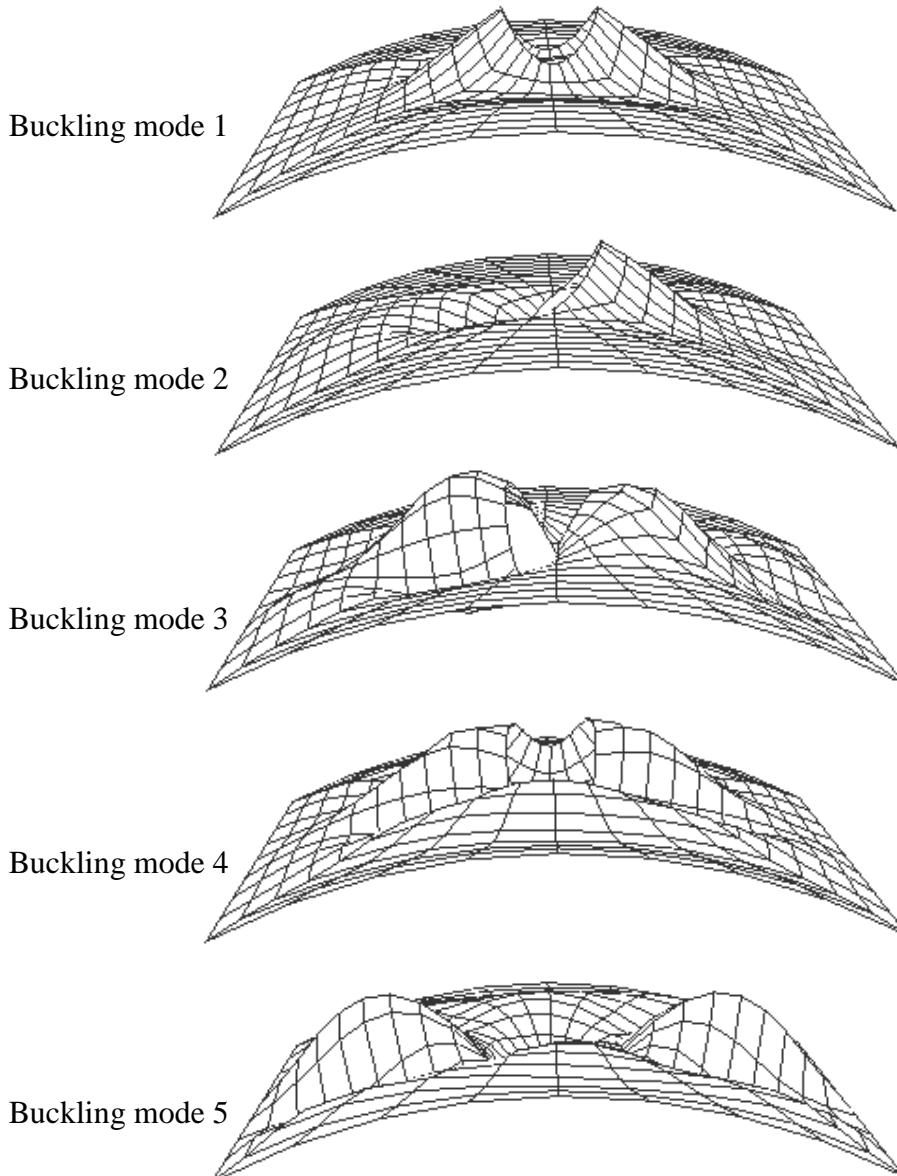


Figure W1–10. Buckling modes.

The primary result of interest in the buckling analysis is the predicted eigenvalues for each of these modes. The eigenvalue, when multiplied by the nominal load value used for the plate, provides a prediction of the buckling load value for each mode. From the main menu bar, select **Result→Step/Frame** to see the predicted eigenvalues. Click **Cancel** when you are done reviewing these results.

The fiber lay-up will have a significant effect on the value at which the plate buckles. To show this, results for a plate in which all fibers are aligned with the load direction (i.e., all plies in this model are oriented at 90° with respect to the 1-axis of the **<layup>** orientation) has been analyzed, and a comparison of these results are shown in the table below. A significant reduction in load carrying capability is predicted for the plate with all fibers oriented with the load direction.

Table W1–1. Comparison of buckling load prediction for two plate lay-ups.

|               | <b>Plate Buckling Loads (psi)</b> |                |
|---------------|-----------------------------------|----------------|
|               | $[(45, -45, 90, 0)_S]_S$          | $[90_{(8)}]_S$ |
| <b>Mode 1</b> | 25121                             | 18249          |
| <b>Mode 2</b> | 25910                             | 18859          |
| <b>Mode 3</b> | 27368                             | 21146          |
| <b>Mode 4</b> | 34053                             | 24536          |
| <b>Mode 5</b> | 40365                             | 25522          |

## Part 3: Postbuckling analysis

You will now modify the buckling analysis to perform a nonlinear load-deflection analysis to predict the postbuckling behavior. You will use the **Keywords Editor** to specify that the modes from the buckling analysis will be used to seed an initial imperfection in the postbuckling analysis model.

Before you begin the modifications required for the load-deflection analysis, make a copy of buckling analysis model.

### Completing the postbuckling model

To complete the model, do the following:

1. Copy the model named **buckle** to a model named **postbuckle**. Make current the model **postbuckle**.
2. The static, Riks analysis procedure will be used instead of the linear eigenvalue buckling one. Thus, replace the eigenvalue buckling step with a static, Riks step:

- a. In the Model Tree, expand the **Steps** container.
  - b. Click mouse button 3 on the step **Step-1** and select **Replace** from the menu that appears.
  - c. In the **Replace Step** dialog box, replace the linear perturbation buckle step with a static, Riks step. Click **Continue**.  
The **Edit Step** dialog box appears.
  - d. In the **Edit Step** dialog box, toggle on **Nlgeom**.
  - e. In the **Stopping criteria** field, toggle on **Maximum displacement** and enter **-0.08**, then enter **3** for **DOF**. Using the **Node Region** drop down menu, select **point-load**. The parameters are selected to ensure that the job runs long enough to show the collapse.
  - f. In the **Incrementation** tabbed page of the step editor, enter **15** for **Maximum number of increments**.
  - g. Click **OK** to accept other default settings and to exit the step editor.
3. Add a history output request to write the displacement history for the set **point-load**.
    - a. In the Model Tree, double-click **History Output Requests**.
    - b. In the **Create History** dialog box, name the history output request **displacement\_history** and click **Continue**.
    - c. In the **Edit History Output Request** dialog box, select the domain **Set** and the set **point-load**.
    - d. Expand the **Displacement/Velocity/Acceleration** list in the **Output Variables** field and toggle on the variable **U, Translations and rotations**.
    - e. Click **OK** to exit the history output editor.
  4. Increase the magnitude of the axial force to 10000.
    - a. In the Model Tree, expand the **Loads** container.
    - b. Double-click on the load **Load-1**.  
The **Edit Load** dialog box appears.
    - c. Enter **-10000** for **CF3**.
    - d. Click **OK** to save the change and to exit the load editor.
  5. Use the **Keywords Editor** to introduce geometric imperfections into the postbuckling model. The imperfections allow the buckling to occur in a smooth manner. The imperfections included in this model are based on the first four eigenmodes extracted from the eigenvalue buckling analysis; the magnitude of the imperfection is scaled so that the maximum imperfection is 10% of the shell thickness.
    - a. At the bottom of the **Keywords Editor**, click **Discard All Edits** to clear all previous keyword edits.
    - b. Select the text block containing the **\*Elastic** option, and click **Add After** to add an empty block of text.

- c. Add the following in this text block:

```
*Imperfection, file=Buckle, step=1
1, 0.001
2, 0.0005
3, 0.00025
4, 0.00025
```

6. Create a job for the postbukling analysis named **postBuckle** with the following description: **Composite panel -- postbuckling analysis**. Save your model database file, submit the job for analysis, and monitor its progress.

### Postprocessing the postbuckling analysis

Follow the procedure listed below to view the results of the postbuckling analysis.

1. Open **postBuckle.odb** in the Visualization module.
2. Plot the deformed model shape.

Figure W1–11 shows the deformed configuration at the end of simulation.

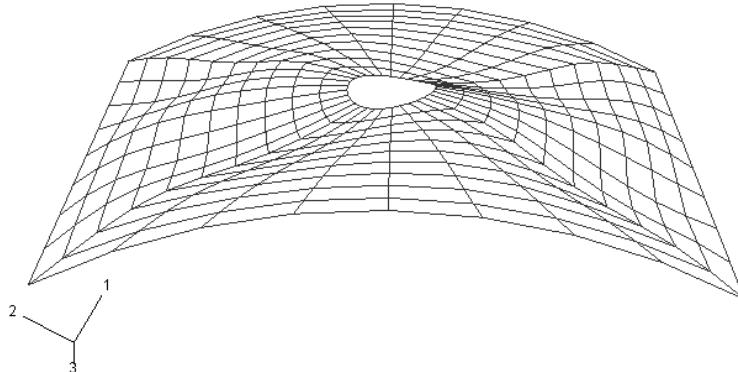


Figure W1–11. Deformed mesh.

3. Animate the deformation to see the behavior of the plate after buckling (click ). This animation will help you follow the non-linear deformation path of the postbuckling field.
4. One of the primary considerations for the behavior of composite panels is the residual stiffness after buckling. To make this assessment complete the following steps.
  - a. In the Results Tree, expand the **History Output** branch for the output database file named **postBuckle.odb**.
  - b. From the list of available output, choose **Load proportionality factor: LPF for Whole Model** and click mouse button 3. From the menu that appears, select **Save As**. Name the data **LPF**.

- c. Similarly, save the curve for the U3 displacement (**Spatial displacement: U3 at Node ... in NSET POINT-LOAD**) and name it **u3**.
- d. In the Results Tree, double-click **XYData**.
- e. Select the source **Operate on XY data** and click **Continue**.
- f. From the **Operators** menu on the right, select **combine(X, X)**.
- g. In the **XY Data** menu on the left, select **U3** and click **Add to Expression**.
- h. In the **XY Data** menu, select **LPF** and click **Add to Expression**.
- i. Insert a minus (“–”) sign in front of **U3** in the top of the dialog box, so that the final expression reads **combine(–“U3”, “LPF” )**. This is only done to make the subsequent plot easier to read.
- j. Click **Plot Expression** and then click **Cancel**.

Figure W1–12 shows the load-displacement curve of the plate buckling analysis. This plot shows the variation in plate load with displacement of the node at which the load is applied. The initial slope of the curve is indicative of the initial stiffness of the plate. Buckling occurs at approximately 25000 psi. Finally a new load-displacement path is followed. The final slope of this path is shallower than the initial slope at the start of loading, and indicates the post-buckling reduction in stiffness.

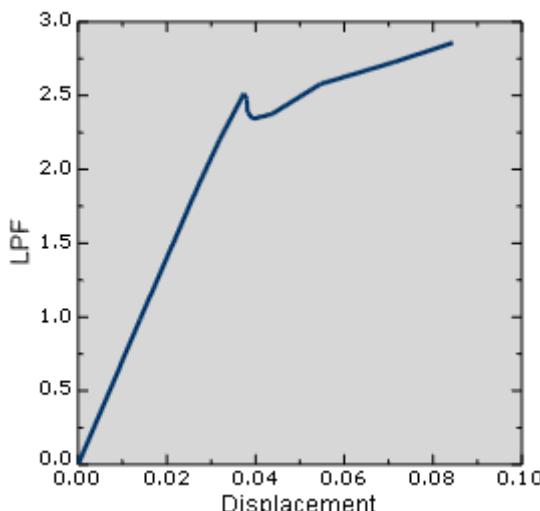


Figure W1–12. Load-displacement curve during buckling.

**Note: A script that creates the complete models described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions outlined here or if you wish to check your work. The script is named**

**`ws_composites_panel_answer.py`**

**and is available using the Abaqus fetch utility.**

# Notes

## Notes



## Workshop 2

### Composite Yacht Hull

**Note: This workshop provides instruction in terms of the Abaqus GUI interface. There is no Keywords version of this workshop.**

#### Goals

- Define the material properties of a fiber-matrix layer.
- Create a composite layup using Abaqus/CAE.
- Read the composite layup information from a text file.
- View a ply stack plot of a region/section.
- Use the Visualization module to view the results on an individual ply and create an envelope plot and a through thickness X-Y plot.

#### Introduction

Composite hulls are used routinely in the yacht industry. Composite materials allow manufacturers to create high-performance marine vessels that incorporate complex hull shapes. Composites also provide the strength, rigidity, and low mass that high-performance yachts require.

In this workshop you will define the composite material properties and the composite layup of a three-dimensional yacht hull model to study the pre- and postprocessing capabilities in Abaqus/CAE for analyzing large-scale composite structures.

Generally, a yacht model consists of the hull, mast, rigging, and keel, as shown in Figure W2–1. In this workshop the geometry of the model is imported as a single part from an ACIS (.sat) file, as shown in Figure W2–2. It includes only one half of the hull, and the center of the hull is constrained to be symmetric about the Y-axis. In general, a realistic composite hull is not symmetric; however, in this case, it enables the demonstration of composite modeling and postprocessing in Abaqus/CAE, consistent with the requirements of a training exercise. The hull represents a high-performance 20-meter yacht with reinforced bulkheads that stiffen the structure. The infrastructure above the deck does not play a role in modeling the performance of the hull and is not included in the model.

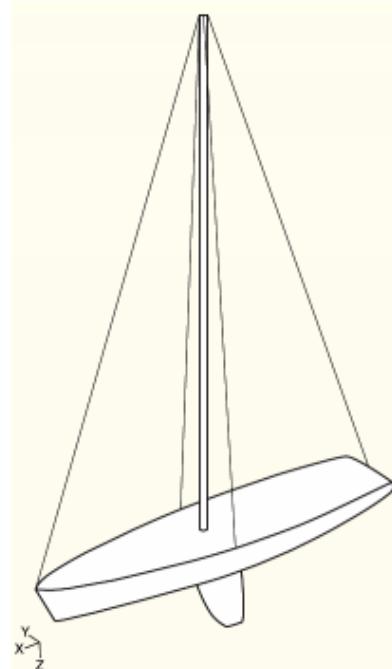


Figure W2-1. Yacht Model

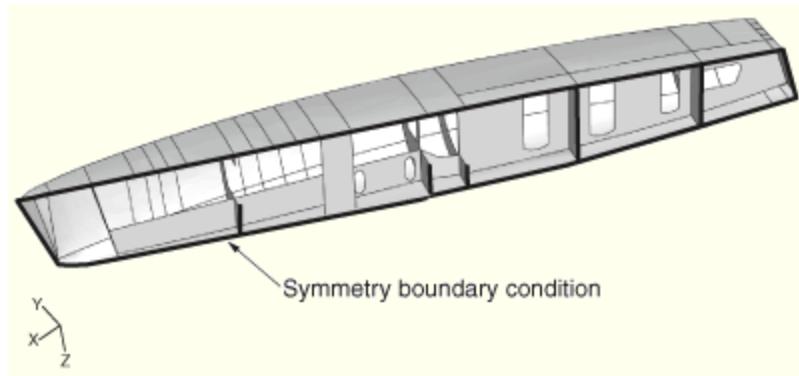


Figure W2-2. Model geometry and boundary condition

## Preliminaries

1. Enter the working directory for this workshop:  
`..../composites/interactive/hull`
2. Run the script `ws_composites_yacht_hull.py` using the following command:  
`abaqus cae startup=ws_composites_yacht_hull.py`

The above command creates an Abaqus/CAE database named `composite-hull.cae` in the current directory. The geometry, mesh, loading and boundary condition definitions for the composite hull are included in the model named **CompositeHull**. The shell

element type S4R is used in the model. You will complete the material definitions, create the composite layup definition, run the analysis, and postprocess the results.

## Defining materials

This hull model has been partitioned into 27 regions. Each region contains plies of glass-epoxy cloth surrounding a Nomex core. Most regions contain nine plies—four glass-epoxy plies on either side of the Nomex core, with the stacking sequence [0, 45, -45, 90, 0, 90, -45, 45, 0] degrees. However, additional plies are added as a patch to reinforce regions of high strain. Some bulkheads are reinforced by stringers with an effective Young's modulus of 1.28e5 MPa.

The material properties of the glass-epoxy cloth are

$$\begin{aligned}E_{11} &= 3.5\text{e}4 \text{ MPa}, \\E_{22} &= 7500 \text{ MPa}, \\G_{12} &= 3600 \text{ MPa}, \\G_{13} = G_{23} &= 3000 \text{ MPa}, \text{ and} \\v_{12} &= 0.3\end{aligned}$$

The material properties of the Nomex core are

$$\begin{aligned}E_{11} &= 10 \text{ MPa}, \\E_{22} &= 10 \text{ MPa}, \\G_{12} &= 1 \text{ MPa}, \\G_{13} = G_{23} &= 30 \text{ MPa}, \text{ and} \\v_{12} &= 0.3\end{aligned}$$

where the 1-direction is along the fibers, the 2-direction is transverse to the fibers in the surface of the lamina, and the 3-direction is normal to the lamina.

You will begin by defining the orthotropic elastic behavior of the lamina with the material properties given above. Note that the material properties of the stringers have already been defined.

1. Before defining materials, review the predefined stringer, load, and constraint definitions.
2. Define a material definition with the material properties of the glass-epoxy cloth.
  - a. In the Model Tree, double-click **Materials**.  
Abaqus/CAE switches to the Property module, and the material editor appears.
  - b. In the **Edit Material** dialog box, name the material **Glass-Epoxy**.
  - c. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic**.
  - d. Under the **Elastic** field, select **Lamina** as the type, as shown in Figure W2–3.



Figure W2–3. **Type** field in the material editor

- e. Enter the data for the orthotropic elastic material properties of the glass-epoxy cloth, as shown in Figure W2–4:

| Data |       |      |      |      |      |      |
|------|-------|------|------|------|------|------|
|      | E1    | E2   | Nu12 | G12  | G13  | G23  |
| 1    | 35000 | 7500 | 0.3  | 3600 | 3000 | 3000 |

Figure W2–4. **Glass-Epoxy** material properties

- f. Click **OK** to exit the material editor.
3. Using the similar procedure, define another lamina material definition named **Core** with the material properties of the Nomex core, as shown in Figure W2–5.

| Elastic                                                 |           |            |      |     |     |     |
|---------------------------------------------------------|-----------|------------|------|-----|-----|-----|
| Type:                                                   | Lamina    | Suboptions |      |     |     |     |
| <input type="checkbox"/> Use temperature-dependent data |           |            |      |     |     |     |
| Number of field variables:                              | 0         |            |      |     |     |     |
| Moduli time scale (for viscoelasticity):                | Long-term |            |      |     |     |     |
| <input type="checkbox"/> No compression                 |           |            |      |     |     |     |
| <input type="checkbox"/> No tension                     |           |            |      |     |     |     |
| Data                                                    |           |            |      |     |     |     |
|                                                         | E1        | E2         | Nu12 | G12 | G13 | G23 |
| 1                                                       | 10        | 10         | 0.3  | 1   | 30  | 30  |

Figure W2–5. **Core** material properties

## Defining the conventional shell composite layup

After defining the materials, you will create a conventional shell composite layup that includes all plies of the entire composite hull. Note that sets that correspond to the region of the composite layup to which plies will be applied have been already created in each region.

- In the Model Tree, expand the branch of the part named **compositehull** underneath the **Parts** container.
- Double-click **Composite Layups**.
- In the **Create Composite Layup** dialog box, name the composite layup **CompositeHull\_Layup**, choose **Conventional Shell** as the element type, and click **Continue**.

The composite layup editor appears.

4. In the composite layup editor, accept the default layup orientation system and integration settings.
5. Define the ply data using the ply table of the composite layup editor. Note that three rows are available in the ply table by default.

Since the geometry of the yacht hull is so complex, it will be too time consuming to enter all ply data manually. In this workshop, as a demo, you will (1) manually define the ply data of a region, e.g., the highlighted region (set **BLKHD\_3**) shown in Figure W2–6a; (2) manually add the local reinforcements on the highlighted region (set **BLKHD\_3\_PATCH**), as shown in Figure W2–6b; and (3) read all other ply data in the composite layup from a text file to complete the composite layup definition.

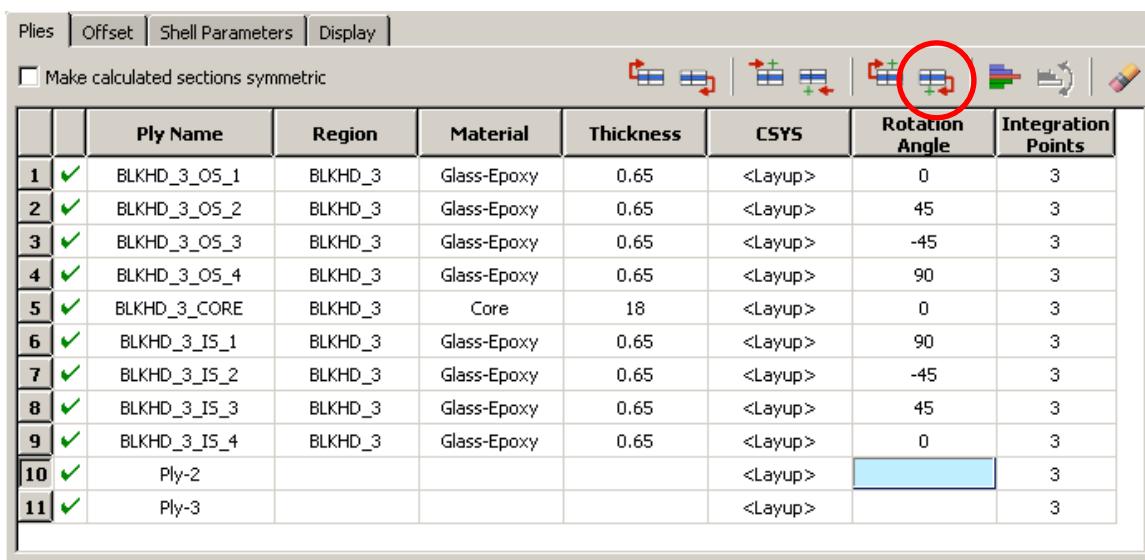


Figure W2–6. Highlighted regions: (a) set **BLKHD\_3**, (b) set **BLKHD\_3\_PATCH**

6. Enter the ply data of the layup for **BLKHD\_3** that contains nine plies—four glass-epoxy plies with the thickness of 0.65 mm on either side of the Nomex core with the thickness of 18 mm, with the stacking sequence [0, 45, -45, 90, 0, 90, -45, 45, 0].
  - a. In the ply table, rename the first ply **BLKHD\_3\_OS\_1**.
  - b. Double-click the cell in the first row under the **Region** column.
  - c. In the prompt area, click **Sets**. In the **Region Selection** dialog box that appears, select the set **BLKHD\_3** from the set list. If necessary, toggle on **Highlight the selections in viewport** to view your selection. Click **Continue**.
  - d. In the ply table, double-click the cell in the first row under the **Material** column.

- e. From the **Select Material** dialog box that appears, select the material **Glass-Epoxy** and click **OK**.
  - f. In the first row of the ply table, enter a value of **0 . 65** for **Thickness** and **0** for **Rotation Angle** (with respect to the X-direction of the default reference coordinate system).
  - g. Select any cell of ply **BLKHD\_3\_OS\_1** and click the **Copy Plies After** icon  above the ply table.
- A new ply named **BLKHD\_3\_OS\_1-Copy1** (a copy of the ply **BLKHD\_3\_OS\_1**) appears.
- h. Repeat the above step seven times to make seven more copies of the ply **BLKHD\_3\_OS\_1**.
  - i. Rename the plies for easy tracking during postprocessing and change the rotation angle accordingly for each ply, as shown in Figure W2–7. Change the material of the ply named **BLKHD\_3\_CORE** to **Core** and the thickness to **18**.

The ply table should look like Figure W2–7.



|    | Ply Name     | Region  | Material    | Thickness | CSYS    | Rotation Angle | Integration Points |
|----|--------------|---------|-------------|-----------|---------|----------------|--------------------|
| 1  | BLKHD_3_OS_1 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | 0              | 3                  |
| 2  | BLKHD_3_OS_2 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | 45             | 3                  |
| 3  | BLKHD_3_OS_3 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | -45            | 3                  |
| 4  | BLKHD_3_OS_4 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | 90             | 3                  |
| 5  | BLKHD_3_CORE | BLKHD_3 | Core        | 18        | <Layup> | 0              | 3                  |
| 6  | BLKHD_3_IS_1 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | 90             | 3                  |
| 7  | BLKHD_3_IS_2 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | -45            | 3                  |
| 8  | BLKHD_3_IS_3 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | 45             | 3                  |
| 9  | BLKHD_3_IS_4 | BLKHD_3 | Glass-Epoxy | 0.65      | <Layup> | 0              | 3                  |
| 10 | Ply-2        |         |             |           | <Layup> |                | 3                  |
| 11 | Ply-3        |         |             |           | <Layup> |                | 3                  |

Figure W2–7. **Plies** field in the composite layup editor

7. Add the local reinforcement on either side of the bulkhead. One is on top of the ply **BLKHD\_3\_OS\_1** and the other is under the ply **BLKHD\_3\_IS\_4**. Note that the ply data entered in the ply table is according to the stack order.
  - a. Insert a new row before the ply **BLKHD\_3\_OS\_1** to define the patch on the top under the ply **BLKHD\_3\_OS\_1**:
    - In the ply table, select any cell of ply **BLKHD\_3\_OS\_1**:
    - Click the **Insert Plies Before** icon 
    - Click the **Insert Plies Before** icon  above the ply table).
    - Name the new ply **BLKHD\_3\_OS\_PATCH**.

- Select the set **BLKHD\_3\_PATCH** as the region and **Glass-Epoxy** as the material.
  - Enter a value of **2** for the ply thickness.
  - Accept the default reference CSYS and enter a value of **0** for the rotation angle.
- b. Define the patch under the ply **BLKHD\_3\_IS\_4**. Rename the new ply **BLKHD\_3\_IS\_PATCH**. Select the set **BLKHD\_3\_PATCH** as the region and **Glass-Epoxy** as the material. Enter a value **2** for the ply thickness. Accept the default reference CSYS and enter a value of **0** for the rotation angle.

The ply table appears as shown in Figure W2–8.

|    |   | Ply Name         | Region        | Material    | Thickness | CSYS    | Rotation Angle | Integration Points |
|----|---|------------------|---------------|-------------|-----------|---------|----------------|--------------------|
| 1  | ✓ | BLKHD_3_OS_PATCH | BLKHD_3_PATCH | Glass-Epoxy | 2         | <Layup> | 0              | 3                  |
| 2  | ✓ | BLKHD_3_OS_1     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | 0              | 3                  |
| 3  | ✓ | BLKHD_3_OS_2     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | 45             | 3                  |
| 4  | ✓ | BLKHD_3_OS_3     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | -45            | 3                  |
| 5  | ✓ | BLKHD_3_OS_4     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | 90             | 3                  |
| 6  | ✓ | BLKHD_3_CORE     | BLKHD_3       | Core        | 18        | <Layup> | 0              | 3                  |
| 7  | ✓ | BLKHD_3_IS_1     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | 90             | 3                  |
| 8  | ✓ | BLKHD_3_IS_2     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | -45            | 3                  |
| 9  | ✓ | BLKHD_3_IS_3     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | 45             | 3                  |
| 10 | ✓ | BLKHD_3_IS_4     | BLKHD_3       | Glass-Epoxy | 0.65      | <Layup> | 0              | 3                  |
| 11 | ✓ | BLKHD_3_IS_PATCH | BLKHD_3_PATCH | Glass-Epoxy | 2         | <Layup> | 0              | 3                  |
| 12 | ✓ | Ply-3            |               |             |           | <Layup> |                | 3                  |

Figure W2–8. **Plies** field in the composite layup editor

8. Read all other ply data into the ply table from the provided text file.
  - In the composite layup editor, click mouse button 3 on any cell in the ply table and select **Read From File** from the menu that appears.  
The **Read Data from ASCII file** dialog box that appears.
  - In the **Read Data from ASCII file** dialog box, click **Select** to select the text file **ws\_composites\_yacht\_hull\_layup.txt**, enter **12** for **Start reading values into table row** and **2** for **Start reading values into table column**, and click **OK** to read the pre-stored data into the ply table.
  - Review the data written into the ply table.
9. Click **OK** to save the data and to exit the composite layup editor.

## Viewing ply stack plots

You will now view the ply stack plot for selected regions of the shell composite layup.

1. In the Property module, click the **Query information** tool  in the tool bar.
2. In the **Query** dialog box, select **Ply stack plot** from the list of **Property Module Queries**.

- Note that a new viewport is created and tiled vertically with respect to the original viewport.
3. In the viewport displaying the part, select the highlighted region shown in Figure W2–9a. The ply stack plot of this region appears in the new viewport, as shown in Figure W2–9b.

Note that the ply stack plot is with respect to the local coordinate system.

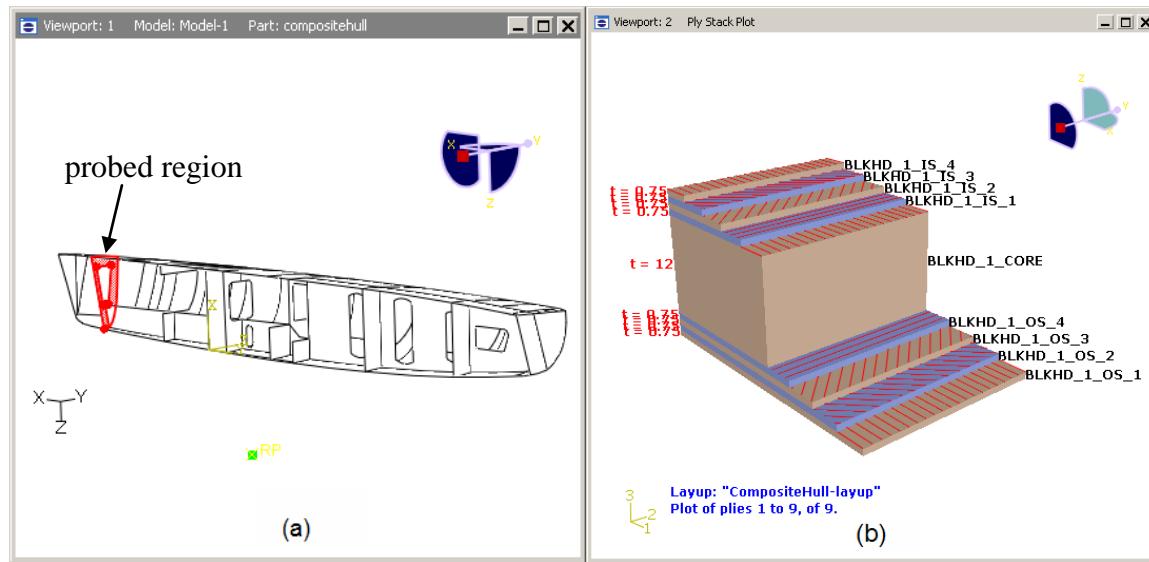


Figure W2–9. Stack ply plot of a probed region

4. Probe the region shown in Figure W2–10a. The ply stack plot of this region appears in a new viewport, as shown in Figure W2–10b.

Note that a five-layer patch is applied at this region.

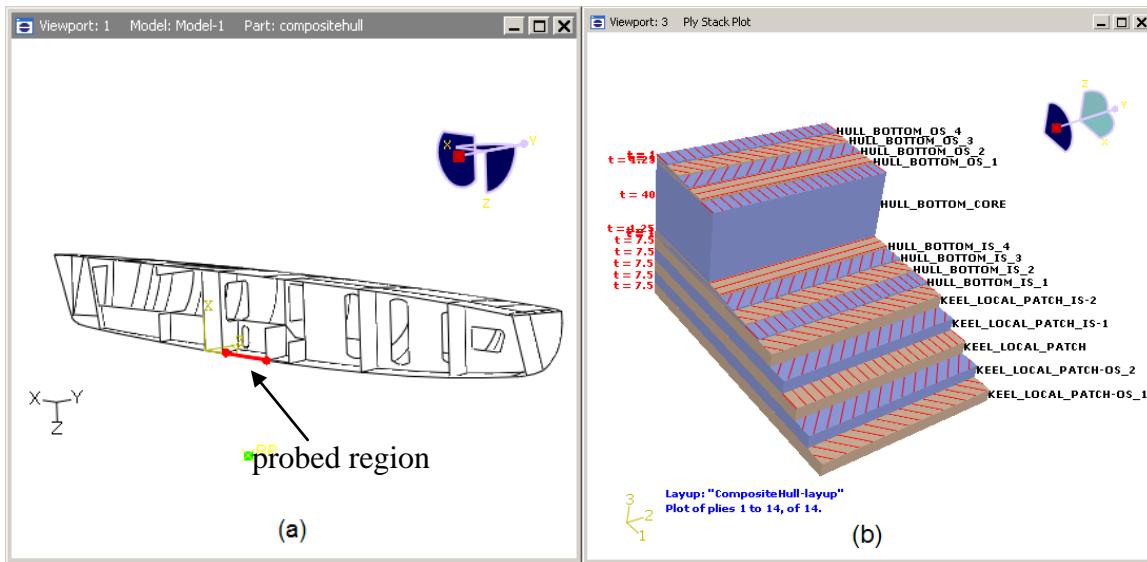


Figure W2–10. Stack ply plot of a patched region

5. You can also probe other regions to view their ply stack plots, if you wish.
6. Close all viewports displaying the ply stack plots and maximize the viewport displaying the part.

## Defining composite layup output

By default, Abaqus/CAE writes field output data from only the top and bottom section points of a composite layup, and no data are generated from the other plies. Here you will create a new field output request to write the output at all section points in all plies of the composite layup.

1. In the Model Tree, double-click **Field Output Requests**. In the **Create Field** dialog box, name the field output request **composite\_hull\_layup** and click **Continue**.
- The field output request editor appears.
2. In the field output request editor, select **Composite layup** as the domain and **compositehull-1.CompositeHull-layup** as the layup.
  3. In the output variable list, expand the branch of **Stresses** and toggle on **S, Stress components and invariants**; expand the branch of **Strains** and toggle on **E, Total strain components**.
  4. At the bottom of the field output request editor, choose the output request at **All section points in all plies**.
- Note that you can also request output at specific section points for each ply.
5. Click **OK** to exit the field output request editor.

## Running the job and postprocessing the analysis

1. Create a job named **compHull** with the following description: **Analysis of a composite yacht hull**.  
**Tip:** to create a job, double-click **Jobs** in the Model Tree.
2. Save your model database file, and submit the job for analysis (in the Model Tree, click mouse button 3 on the job name and select **Submit** from the menu that appears). From the same menu, you can select **Monitor** to monitor the job's progress.

When the analysis is complete, use the following procedure to view the ply-based results in the Visualization module:

1. In the Model Tree, click mouse button 3 on the job **compHull** and select **Results** from the menu that appears to open the file **compHull.odb** in the Visualization module.
2. Plot the deformed model shape by click the **Plot Deformed Shape** tool  in the toolbox.  
Note that the deformation scale factor is not 1. Adjust the deformation scale factor, if necessary, to see the actual deformed configuration.
3. From the main menu bar, select **Result→Section Points**. The **Section Points** dialog box appears. Choose **Plies** as the selection method. A list of all plies in the composite layup appears in the **Plies** field, as shown in Figure W2–11. You can select any individual ply to display output.

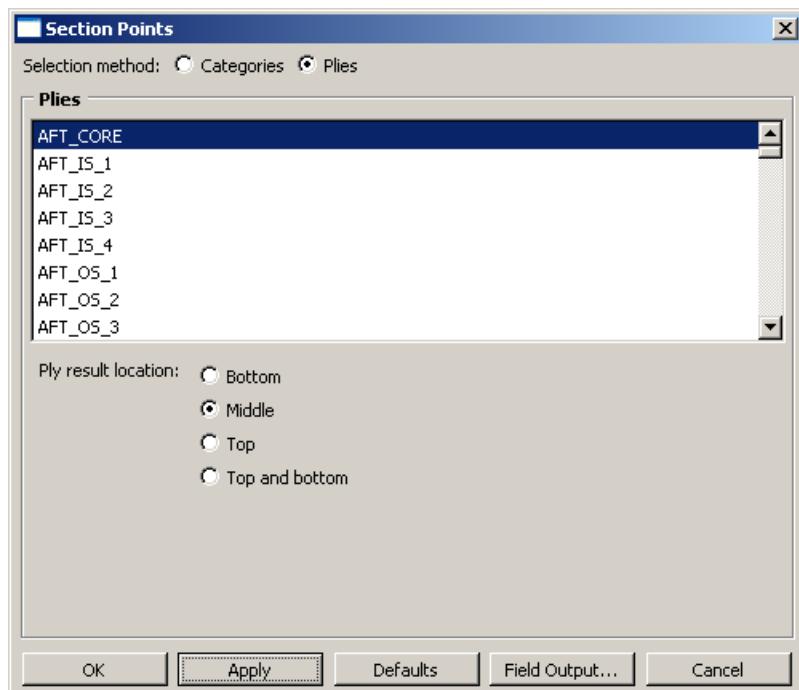


Figure W2–11. **Plies** field in the **Section Points** dialog box

4. From the list in the **Plies** field, locate and select the ply **BLKHD\_3\_OS\_1**. Accept **Middle** as the result location. Click **Apply**.

5. Click the **Plot Contours on Deformed Shape** tool  in the toolbox.

A contour plot of the Mises stress at the middle of the ply **BLKHD\_3\_OS\_1** appears. Adjust your view, if necessary, to see the results more clearly.

6. Contour the stress component along the fiber direction (variable S11) of **BLKHD\_3\_OS\_1**. Use the **Field Output** toolbar to change the displayed output variable.

7. For a better view, create a display group displaying only the set **BLKHD\_3** of the model.

- In the Results Tree, expand the **Element Sets** container for the output database file **compHull.odb**.

- Click the **Element Sets** container and press **F2**.

- Filter the container according to **\*BLKHD\_3\***.

- Locate and choose the set **COMPOSITEHULL-1.BLKHD\_3**, and click mouse button 3. From the menu that appears, select **Replace**.

The set **COMPOSITEHULL-1.BLKHD\_3** with the S11 contour plot at the middle of ply **BLKHD\_3\_OS\_1** is displayed in the viewport.

8. In the **Section Points** dialog box, select the plies **BLKHD\_3\_OS\_2** through **BLKHD\_3\_IS\_4** in turn to display output for each ply.

9. You can also view the results of the patch on the either side of **BLKHD\_3**.

- In the **Section Points** dialog box, select the ply named **BLKHD\_3\_OS\_PATCH** and click **Apply** to view the S11 contour plot of the patch at the top of **BLKHD\_3**.

- Select the ply named **BLKHD\_3\_IS\_PATCH** and click **Apply** to view the S11 contour plot of the patch at the bottom of **BLKHD\_3**.

10. View the ply stack plot of a probed section.

- Click the **Query information** tool  in the tool bar.

- In the **Query** dialog box, select **Ply stack plot** from the list of **Visualization Module Queries**.

- In the viewport displaying **BLKHD\_3**, click anywhere on the mesh that displays the contour plot to select the section, as shown in Figure W2-12a.

The ply stack plot of this section appears in a new viewport that is titled vertically with respect to the original viewport, as shown in Figure W2-12b.

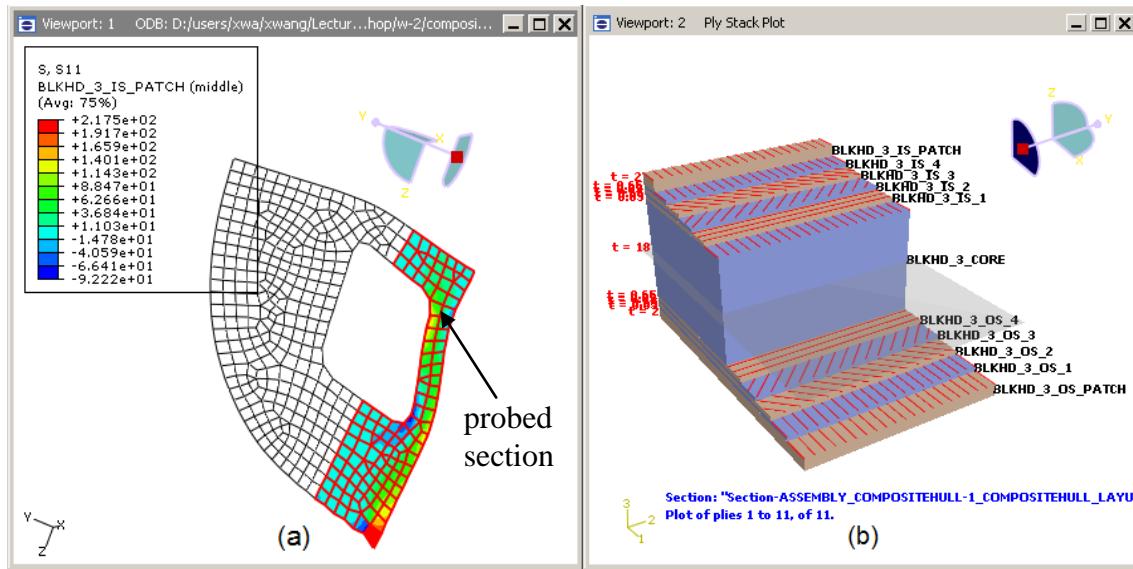


Figure W2–12. Ply stack plot of a probed section

11. Close the viewport displaying the ply stack plot and maximize the viewport displaying **BLKHD\_3** for further postprocessing.
12. Create a through-thickness X–Y plot to view the variation of the strain in the fiber direction (variable E11) across the entire thickness of a shell element.
  - a. In the Results Tree, double-click **XYData**.
  - b. In the **Create XY Data** dialog box, select **Thickness** as the source and click **Continue**.  
The **XY Data From Shell Thickness** dialog box appears.
  - c. In the variable list of the **Variables** tabbed page, expand the branch of **E: Strain components** and toggle on **E11**.
  - d. In the **Elements** tabbed page, accept **Pick from viewport** as the selection method and click **Edit Selection** to select an element, as shown in Figure W2–13.

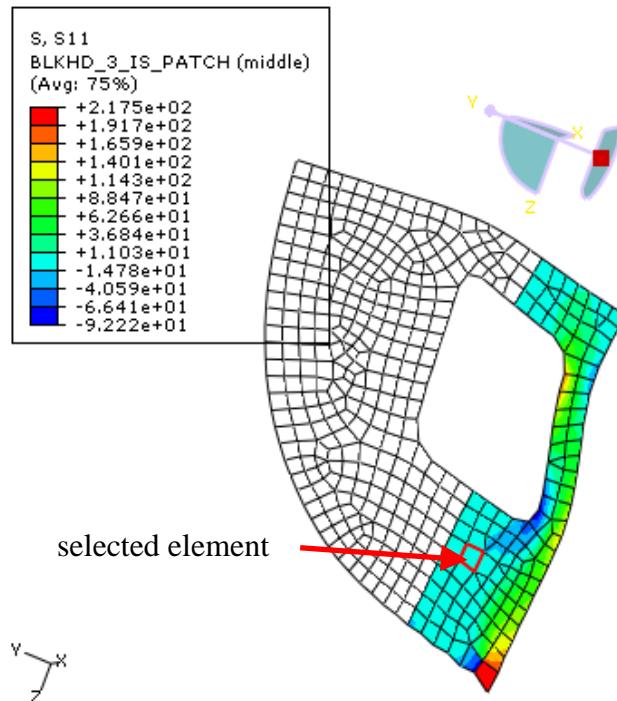


Figure W2–13. Ply stack plot of a probed region

- In the **XY Data From Shell Thickness** dialog box, click **Plot**.

The through-thickness X–Y plot of E11 appears, as shown in Figure W2–14.

The strain is discontinuous because the orientation of the fiber changes between the plies (refer to the ply stack plot in Figure W2–12b).

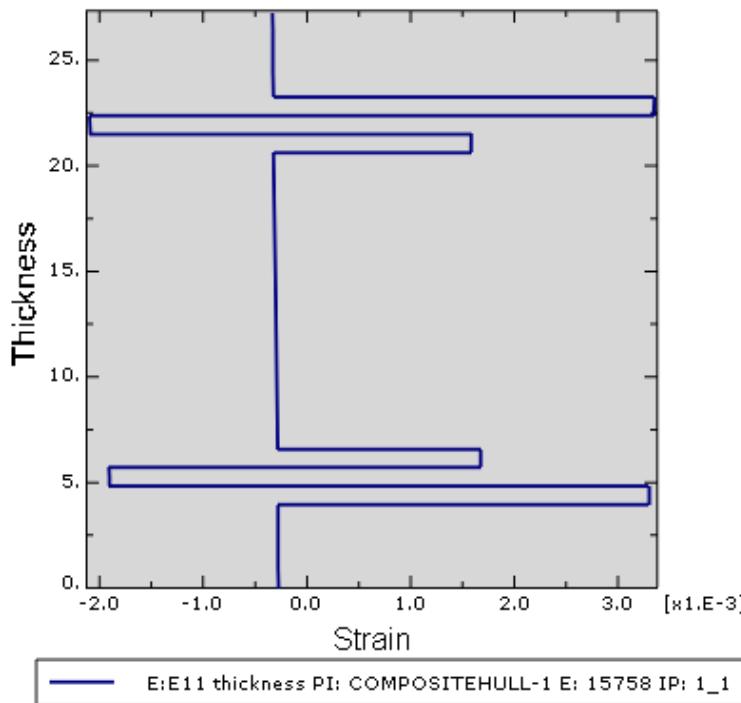


Figure W2–14. Strain (E11) across the thickness of an element

13. Create an *envelope* plot to show the critical plies in **BLKHD\_3**.
  - a. In the **Section Points** dialog box, choose **Categories** as the selection method and then **Envelope** as the active location.
  - b. Select **Max value** as the criterion and **Integration point** as the position.
  - c. Click **OK**.
  - d. Click to create a contour plot the stress envelope of the shell.
  - e. Use the contour plot options to display a contour plot showing the names of the critical plies to identify the critical ply and the stress value in the critical ply.
14. To view the envelope plot of the entire composite hull, click the **Replace All** tool in the tool bar to restore the entire model.
15. Similarly, create a display group of any other set and use the **Section Points** dialog box to select a ply in that set and view the output.

**Note:** A script that creates the complete model described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions outlined here or if you wish to check your work. The script is named `ws_composite_yacht_hull_answer.py` and is available using the Abaqus fetch utility.



## Notes

# Notes



## Workshop 3

### Perforation of a Composite Plate

#### Interactive Version

**Note:** This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the “Keywords” version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

#### Goals

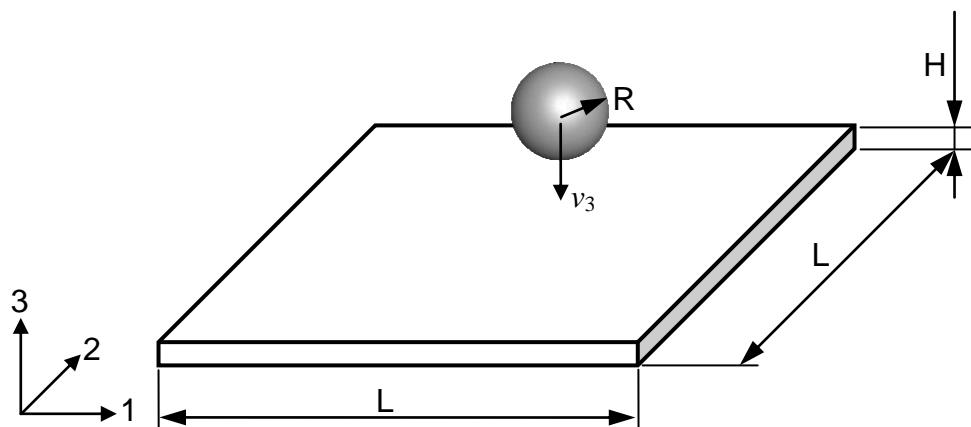
- Define composite material properties.
- Define a composite layup with different layer orientations.
- Perform an impact analysis.
- Use the Visualization module to view the material orientations, create contour plots on different layers, and investigate composite damage and failure.

#### Introduction

Because composite materials offer high directional stiffness at significant weight savings when compared to metals, they continue to find increased use in many industries. These include civilian and military vehicles, marine craft, and aircraft. In addition, composite materials such as Kevlar are used extensively in the design of protective armor. Whether attempting to predict the ballistic limit velocity of body armor or the resistance of an aircraft panel to impact from runway debris, it is critical to accurately predict damage due to impact on composite materials.

In this workshop, you will analyze the normal impact of a rigid steel ball onto a composite plate at a velocity of 1e5 mm/sec, using Abaqus/Explicit. A schematic of the model is shown in Figure W3–1. You will define the composite material properties and layup, study the ability of the general contact algorithm to model surface erosion on multiple contacting bodies during high-speed impact, and investigate composite damage and failure. Figure W3–2 shows the details of the model, including the geometry and boundary conditions. The plate is fully clamped along its edges.

The composite plate is made of 8 layers of unidirectional carbon fibers in an epoxy resin in a { 0/90/ $\pm 45$ / $\mp 45$ /90/0 } layup. Each layer has a thickness of 0.2 mm. To simulate impact on composite materials, damage and failure modeling is critical since it allows the



Geometric properties:

$$L = 100 \text{ mm}$$

$$H = 1.6 \text{ mm}$$

$$R = 2.5 \text{ mm}$$

$$h_{\text{layer}} = 0.2 \text{ mm}$$

The velocity of the ball:

$$v_3 = -1e5 \text{ mm/sec}$$

Figure W3–1. Impact of a rigid ball onto a flat composite plate.

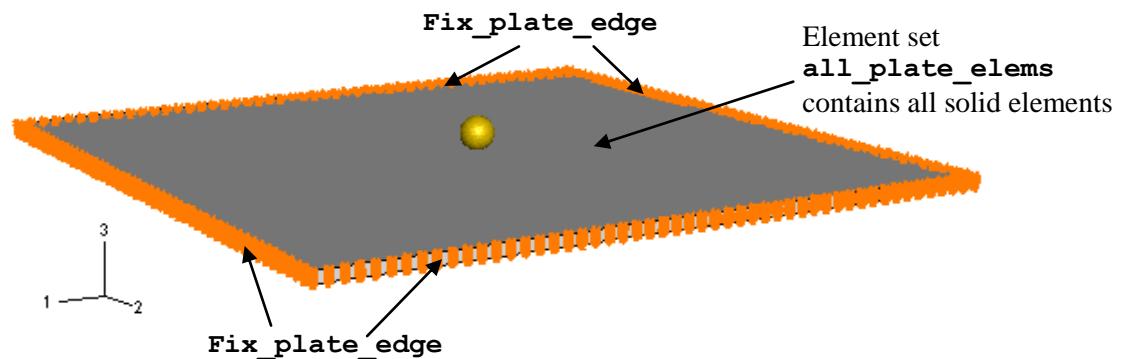


Figure W3–2. Model geometry and boundary conditions.

ball to perforate the plate. In this workshop, a user-defined material model is provided as a VUMAT. The material model is based on Hashin's failure criteria for unidirectional fiber composites (Hashin, 1980) and considers four possible failure modes: matrix compression and tension, fiber tension, and fiber compression (fiber buckling). The source code is in **uniFiber.for**; if it is not already provided to you, you can download this user subroutine from SIMULIA Answer 3123.

Typical linear elastic properties are used for the carbon fiber composite (CFC) material:

$$\begin{aligned}E_{11} &= 1.64e5 \text{ MPa}, \\E_{22} = E_{33} &= 1.2e4 \text{ MPa}, \\G_{12} = G_{13} &= 4500 \text{ MPa}, \\G_{23} &= 2500 \text{ MPa}, \\\nu_{12} = \nu_{13} &= 0.32, \text{ and} \\\nu_{23} &= 0.45\end{aligned}$$

The strength properties are:

$$\begin{aligned}\text{Tensile failure stress in fiber direction: } X_{1t} &= 2724 \text{ MPa}, \\\text{Compressive failure stress in fiber direction: } X_{1c} &= 111 \text{ MPa}, \\\text{Tensile failure stress in direction 2: } X_{2t} &= 50 \text{ MPa}, \\\text{Compressive failure stress in direction 2: } X_{2c} &= 1690 \text{ MPa}, \\\text{Tensile failure stress in direction 3: } X_{3t} &= 290 \text{ MPa}, \\\text{Compressive failure stress in direction 3: } X_{3c} &= 290 \text{ MPa}, \\\text{Shear strength in 12 plane: } S_{12} &= 120 \text{ MPa}, \\\text{Shear strength in 13 plane: } S_{13} &= 137 \text{ MPa}, \\\text{Shear strength in 23 plane: } S_{23} &= 90 \text{ MPa}\end{aligned}$$

where the 1-direction is along the fibers, the 2-direction is transverse to the fibers in the surface of the ply, and the 3-direction is normal to the ply.

## Preliminaries

1. Enter the working directory for this workshop:  
`.. /composites/interactive/impact`
2. Run the script `ws_composites_impact.py` using the following command:  
`abaqus cae startup=ws_composites_impact.py`

The above command creates an Abaqus/CAE database named `composites_impact.cae` in the current directory. The model includes the geometry and boundary condition definitions of the plate, mesh and material definitions of the ball, initial ball velocity, and the explicit dynamic step. Note that a mass scaling factor of 160 is used to speed up the analysis. In general, mass scaling is not suitable for impact problems since the dynamic response is of primary importance; however, in this case, it enables the job to run in a reasonable time frame, consistent with the requirements of a training exercise. Note that the results obtained in this workshop may not be physically realistic due to mass scaling. In order to obtain results that are more physically meaningful, you may re-run this job without mass scaling at a later time.

You will add the necessary data to complete the model.

## Creating layers to define composite structure

You will create 8 layers along the thickness of the plate using the partition tool; these layers will be used to define the composite structure.

1. In the Model Tree, expand the **Parts** container and double-click the **plate** subcontainer.  
Abaqus/CAE switches to the Part module and makes current the part named **plate**.
2. In the **Views** toolbar, click the **Apply Bottom View** tool . The model is now oriented so that the global Z-axis is vertical. Click the **Turn Perspective Off** tool in the toolbar to use parallel projection.  
**Note:** If necessary, select **View**→**Toolbars**→**View** from the main menu to display the **Views** toolbar.
3. Use the **Partition Cell: Use Datum Plane** tool to partition the plate. Note that seven equally-spaced datum planes have been predefined; the distance between each plane and its neighboring planes is 0.2 mm, as shown in Figure W3–3. Adjust your view, if necessary, to see the model geometry and datum planes more clearly.

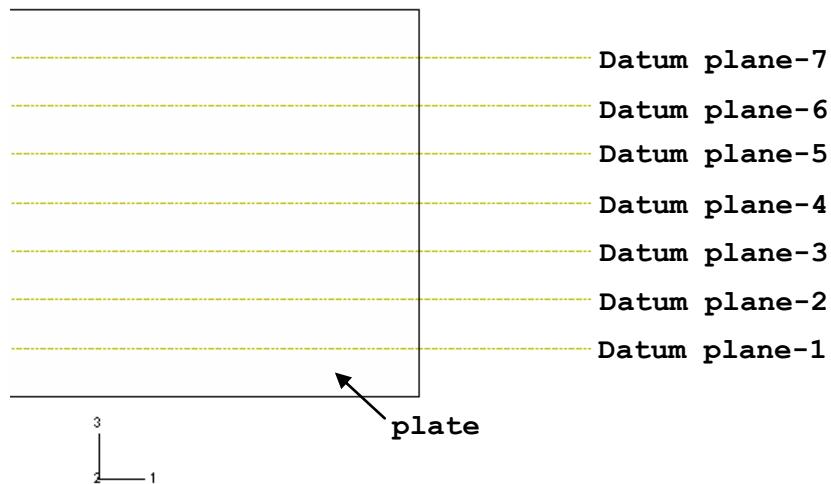


Figure W3–3. Datum planes

- a. Note the small black triangles at the base of the toolbox icons. These triangles indicate the presence of hidden icons that can be revealed. Click the **Partition Cell: Define Cutting Plane** tool but do not release the mouse button. When additional icons appear, release the mouse button.
- b. Select the **Partition Cell: Use Datum Plane** tool . It appears in the toolbox with a white background indicating that you selected it.
- c. Select the bottom datum plane (**Datum plane-1**), as shown in Figure W3–3.

- d. Click mouse button 2 in the viewport or click **Create Partition** in the prompt area to partition the plate. The first layer appears at the bottom of the plate.  
Note that the cell partition interface remains active and you are prompted to **Select the cells to partition**.
  - e. Continue partitioning the plate using the remaining datum planes.
  - f. When you have finished, click mouse button 2 in the viewport or click **Done** in the prompt area to exit the cell partition interface.
4. Define a part-level set for each layer, as shown in Figure W3–4. In the viewport, double-click the 3D Compass. In the **Specify View** dialog box, select **Viewpoint** as the specification method, enter **1, 1, 0.2** as the viewpoint and **0, 0, 1** as the up vector, and click **OK**. Adjust your view, if necessary, to see the model geometry more clearly.

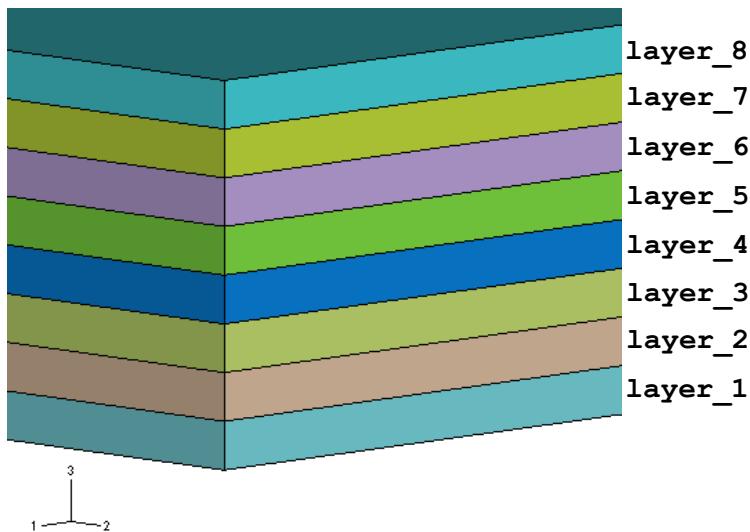


Figure W3–4. Model sets

- a. In the Model Tree, expand the branch of the part **plate**.
- b. Double-click on the **Sets** container.
- c. In the **Create Set** dialog box, name the set **layer\_1** and click **Continue**.
- d. Select the bottom layer (see Figure W3–4) as the region which will be defined in the set **layer\_1**.

**Tip:** Use the **Selection** toolbar to restrict your selections to **Cells**.

- e. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected the geometry. The set **layer\_1** is created.

Use a similar procedure to create sets **layer\_2** through **layer\_8**, as shown in Figure W3–4. To distinguish between the different layers as you select them, color code the part based on sets.

## Defining material properties

You will define the material properties of the composite plate, which include the density, elastic properties, failure criteria, and solution-dependent variables.

1. Define the material density.
  - a. In the Model Tree, double-click the **Materials** container.  
Abaqus/CAE switches to the Property module, and the material editor appears.
  - b. In the **Edit Material** dialog box, name the material **CFC**.
  - c. From the material editor's menu bar, select **General→Density**.
  - d. In the **Density** field, enter the value **1.e-9** for **Mass Density**.
2. Define the material constants and solution-dependent state variables which will be used in the VUMAT user subroutine.
  - a. From the material editor's menu bar, select **General→User Material**.
  - b. Under the **User Material** field, select the material type **Mechanical**. In the **Data** table, enter the 27 material constants, as shown in Figure W3–5.  
Note that only 19 of these material constants are meaningful; they include the elastic properties and failure criteria noted earlier. Recall that the other 8 are essentially “padding” to accommodate the input data structure required by the VUMAT routine.
  - c. From the material editor's menu bar, select **General→Depvar**.
  - d. The VUMAT user subroutine requires 17 solution-dependent state variables and variable 5 is the controlling parameter to control element deletion.  
In the **Depvar** field, enter the value **17** for **Number of solution-dependent state variables** and **5** for **Variable number controlling element deletion (Abaqus/Explicit only)**.
  - e. Click **OK** to exit the material editor.

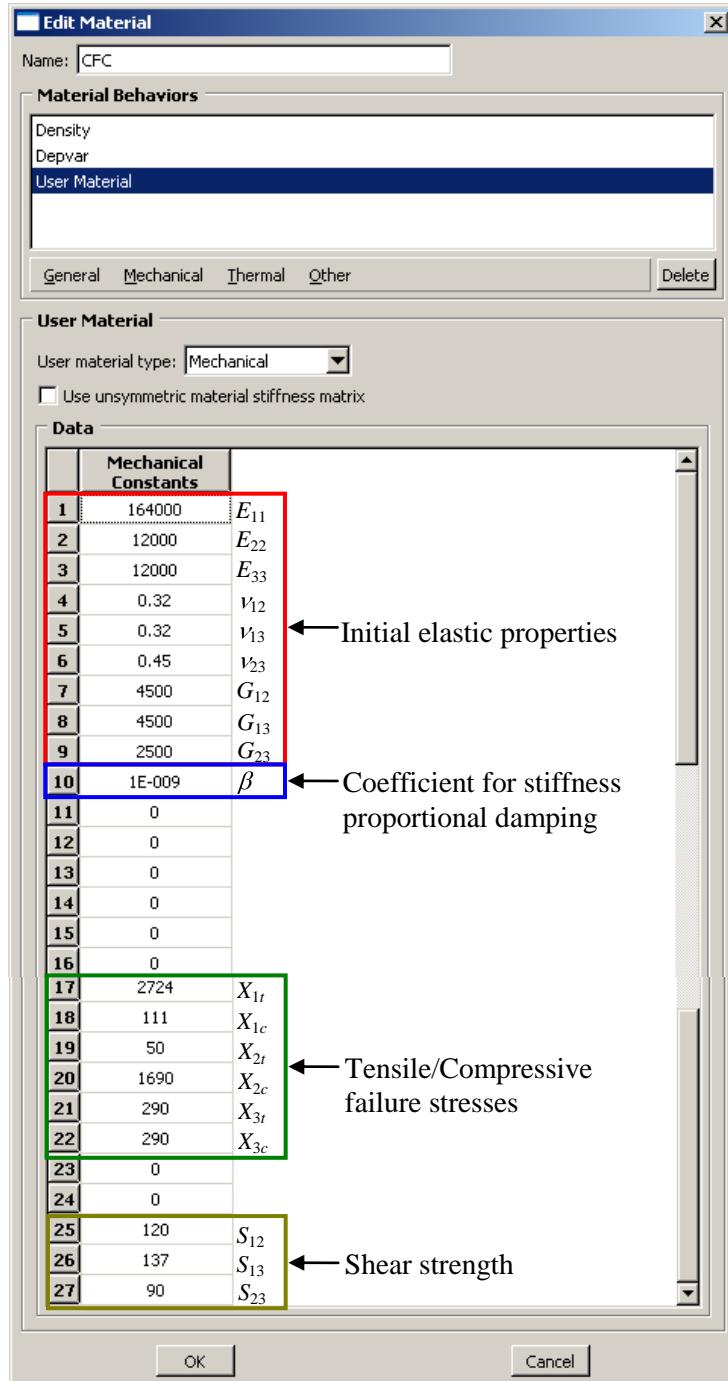


Figure W3–5. The material editor

3. Define a solid section which will be assigned to the composite plate.
  - a. In the Model Tree, double-click the **Sections** container.

- b. In the **Create Section** dialog box, name the section **plate\_section**, choose the category **Solid** and the type **Homogenous**, and click **Continue**.  
The section editor appears.
  - c. In the **Edit Section** dialog box, choose the material **CFC**.
  - d. Click **OK** to exit the section editor.
4. Assign the section definition to the part **plate**.
- a. In the Model Tree, expand the branch of the part **plate**.
  - b. Double-click **Section Assignments**.
  - c. Select the entire part as the region to which the section will be assigned.
  - d. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.
  - e. In the **Edit Section Assignment** dialog box, select **plate\_section** and click **OK**.  
Abaqus/CAE colors the plate green to indicate that the section has been assigned.
  - f. Click mouse button 2 again in the viewport or click **Done** in the prompt area to exit the section assignment interface.

## Defining the composite layup

You will define four datum coordinate systems (CSYSs) representing four material orientations (noted earlier) and then assign them to the appropriate layers of the composite plate.

- 1. Define a datum coordinate system (CSYS) named **zero\_degree** with its local 1-axis along the global X-direction and its local 3-axis along the global Z-direction.
  - a. Click the **Create Datum CSYS: 3 Points** tool  in the toolbox.
  - b. In the **Create Datum CSYS** dialog box, name the datum CSYS **zero\_degree**, select the **Rectangular** coordinate system type, and click **Continue**.
  - c. Enter **0.0,0.0,0.0** as the origin, **1.0,0.0,0.0** as the coordinates of a point on the X-axis, and **0.0,1.0,0.0** as the coordinates of a point in the X-Y plane to create the datum CSYS.
- 2. Using a similar procedure, create the following three CSYSs centered at the origin.
  - a. Create a datum CSYS named **ninety\_degrees** with its local 1-axis along the global Y-direction and its local 3-axis along the global Z-direction.

- b. Create a datum CSYS named **plus四十five\_degrees** with its local 1-axis at  $+45^\circ$  degrees with respect to the global X-direction and its local 3-axis along the global Z-direction.
  - c. Create a datum CSYS named **minus四十five\_degrees** with its local 1-axis at  $-45^\circ$  degrees with respect to the global X-direction and its local 3-axis along the global Z-direction.
3. Use the **Assign Material Orientation** tool  to assign the material orientation to each layer according to the layup noted earlier.
- a. Click the **Assign Beam Orientation** tool  but do not release the mouse button. When additional icons  appear, release the mouse button. Select the **Assign Material Orientation** tool .
  - b. Click **Sets** in the prompt area. In the **Region Select** dialog box, select the set **layer\_1** and click **Continue**.
  - c. Click **Datum CSYS List** in the prompt area. In the **Datum CSYS List** dialog box, select the datum CSYS **zero\_degree** and click **OK**.
  - d. In the **Edit Material Orientation** dialog box, accept **Axis 3** as the default direction for additional axis rotation with zero additional rotation.
  - e. Click **OK** to confirm the input.
  - f. Using a similar procedure, assign the following material orientations. For all assignments, specify no additional rotation about the 3-axis:

| Layer          | CSYS                       |
|----------------|----------------------------|
| <b>layer_2</b> | <b>ninety_degrees</b>      |
| <b>layer_3</b> | <b>plus四十five_degrees</b>  |
| <b>layer_4</b> | <b>minus四十five_degrees</b> |
| <b>layer_5</b> | <b>minus四十five_degrees</b> |
| <b>layer_6</b> | <b>plus四十five_degrees</b>  |
| <b>layer_7</b> | <b>ninety_degrees</b>      |
| <b>layer_8</b> | <b>zero_degree</b>         |

## Meshing the model

You will use the Mesh module to generate the finite element mesh for the composite plate. Note that the global seeds, element type, and mesh controls have been predefined.

1. In the Model Tree, double-click **Mesh** in the branch for the part **Plate**.  
Abaqus/CAE switches to the Mesh module and displays the part **Plate**.

2. Click the **Mesh Part** tool  in the toolbox.
3. Click mouse button 2 in the viewport or click **Yes** in the prompt area to mesh the plate with the solid element type C3D8R.

## Defining damage output request

The preselected default output does not include the damage output variables STATUS and SDV. STATUS is used to identify elements which have failed, and SDVi are the solution-dependent variables which include tensile and compressive damage in the 1- and 2-directions. To visualize the damage and failure in the Visualization module, you will write additional field output to the output database file.

1. In the Model Tree, double-click the **Field Output Requests** container. Accept the default name and step selection, and click **Continue**.  
The field output request editor appears.
2. In the **Edit Field Output Request** dialog box, expand the **State/Field/User/Time** list in the **Output Variables** field and toggle on the variables **SDV** and **STATUS**.
3. Click **OK** to exit the field output editor.

## Defining general contact

General contact will be defined for this impact analysis. Since erosion will occur during the analysis (failure of elements will occur as they become fully damaged), contact of the rigid steel ball must be allowed not only with the external surfaces of the plate but also with its internal surfaces. By default, general contact in Abaqus/Explicit allows for contact between all exterior surfaces in a model including a surface with itself. To include interior surfaces in the contact domain, you must define a surface that includes the interior faces of the plate. Note that a part-level surface named **plate\_interior\_surf** has been predefined on the exterior surfaces of the plate. This surface definition will be modified to include only the interior surfaces of the plate by manually editing the input file. You will first define general contact and then edit the interior surface definition.

1. Define the contact property.
  - a. In the Model Tree, double-click **Interaction Properties** to create a **Contact** property named **Friction**.
  - b. In the **Edit Contact Property** dialog box, select **Mechanical→Tangential Behavior** and choose the **Penalty** friction formulation.
  - c. Enter a friction coefficient of **0 . 5**.
  - d. Click **OK** to exit the contact property editor.
2. Define general contact.
  - a. In the Model Tree, double-click **Interactions**.

- b. In the **Create Interaction** dialog box, name the interaction **sphere\_plate**, select **impact** as the step, and select **General contact (Explicit)** as the interaction type. Click **Continue**.

The interaction editor appears.

- c. In the **Edit Interaction** dialog box, choose **Selected surface pairs** and click **Edit** in the **Contact Domain** field, as shown in Figure W3–6.

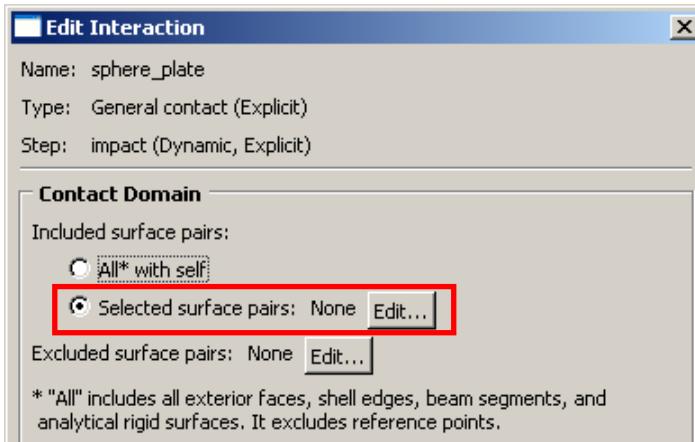


Figure W3–6. Contact Domain field in the interaction editor

- d. In the **Edit Included Pairs** dialog box, select the pairs to be included in general contact, as shown in Figure W3–7.

Note that the following definitions allow for (1) contact between the ball and all surfaces (interior and exterior) of the plate, and (2) self-contact between all surfaces (interior and exterior) of the plate.

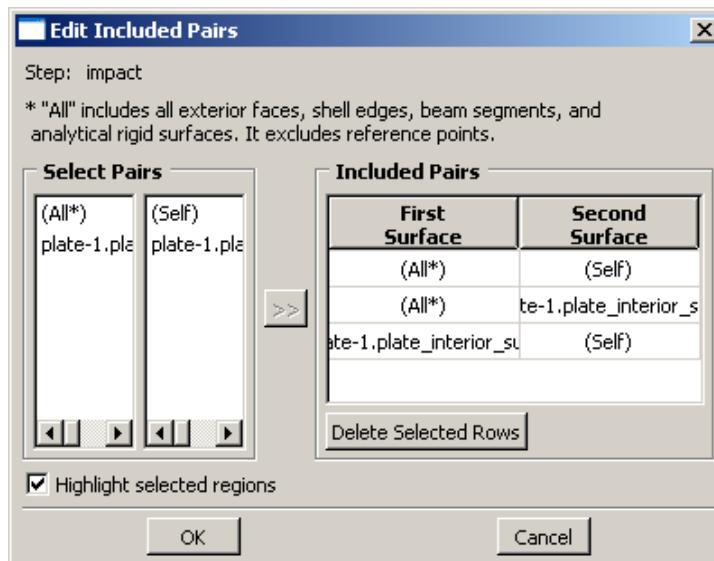


Figure W3–7. Edit Included Pairs dialog box

- e. Click **OK** to close the dialog box.

- f. Assign the global contact property.
  - g. Click **OK** to exit the interaction editor.
3. Save your model database file, create a job named **plate\_impact**, and write the input file.
- Tip:** In the Model Tree, click mouse button 3 on the job name and select **Write Input** from the menu that appears.
4. In the working directory of this workshop, use your text editor to open the file **plate\_impact.inp**.
  5. Locate the **\*SURFACE, TYPE=ELEMENT, NAME=plate\_interior\_surf** option. Modify the option so it appears as follows:
- ```
*SURFACE, TYPE=ELEMENT, NAME=plate_interior_surf
all_plate_elems, interior
```
6. Save and close the modified input file.

Run the job and postprocessing the results:

Run the analysis by entering the following command at the prompt in the working directory of this workshop:

```
abaqus job=plate_impact user=uniFiber
```

When the analysis is complete, use the following procedure to view the stress contour plots in different layers and investigate damage and failure of the plate in the Visualization module:

1. Open **plate_impact.odb** in the Visualization module.
 2. Click the **Plot Contours on Deformed Shape** tool  to contour the stress state in the model. Notice that the failed elements are removed automatically.
- Note:** Automatic removal of failed elements is enabled when the STATUS output variable is written to the output database. The **Status Variable** tabbed page of the **Field Output** dialog box provides other options for the removal of failed elements, as shown in Figure W3–8.

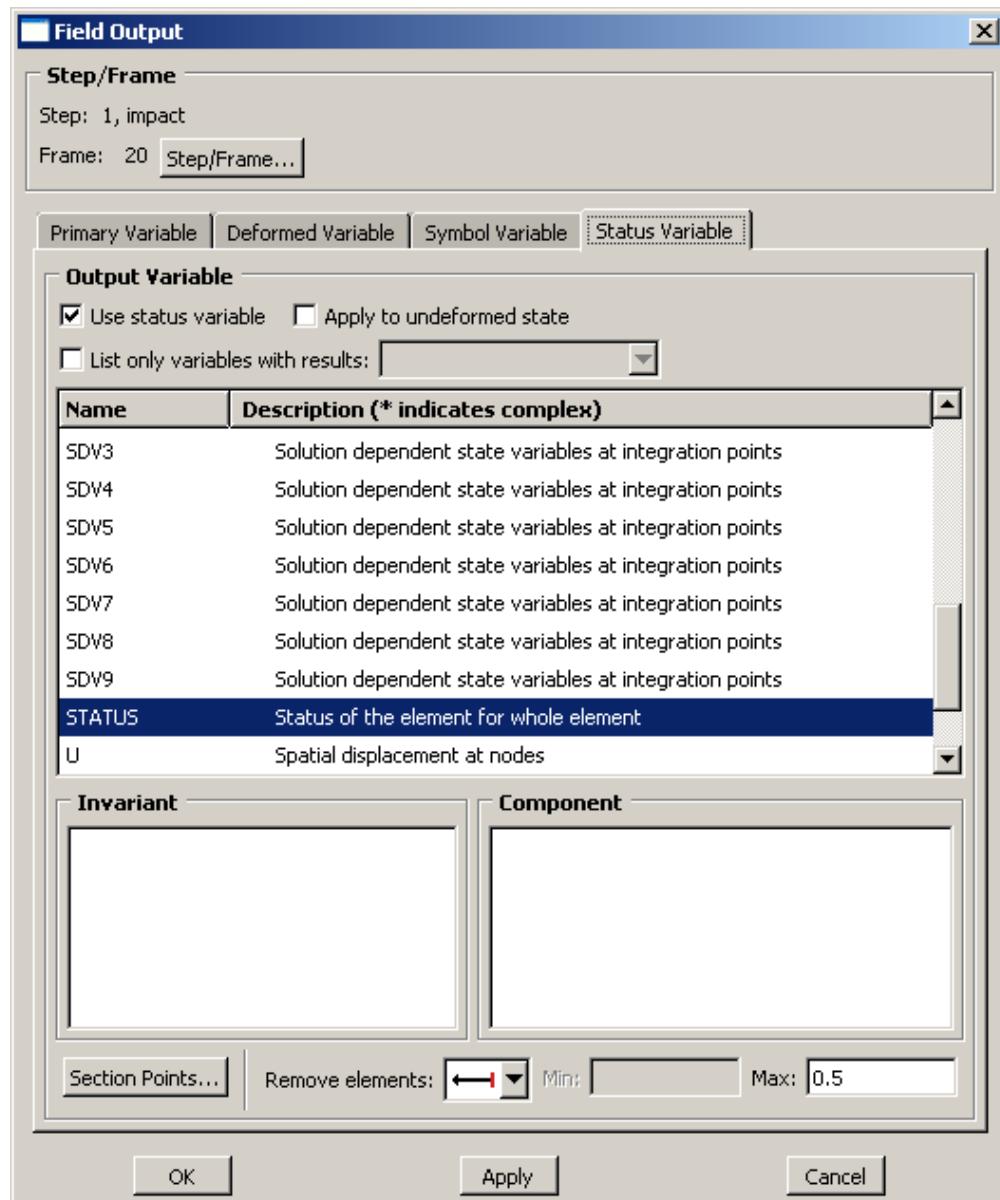


Figure W3–8. Status Variable options

3. Use the **Create Display Group** tool  in the toolbar to create a display group that includes a single layer of the composite plate.

In the **Create Display Group** dialog box, choose the item **Elements** and the method **Element sets**, as shown in Figure W3–9.

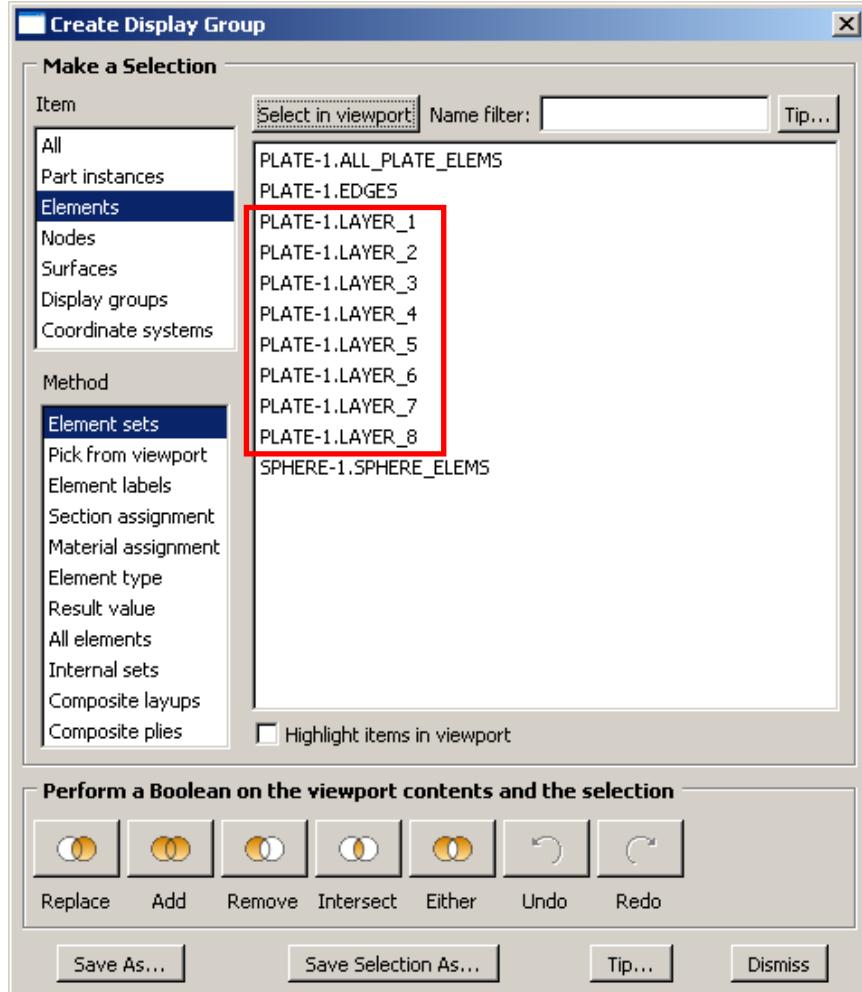


Figure W3–9. **Create Display Group** dialog box

4. Select the element set **PLATE-1.LAYER_1** and click **Replace** .
- Note that the contour plot displays the stress contour in the first layer.
5. Click the **Plot Material Orientations on Deformed Shape** tool  in the toolbox.
- The material orientation plot appears and displays the material orientations of the first layer.
- Using a similar procedure, display the stress contour plots and material orientations of the other layers.
6. Click **Dismiss** to close the **Create Display Group** dialog box.
 7. Click the **Replace All** tool  in the toolbar to display the whole model.
 8. Plot the deformed model shape, as shown in Figure W3–10.

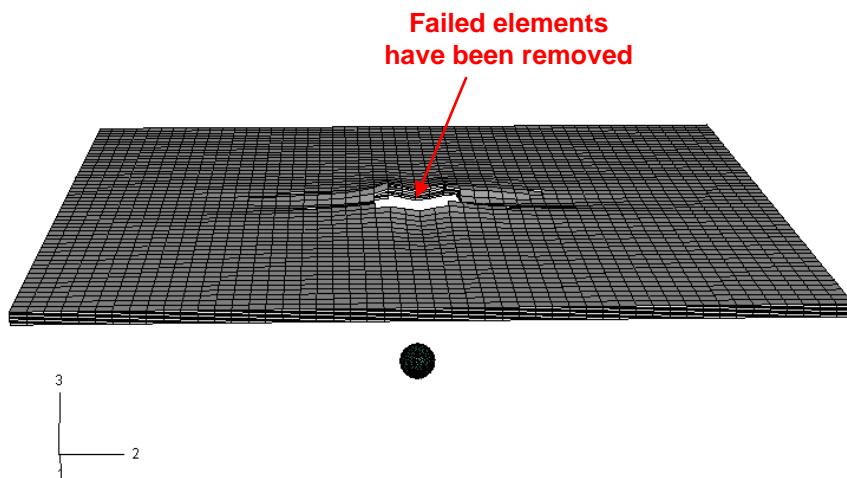


Figure W3–10. Deformed shape with failed elements removed.

9. Plot the velocity history of the rigid ball to evaluate the residual velocity of the projectile.
 - a. In the Results Tree, expand the **History Output** container underneath the output database named **plate_impact.odb**.
 - b. Double-click **Spatial velocity: V3 PI: SPHERE-1 Node ... in NSET SPHERE_RP**. An *X-Y* plot appears and displays the velocity of the projectile, as shown in Figure W3–11.

The plot clearly shows that the projectile decelerates during impact and then maintains a constant velocity after perforating the plate. If this were a model of debris impact on an aircraft panel or a ballistic impact on body armor, the design under consideration would be inadequate to keep the fragment from penetrating.

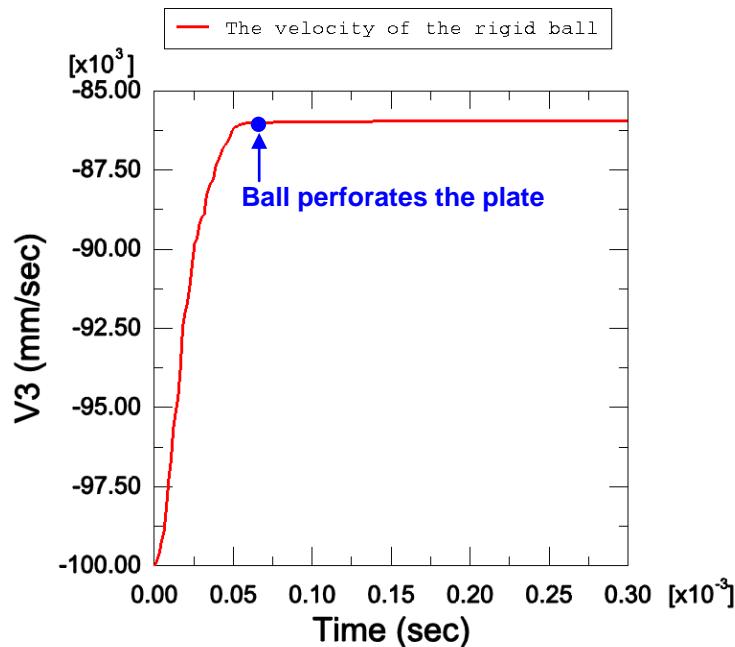


Figure W3–11. The velocity history of the projectile.

Re-running the job without mass scaling

You may re-run the job without mass scaling to obtain results that are more physically meaningful. Note that this analysis may take approximately 2 hours to complete. If you are interested in running it, it is recommended you run the job outside of the course. The following discusses the results obtained without mass scaling.

1. The deformed model shape without the failed elements is shown in Figure W3–12. The plot shows that the deformation and failure of the composite plate are completely different from those with mass scaling (see Figure W3–10).

Unlike the model with mass scaling, the projectile does not fully perforate the plate; instead it partially perforates the plate then rebounds (failure is inside the plate).

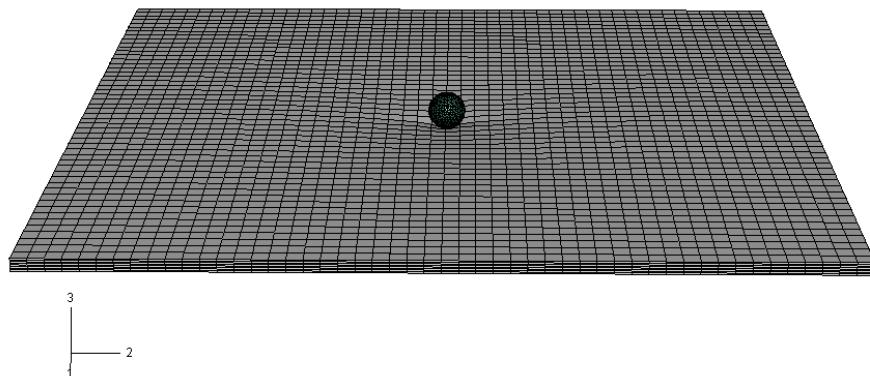


Figure W3–12. Deformed shape of the model without mass scaling.

2. The velocity history of the rigid ball is shown in Figure W3–13.

The plot shows that the projectile decelerates during impact until its velocity becomes zero, and then accelerates to a constant positive value after rebounding from the plate. Therefore we see the dangers of using mass scaling in a model where kinetic effects are significant. When modeled correctly, this plate layup indicates an ability to withstand the impact of the steel ball. Some damage to the plate occurs, but penetration is prevented.

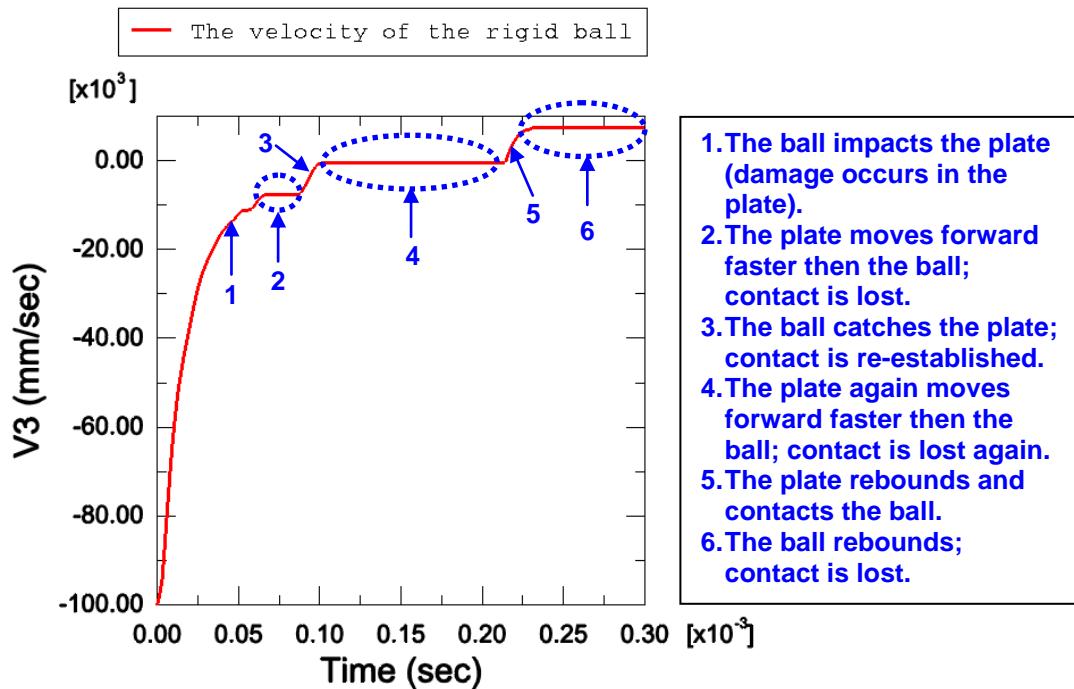


Figure W3–13. The velocity history of the projectile without mass scaling.

Note: A script that creates the complete model described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions outlined here or if you wish to check your work. The script is named `ws_composite_impact_answer.py` and is available using the Abaqus fetch utility.

Notes

Notes



Workshop 1

Laminated Composite Panel

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the “Interactive” version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define the material properties of a fiber-matrix layer.
- Define a composite layup with different layer orientations.
- Perform prebuckling, eigenvalue buckling, and postbuckling analyses.
- Use Abaqus/Viewer to view the material orientations and create contour plots on different plies, and to view a ply stack plot and an envelope plot.

Introduction

In recent years, fiber-reinforced composite laminated shell structures have been widely used in the aerospace, marine, automobile, and other engineering industries for lightweight applications. The buckling and post-buckling analysis of composite panels is of primary importance to an optimized design for a specific composite light-weight-component as well as the entire structure. The load at which buckling occurs and whether the subsequent deformation is catastrophic or well-behaved are critical design issues and depend heavily on the lay-up of the composite panel material.

In this workshop you will define the material properties of a fiber-matrix layer and composite layup with different ply orientations and study the buckling of a composite structure. Figure W1–1 shows the structure analyzed in this workshop. It is a cylindrical composite panel with a circular hole. The panel is fully clamped on the bottom edge, free to move axially along the top edge, and simply supported along its vertical edges. Three analyses will be performed. The first is a linear (prebuckling) analysis in which the panel is subjected to a uniform end shortening of 0.0316 in. The second analysis consists of an eigenvalue extraction of the first five buckling modes. The total axial force (distributed along the midsection) is used in this analysis, instead of the shortening, since the buckling load can be easily calculated using the axial force (buckling load = eigenvalue \times axial

load). Finally, a nonlinear load-deflection analysis is performed using the modified Riks algorithm. The postbuckling behavior is induced with an initial imperfection based on the extracted buckling modes.

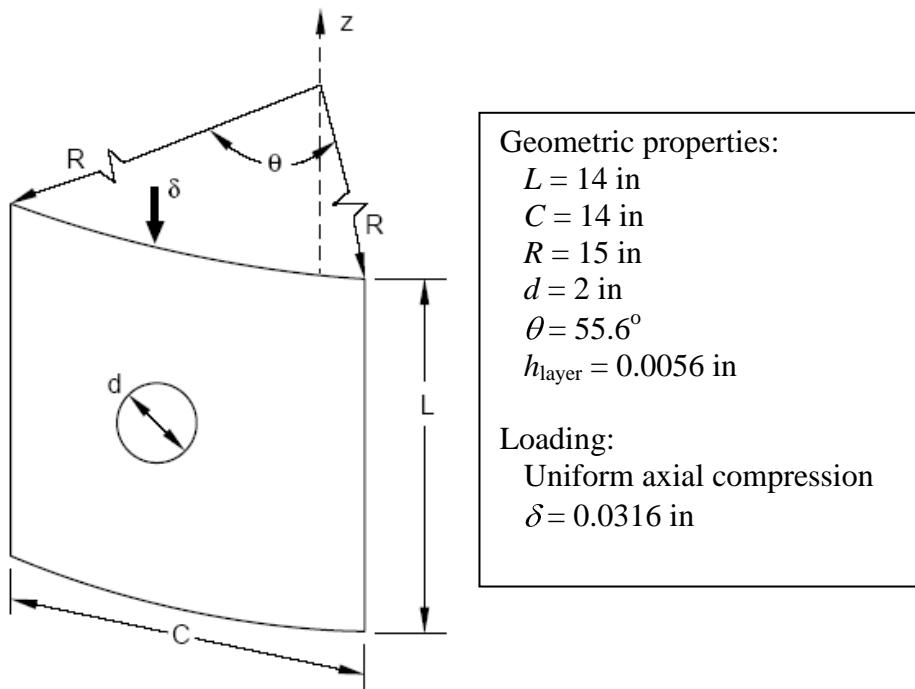


Figure W1–1. Geometry of the cylindrical panel with hole.

Figure W1–2 shows the details of the model, including the mesh and boundary conditions. Note that the symmetry conditions indicated in the figure represent real physical boundary conditions and are not intended to imply a mirrored structure. Thus, the comments regarding symmetry conditions in Lecture 3 do not apply in this particular case.

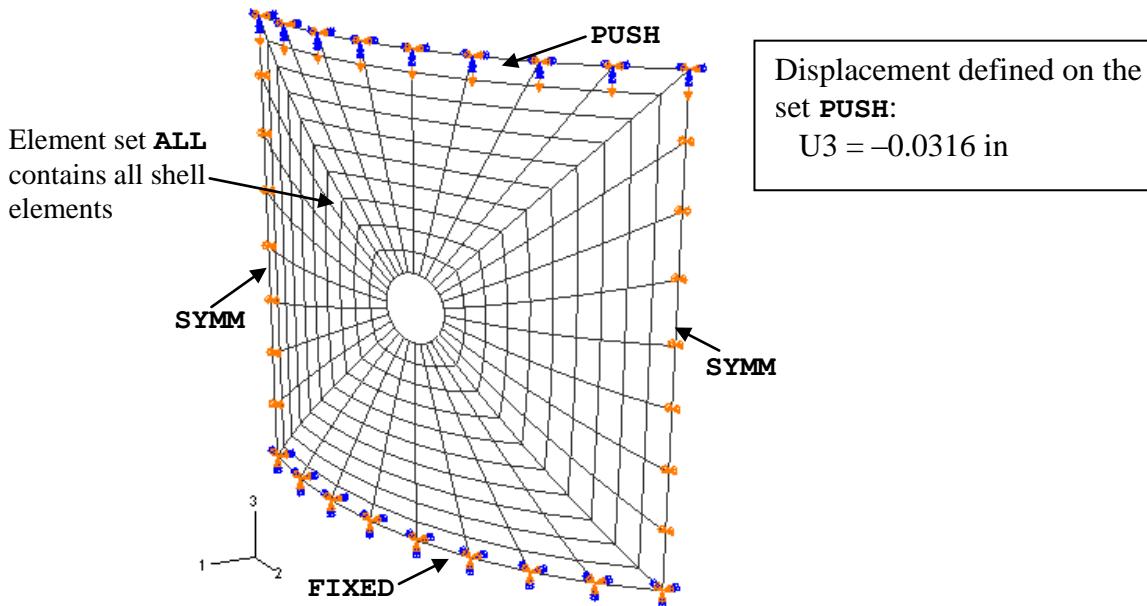


Figure W1–2. Model geometry and boundary conditions

Part 1: Prebuckling analysis

Enter the working directory for this workshop:

```
..../composites/keywords/buckle
```

The file **w_laminpanel_prebuckle.inp** contains an incomplete model of the laminated composite panel in which the mesh, boundary conditions, and analysis procedure (general, static) are already defined. The shell element type S4R is used in the model.

You will define the multilayered, anisotropic, laminated shell section for the composite panel. The shell consists of 16 plies of unidirectional graphite fibers in an epoxy resin. The plies are arranged in the symmetric stacking sequence $[45, -45, 90, 0]_S$ degrees. Each ply is 0.0056 in thick. The nominal orthotropic elastic material properties of the lamina are

$$\begin{aligned} E_{11} &= 19.6 \times 10^6 \text{ lb/in}^2, \\ E_{22} &= 1.89 \times 10^6 \text{ lb/in}^2, \\ G_{12} = G_{13} &= 0.93 \times 10^6 \text{ lb/in}^2, \\ G_{23} &= 0.63 \times 10^6 \text{ lb/in}^2, \text{ and} \\ \nu_{12} &= 0.38 \end{aligned}$$

where the 1-direction is along the fibers, the 2-direction is transverse to the fibers in the surface of the lamina, and the 3-direction is normal to the lamina.

Completing the prebuckling model

To complete the model, do the following:

1. Use your text editor to review the supplied workshop model contained in the file **w_laminpanel_prebuckle.inp**.

2. Define the orthotropic elastic behavior of the lamina with the material properties given above using the *MATERIAL option. The required option is:

```
*MATERIAL, NAME=LAMINA
*ELASTIC, TYPE=LAMINA
19.6E6, 1.89E6, 0.38, 0.93E6, 0.93E6, 0.63E6
```

3. Define a cylindrical coordinate system with the local *z*-axis along the global Z-axis. This local coordinate system will be used to define the material orientation. The required option is:

```
*ORIENTATION, SYSTEM=CYLINDRICAL, NAME=SECORI
0., 0., 0., 0., 0., 1.
1, 0.
```

4. Since the material behavior is linear elastic, a composite shell general section will be defined to reduce the computational cost of the job. The section uses the local material orientation **SECORI** and consists of the 16 layers noted above. Each layer is assigned the material **LAMINA**. Use the SYMMETRIC parameter on the *SHELL GENERAL SECTION option to activate simplified modeling of symmetric composites. The following option defines the shell general section:

```
*SHELL GENERAL SECTION, ELSET=ALL, COMPOSITE,
ORIENTATION=SECORI, SYMMETRIC
0.0056, , LAMINA, 45., PLY-1
0.0056, , LAMINA, -45., PLY-2
0.0056, , LAMINA, 90., PLY-3
0.0056, , LAMINA, 0., PLY-4
0.0056, , LAMINA, 0., PLY-5
0.0056, , LAMINA, 90., PLY-6
0.0056, , LAMINA, -45., PLY-7
0.0056, , LAMINA, 45., PLY-8
```

PLY-1, **PLY-2**, etc. are names for each ply and are available in ODB for easy tracking in postprocessing operations.

Note: If no ply name is defined in the input file, Abaqus will assign internal ply names, i.e. *layer_1*, *layer_2*, etc., for each ply according the order of the data line (e.g., *layer_1* represents the first ply defined in the data line).

5. Specify the section points at which you would like output. Note that even though you use the *SHELL GENERAL SECTION option, Abaqus uses section points for output purposes. You can view the layer orientation and contour the specified layer in Abaqus/Viewer. By default, each layer has 3 section points. In this workshop, you will request output at the middle section point for each ply. The required option is:

```
*ELEMENT OUTPUT, DIRECTION=YES, VARIABLE=PRESELECT  
2,5,8,11,14,17,20,23,26,29,32,35,38,41,44,47
```

Note: A maximum number of 16 section points can be specified. Repeat *ELEMENT OUTPUT as often as needed if output at additional points is required.

6. Save the modified input file, and run the analysis by entering the following command at the prompt:

```
abaqus job=w_laminpanel_prebuckle
```

Postprocessing the prebuckling analysis using Abaqus/Viewer

When the analysis is complete, use the following procedure to create contour plots and view material orientations of different layers using Abaqus/Viewer:

1. Start Abaqus/Viewer and open the file `w_laminpanel_prebuckle.odb`:
- ```
abaqus viewer odb=w_laminpanel_prebuckle.odb
```
2. From the main menu bar, select **Result→Section Points**. The **Section Points** dialog box appears. Choose the **Plies** selection method. A list of all plies in the composite layup appears in the **Plies** field, as shown in Figure W1–3. You can select any individual ply to display output.

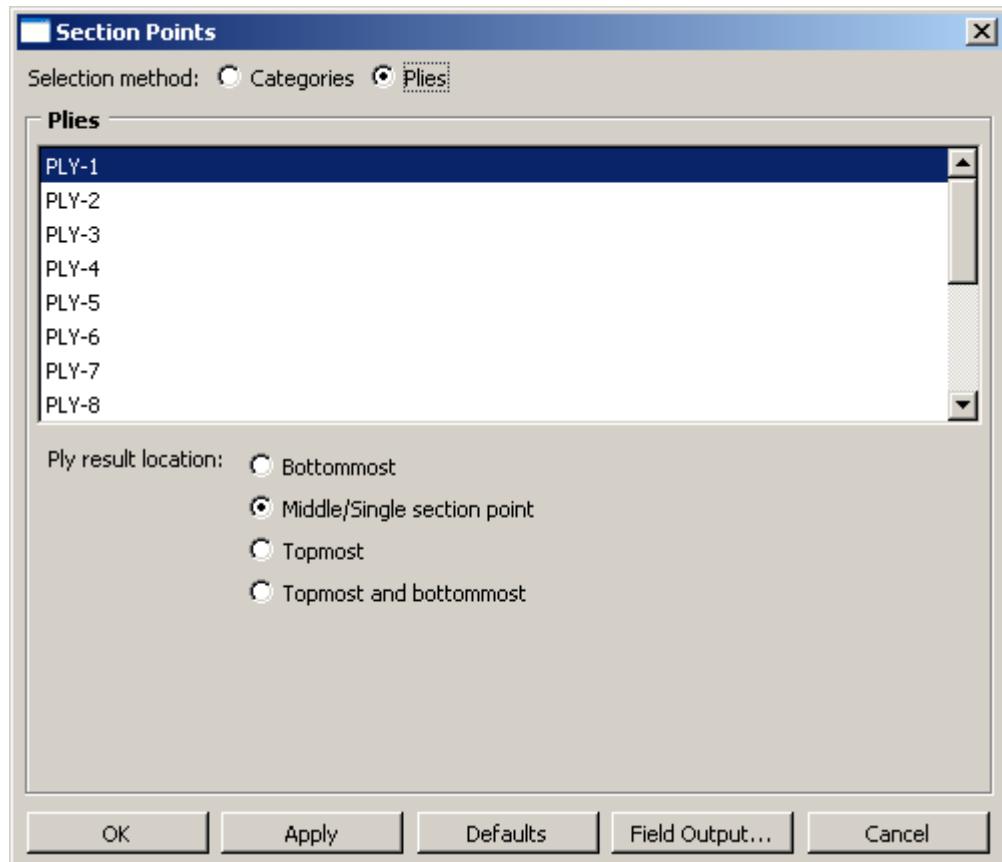


Figure W1–3. **Section Points** dialog box.

3. Accept the default selections (**PLY-1** and the **Middle** result location) and click **Apply**.
4. Click the **Plot Contours on Deformed Shape** tool in the toolbox. A contour plot of Mises stress at the middle of **PLY-1** appears.

5. Contour the stress component along the fiber direction (variable S11) of **PLY-1**. Use the **Field Output** dialog box to change the displaying variable:

- From the list of variable types on the left side of the **Field Output** toolbar, select **Primary** if it is not already selected.
- From the list of available output variables in the center of the toolbar, select output variable **S** if it is not already selected.
- From the list of available components and invariants on the right side of the **Field Output** toolbar, select **S11**.

A contour plot of S11 at the middle of **PLY-1** appears.

6. Click the **Plot Material Orientations on Deformed Shape** tool  in the toolbox.

A plot of the material orientations at the middle of **PLY-1** appears.

Note that you can also view the output and material orientations at the bottom, top, or top and bottom locations simultaneously by selecting the corresponding available ply result locations in the **Section Points** dialog box.

7. Create a ply stack plot from a probed composite layup section. The combination of the ply stack plot and the ply-based material orientation plot provides a complete view of the ply orientation.

- Click the **Query information** tool  in the tool bar.
- In the **Query** dialog box, select **Ply stack plot** from the list of **Visualization Module Queries**.

Note that a new viewport is created and tiled vertically with respect to the original viewport.

- The ply stack plot in Abaqus/Viewer is section-based. In the viewport displaying the shell, click anywhere on the mesh to select the entire section, as shown in Figure W1–4 (left).
- The ply stack plot from the probed section appears in the new viewport (see Figure W1–4 (right)).

**Note:** The ply stack plot displays all plies in the composite layup, including those generated using the symmetry option. These plies contain the prefix **Sym\_** in their names to indicate that they are repeated plies. The staircase appearance in the ply stack plot has no physical meaning; it is simply a graphical representation that allows you to see the number of plies in the layup and, for example, the relative thickness of a ply and the orientation of its fibers.

From Figure W1–4 (right) you can see that the ply **PLY-1** is the bottom ply in the composite layup, and the fiber in this ply is oriented along 45° with respect to 1-axis of the layup orientation.

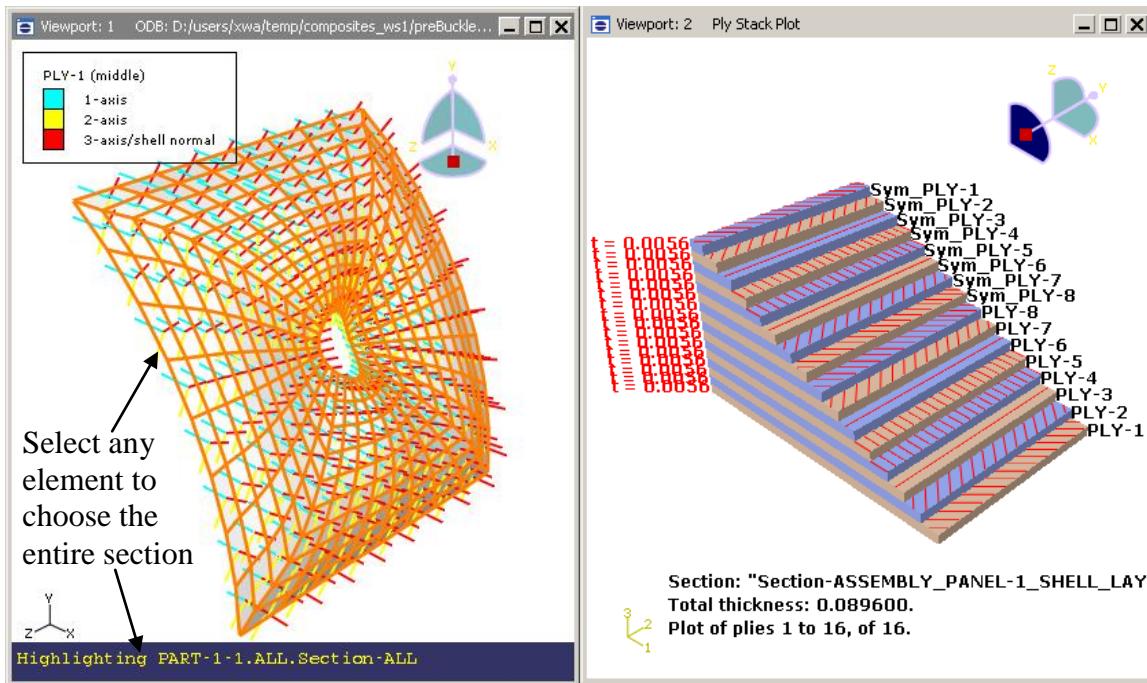


Figure W1-4. Ply stack plot of a probed section

8. Make current the viewport displaying the material orientation plot, if necessary. In the **Section Points** dialog box, select the ply **PLY-2** and click **Apply**.  
The material orientation plot now displays the material orientations at the middle of **PLY-2**. Again, from the ply stack plot, you can see that the ply **PLY-2** is the second ply from the bottom in the composite layup, and the fiber in this ply is oriented along  $-45^\circ$  with respect to 1-axis of the layup orientation.
9. Click  in the toolbox.  
A contour plot of S11 at the middle of **PLY-2** appears.  
Note that the S11 contour plot changes with the layer orientation. You can also view the output on other plies.
10. Create an *envelope* plot to identify the maximum values of S11 across all the plies of the layup.
  - a. In the **Section Points** dialog box, choose **Categories** as the selection method and then **Envelope** as the active location.
  - b. Select **Max value** as the criterion and **Integration point** as the position.
  - c. Click **OK** to plot the stress envelope of the shell.
11. Show the critical plies on the envelope plot.
  - a. Delete the viewport displaying ply stack plot.
  - b. Create a new viewport and tile it vertically with the previous one (**Viewport**→**Tile Vertically**).

- c. Change the contour variable to S11 to create the same envelope plot as described in the previous step.

- d. Click the **Contour Options** tool  in the tool box.
- e. In the **Contour Plot Options** dialog box that appears, select the **Other**-tabbed page and toggle on **Show labels of plies that match criteria**. Click **Apply**.

The name of the critical ply at each element appears on the envelope plot.

- f. Toggle off **Show labels of plies that match criteria**; instead toggle on **Color by plies that match criteria, instead of result value**.

- g. Click **OK** to display quilt plot of critical plies.

Note that, using a combination of these plots created in steps 10 and 11, you can determine both the value of S11 in the critical ply and the location of the critical ply in the layup.

## Part 2: Eigenvalue buckling analysis

You will perform an eigenvalue buckling analysis to determine the buckling eigenmodes of the structure. These buckling eigenmodes will be used to introduce imperfections into the geometry of the panel to ensure a physically correct deformed shape in the subsequent postbuckling analysis.

To ensure a physically meaningful simulation, the buckling analysis must be performed on a model that is similar to the postbuckling model. Unlike the previous analysis, the total axial force (distributed along the midsection of the top edge) will be used since it will be easier to determine the buckling load (recall the buckling load = eigenvalue  $\times$  axial load).

The mesh, material data, and loading for the eigenvalue buckling analysis are already defined and can be found in the file `w_laminpanel_buckle.inp`.

### Completing the eigenvalue buckling model

To complete the model, do the following:

1. Use your text editor to review the supplied workshop model contained in the file `w_laminpanel_buckle.inp`. Note that a compressive axial force with an amplitude of 1000 lbf is distributed along the midsection of the top edge. Constraint equations are defined between the center node (node 2041, as shown in Figure W1–5) and the remaining nodes on the top edge for degree of freedom 3.

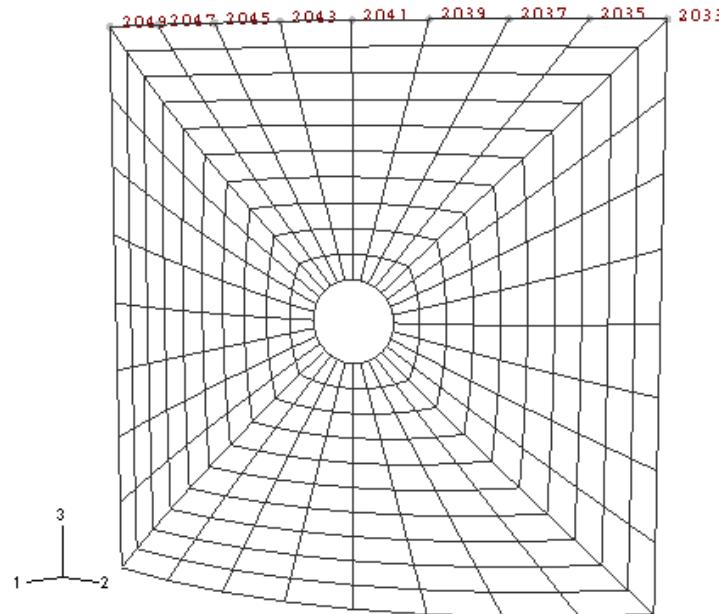


Figure W1–5. Nodes used in the constraint equations.

- Define the analysis step and procedure. The following defines the procedure for an eigenvalue buckling analysis:

```
*STEP, PERTURBATION
*BUCKLE
5, , 50, 20
```

- Save the modified file, and run the analysis using the following command:

```
abaqus job=w_laminpanel_buckle
```

### Postprocessing the buckling analysis using Abaqus/Viewer

When the analysis is complete, use the following procedure to view the eigenmodes from the buckling analysis in Abaqus/Viewer:

- Open **w\_laminpanel\_buckle.odb** in Abaqus/Viewer.
- Plot the deformed model shape.

The deformed shape for the first eigenmode will be displayed in the viewport. The corresponding eigenvalue will be reported in the state block. Adjust your view, if necessary, to see the deformed configuration more clearly.

- View the deformed shapes of the other buckling modes using the frame selector  or the frame control buttons  in the context bar above the viewport.

Figure W1–6 shows the first five eigenmodes of the composite panel.

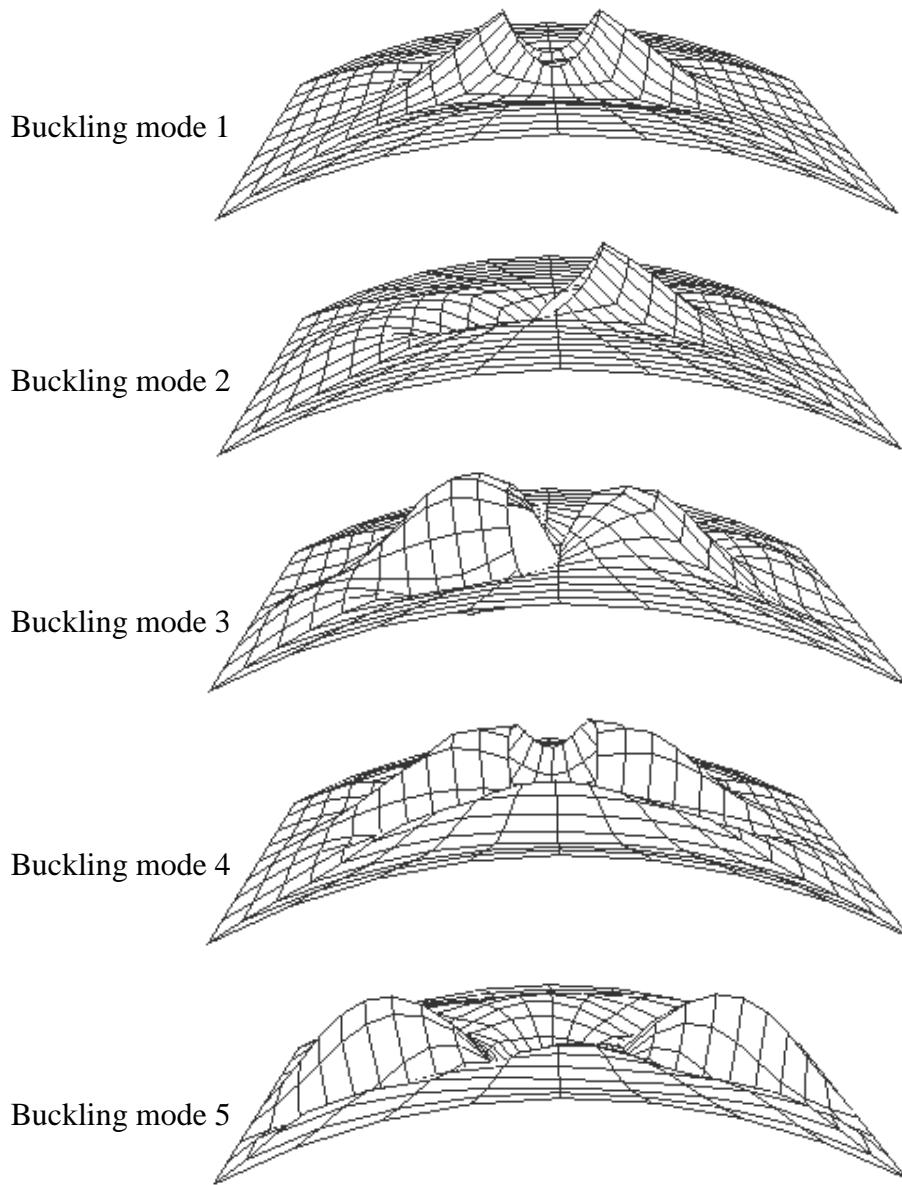


Figure W1–6. Buckling modes.

The primary result of interest in the buckling analysis is the predicted eigenvalues for each of these modes. The eigenvalue, when multiplied by the nominal load value used for the plate, provides a prediction of the buckling load value for each mode. From the main menu bar, select **Result→Step/Frame** to see the predicted eigenvalues. Click **Cancel** when you are done reviewing these results.

The fiber lay-up will have a significant effect on the value at which the plate buckles. To show this, results for a plate in which all fibers are aligned with the load direction (i.e., all plies in this model are along 90° with respect to the 1-axis of the <layup> orientation) has been analyzed, and a comparison of these results are shown in the table below. A significant reduction in load carrying capability is predicted for the plate with all fibers oriented with the load direction.

Table W1–1. Comparison of buckling load prediction for two plate lay-ups.

|               | <b>Plate Buckling Loads (psi)</b> |                |
|---------------|-----------------------------------|----------------|
|               | $[(45, -45, 90, 0)_S]_S$          | $[90_{(8)}]_S$ |
| <b>Mode 1</b> | 25109                             | 18294          |
| <b>Mode 2</b> | 25899                             | 18859          |
| <b>Mode 3</b> | 27349                             | 21145          |
| <b>Mode 4</b> | 34039                             | 24536          |
| <b>Mode 5</b> | 40338                             | 25522          |

## Part 3: Postbuckling analysis

You will now perform a nonlinear load-deflection analysis to predict the postbuckling behavior. You will use the \*IMPERFECTION option to specify that the modes from the buckling analysis will be used to seed an initial imperfection in the postbuckling analysis model.

The mesh, material, and loading for the postbuckling model are already defined and can be found in the file `w_laminpanel_postbuckle.inp`.

### Completing the postbuckling model

To complete the model, do the following:

1. Use your text editor to review the supplied workshop model contained in the file `w_laminpanel_buckle.inp`. Note that the amplitude of the compressive axial force has been increased to 10000.
2. Define the analysis step and procedure. The following defines the procedure definition for a static, Riks analysis (the parameters are selected to ensure that the job runs long enough to show the collapse):

```
*STEP, NLGEOM=YES, INC=15
*STATIC, RIKS
1., 1., , , , 2041, 3, -0.08
```

3. Use the \*IMPERFECTION option to introduce geometric imperfections into the postbuckling model. The imperfections allow the buckling to occur in a smooth manner. The imperfections included in this model are based on the first four eigenmodes extracted from the eigenvalue buckling analysis; the magnitude of the imperfection is scaled so that the maximum imperfection is 10% of the shell thickness.

```
*IMPERFECTION, FILE=w_laminpanel_buckle, STEP=1
1, 0.001
2, 0.0005
3, 0.00025
4, 0.00025
```

Note that the FILE parameter on the \*IMPERFECTION option is set to `w_laminpanel_buckle`, indicating that the results from this file will be used to perturb the panel's geometry.

4. Save the modified file, and run the analysis using the following command:

```
abaqus job=w_laminpanel_postbuckle
```

## Results visualization using Abaqus/Viewer

Follow the procedure listed below to view the results from the postbuckling analysis.

1. Open **w\_laminpanel\_postbuckle.odb** in Abaqus/Viewer.
2. Plot the deformed model shape.

Figure W1–7 shows the deformed configuration at the end of simulation.

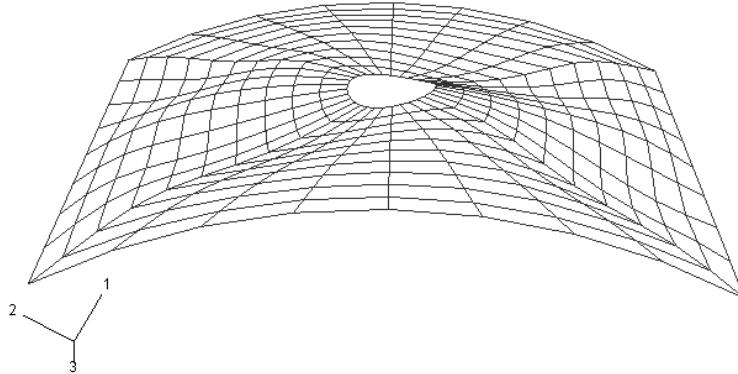


Figure W1–7. Deformed mesh.

3. Animate the deformation to see the behavior of the plate after buckling. This animation will help you follow the non-linear deformation path of the postbuckling field.
4. One of the primary considerations for the behavior of composite panels is the residual stiffness after buckling. To make this assessment complete the following steps.
  - a. In the Results Tree, expand the **History Output** branch for the output database file named **w\_laminpanel\_postbuckle.odb**.
  - b. From the list of available output, choose **Load proportionality factor: LPF for Whole Model** and click mouse button 3. From the menu that appears, select **Save As**. Name the data **LPF**.
  - c. Similarly, save the curve for the U3 displacement (**Spatial displacement: U3 at Node 2041 in NSET MASTER**) and name it **U3**.
  - d. In the Results Tree, double-click **XYData**.
  - e. Select the source **Operate on XY data** and click **Continue**.
  - f. From the **Operators** menu on the right, select **combine(X, X)**.
  - g. In the **XY Data** menu on the left, select **U3** and click **Add to Expression**.
  - h. In the **XY Data** menu, select **LPF** and click **Add to Expression**.

- i. Insert a minus (“–”) sign in front of **U3** in the top of the dialog box, so that the final expression reads **combine( –“U3”, “LPF” )**. This is only done to make the subsequent plot easier to read.
- j. Click **Plot Expression** and then click **Cancel**.

Figure W1–8 shows the load-displacement curve of the plate buckling analysis. This plot shows the variation in plate load with displacement of the node at which the load is applied. The initial slope of the curve is indicative of the initial stiffness of the plate. Buckling occurs at approximately 25000 psi. Finally a new load-displacement path is followed. The final slope of this path is shallower than the initial slope at the start of loading, and indicates the post-buckling reduction in stiffness.

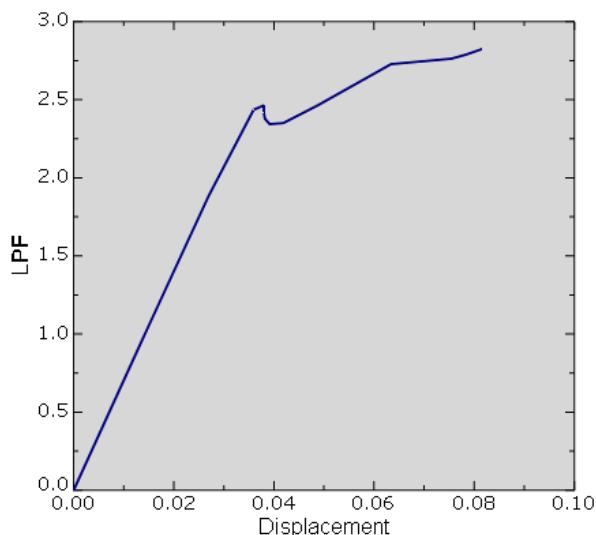


Figure W1–8. Load-displacement curve during buckling.

**Note:** Complete input files are available for your convenience. You may consult these files if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input files are named

`w_laminpanel_prebuckle_complete.inp`  
`w_laminpanel_buckle_complete.inp`  
`w_laminpanel_postbuckle_complete.inp`

and are available using the Abaqus fetch utility.

# Notes

# Notes



## Workshop 3

### Perforation of a Composite Plate

#### Keywords Version

**Note:** This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the “Interactive” version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

#### Goals

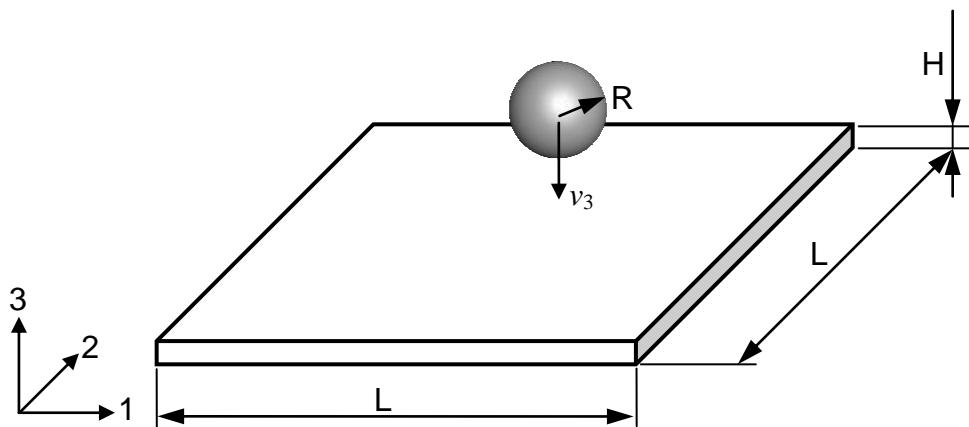
- Define composite material properties.
- Define a composite layup with different layer orientations.
- Perform an impact analysis.
- Use Abaqus/Viewer to view the material orientations, create contour plots on different layers, and investigate composite damage and failure.

#### Introduction

Because composite materials offer high directional stiffness at significant weight savings when compared to metals, they continue to find increased use in many industries. These include civilian and military vehicles, marine craft, and aircraft. In addition, composite materials such as Kevlar are used extensively in the design of protective armor. Whether attempting to predict the ballistic limit velocity of body armor or the resistance of an aircraft panel to impact from runway debris, it is critical to accurately predict damage due to impact on composite materials.

In this workshop, you will analyze the normal impact of a rigid steel ball onto a composite plate at a velocity of 1e5 mm/sec, using Abaqus/Explicit. A schematic of the model is shown in Figure W3–1. You will define the composite material properties and layup, study the ability of the general contact algorithm to model surface erosion on multiple contacting bodies during high-speed impact, and investigate composite damage and failure. Figure W3–2 shows the details of the model, including the geometry and boundary conditions. The plate is fully clamped along its edges.

The composite plate is made of 8 layers of unidirectional carbon fibers in an epoxy resin in a { 0/90/ $\pm 45$ / $\mp 45$ /90/0 } layup. Each layer has a thickness of 0.2 mm. To simulate impact on composite materials, damage and failure modeling is critical since it allows the



Geometric properties:

$$L = 100 \text{ mm}$$

$$H = 1.6 \text{ mm}$$

$$R = 2.5 \text{ mm}$$

$$h_{layer} = 0.2 \text{ mm}$$

The velocity of the ball:

$$v_3 = -1e5 \text{ mm/sec}$$

Figure W3–1. Impact of a rigid ball onto a flat composite plate.

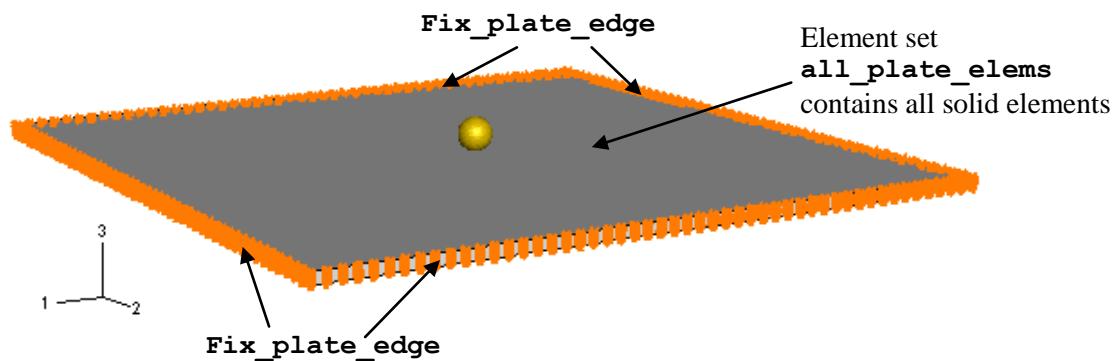


Figure W3–2. Model geometry and boundary conditions

ball to perforate the plate. In this workshop, a user-defined material model is provided as a VUMAT. The material model is based on Hashin's failure criteria for unidirectional fiber composites (Hashin, 1980) and considers four possible failure modes: matrix compression and tension, fiber tension, and fiber compression (fiber buckling). The source code is in **uniFiber.for**; if it is not already provided to you, you can download this user subroutine from SIMULIA Answer 3123.

Typical linear elastic properties are used for the carbon fiber composite (CFC) material:

$$\begin{aligned}E_{11} &= 1.64e5 \text{ MPa}, \\E_{22} = E_{33} &= 1.2e4 \text{ MPa}, \\G_{12} = G_{13} &= 4500 \text{ MPa}, \\G_{23} &= 2500 \text{ MPa}, \\\nu_{12} = \nu_{13} &= 0.32, \text{ and} \\\nu_{23} &= 0.45\end{aligned}$$

The strength properties are:

$$\begin{aligned}\text{Tensile failure stress in fiber direction: } X_{1t} &= 2724 \text{ MPa}, \\\text{Compressive failure stress in fiber direction: } X_{1c} &= 111 \text{ MPa}, \\\text{Tensile failure stress in direction 2: } X_{2t} &= 50 \text{ MPa}, \\\text{Compressive failure stress in direction 2: } X_{2c} &= 1690 \text{ MPa}, \\\text{Tensile failure stress in direction 3: } X_{3t} &= 290 \text{ MPa}, \\\text{Compressive failure stress in direction 3: } X_{3c} &= 290 \text{ MPa}, \\\text{Shear strength in 12 plane: } S_{12} &= 120 \text{ MPa}, \\\text{Shear strength in 13 plane: } S_{13} &= 137 \text{ MPa}, \\\text{Shear strength in 23 plane: } S_{23} &= 90 \text{ MPa}\end{aligned}$$

where the 1-direction is along the fibers, the 2-direction is transverse to the fibers in the surface of the ply, and the 3-direction is normal to the ply.

## Preliminaries

Enter the working directory for this workshop:

```
.. /composites/keywords/impact
```

The file **w\_plate\_impact.inp** contains an incomplete model in which the mesh (contained in the file **w\_plate\_impact\_geometry.inp**), boundary conditions, initial ball velocity, material properties of the ball, and analysis procedure (dynamic, explicit) are already defined. The solid element type C3D8R is used for the composite plate; the eight-layered structure is also defined with one set for each layer, as shown in Figure W3-3.

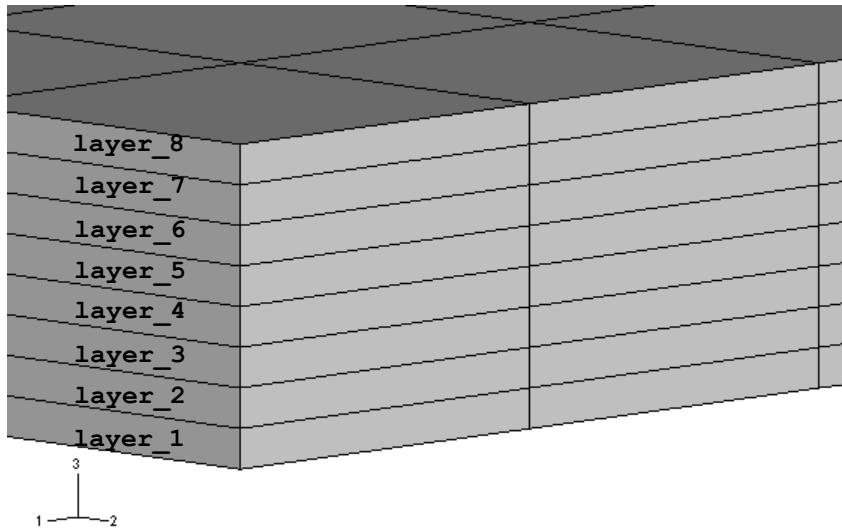


Figure W3–3. Layer definitions of the plate

Note that a mass scaling factor of 160 is used to speed up the analysis. In general, mass scaling is not suitable for impact problems since the dynamic response is of primary importance; however, in this case, it enables the job to run in a reasonable time frame, consistent with the requirements of a training exercise. Note that the results obtained in this workshop may not be physically realistic due to mass scaling. In order to obtain results that are more physically meaningful, you may re-run this job without mass scaling at a later time.

You will add the necessary data to complete the model.

## Defining material properties and orientations

You will define the material definition and layup of the composite plate.

1. Use your text editor to review the supplied workshop model contained in the file **w\_plate\_impact.inp**.
2. Define the material behavior of the composite plate, which includes the density, elastic and strength properties, and solution-dependent variables, using the \*MATERIAL option. The VUMAT user subroutine requires 17 solution-dependent state variables and variable 5 is the controlling parameter to control element deletion. The required options are:

```
*MATERIAL, NAME=CFC
*DENSITY
1.0E-9
*DEPVAR, DELETE=5
17,
*USER MATERIAL, CONSTANTS=27
1.64E5, 1.2E4, 1.2E4, 0.32, 0.32, 0.45, 4500., 4500.
2.5E3, 1.0E-9
2724., 111., 50., 1690., 290., 90.
120., 137., 90.
```

3. Define four rectangular datum coordinate systems (CSYSs) representing four material orientations (noted earlier) and then assign them to the appropriate layers of the composite plate. They have different local 1-axis directions but are all centered at the origin with their local 3-axes along the global Z-direction:

| Datum CSYS                      | Local 1-axis                                                  |
|---------------------------------|---------------------------------------------------------------|
| <b>zero_degree</b>              | along the global X-direction                                  |
| <b>ninety_degrees</b>           | along the global Y-direction                                  |
| <b>plus_forty_five_degrees</b>  | at $+45^\circ$ degrees with respect to the global X-direction |
| <b>minus_forty_five_degrees</b> | at $-45^\circ$ degrees with respect to the global X-direction |

The required options are:

```
*ORIENTATION, NAME=zero_degree
1., 0., 0., 0., 1., 0.
3, 0.
*ORIENTATION, NAME=ninety_degrees
0., 1., 0., -1., 0., 0.
3, 0.
*ORIENTATION, NAME=plus_forty_five_degrees
1., 1., 0., -1., 1., 0.
3, 0.
*ORIENTATION, NAME=minus_forty_five_degrees
1., -1., 0., 1., 1., 0.
3, 0.
```

4. Assign the material definition and orientation to each layer through the solid section definitions. The required options are:

```
*SOLID SECTION, ELSET=layer_1, ORIENTATION=zero_degree,
MATERIAL=CFC

*SOLID SECTION, ELSET=layer_2, ORIENTATION=ninety_degrees,
MATERIAL=CFC

*SOLID SECTION, ELSET=layer_3,
ORIENTATION=plus_forty_five_degrees, MATERIAL=CFC

*SOLID SECTION, ELSET=layer_4,
ORIENTATION=minus_forty_five_degrees, MATERIAL=CFC

*SOLID SECTION, ELSET=layer_5,
ORIENTATION=minus_forty_five_degrees, MATERIAL=CFC

*SOLID SECTION, ELSET=layer_6,
ORIENTATION=plus_forty_five_degrees, MATERIAL=CFC

*SOLID SECTION, ELSET=layer_7, ORIENTATION=ninety_degrees,
MATERIAL=CFC

*SOLID SECTION, ELSET=layer_8, ORIENTATION=zero_degree,
MATERIAL=CFC
```

## Defining damage output request

The preselected default output does not include the damage output variables STATUS and SDV. STATUS is used to identify elements which have failed, and SDVi are the solution-dependent variables which include tensile and compressive damage in the 1- and 2-directions. To visualize the damage and failure in Abaqus/Viewer, you will write additional field output to the output database file.

1. Locate the \*OUTPUT, FIELD, VARIABLE=PRESELECT option. Add the following sub-option:

```
*ELEMENT OUTPUT
SDV, STATUS
```

## Defining general contact

General contact will be defined for this impact analysis. Since erosion will occur during the analysis (failure of elements will occur as they become fully damaged), contact of the rigid steel ball must be allowed not only with the external surfaces of the plate but also with its internal surfaces. By default, general contact in Abaqus/Explicit allows for contact between all exterior surfaces in a model including a surface with itself. To include interior surfaces in the contact domain, you must define a surface that includes the interior faces of the plate.

1. Define a surface including all interior faces of the composite plate using the \*SURFACE option. The required option is:

```
*SURFACE, TYPE=ELEMENT, NAME=plate_interior_surf
all_plate_elems, interior
```

2. Define a surface interaction named **Friction**. Specify a friction coefficient of 0.5:

```
*SURFACE INTERACTION, NAME=Friction
*FRCITION
0.5
```

3. Define general contact for the entire model:

```
*CONTACT
*CONTACT INCLUSIONS

,
, plate_interior_surface
plate_interior_surface,
*CONTACT PROPERTY ASSIGNMENT
, , Friction
```

**Note:** If the first surface name is omitted, the default all-inclusive surface defined by Abaqus/Explicit is assumed; if the second surface name is omitted, Abaqus/Explicit assumes that self-contact is defined. This contact definition allows for (1) contact between the ball and all surfaces (interior and exterior) of the plate, and (2) self-contact between all surfaces (interior and exterior) of the plate.

4. Open the online Abaqus documentation. Examine the meaning of the **\*CONTACT INCLUSIONS** and **\*CONTACT PROPERTY ASSIGNMENT** options using the Abaqus Keywords Manual, if necessary.
5. Save all the changes and close the input file.

## Running the job and postprocessing the results:

Run the analysis using the following command:

```
abaqus job=w_plate_impact user=uniFiber
```

When the analysis is complete, use the following procedure to create contour plots, view material orientations, and investigate composite damage and failure using Abaqus/Viewer:

1. Start Abaqus/Viewer and open the file **w\_plate\_impact.odb**:

```
abaqus viewer odb=w_plate_impact.odb
```

2. Click the **Plot Contours on Deformed Shape** tool  to contour the stress state in the model. Notice that the failed elements are already removed.

**Note:** Automatic removal of failed elements is enabled when the STATUS output variable is written to the output database. The **Status Variable** tabbed page of the **Field Output** dialog box provides other options for the removal of failed elements, as shown in Figure W3–4.

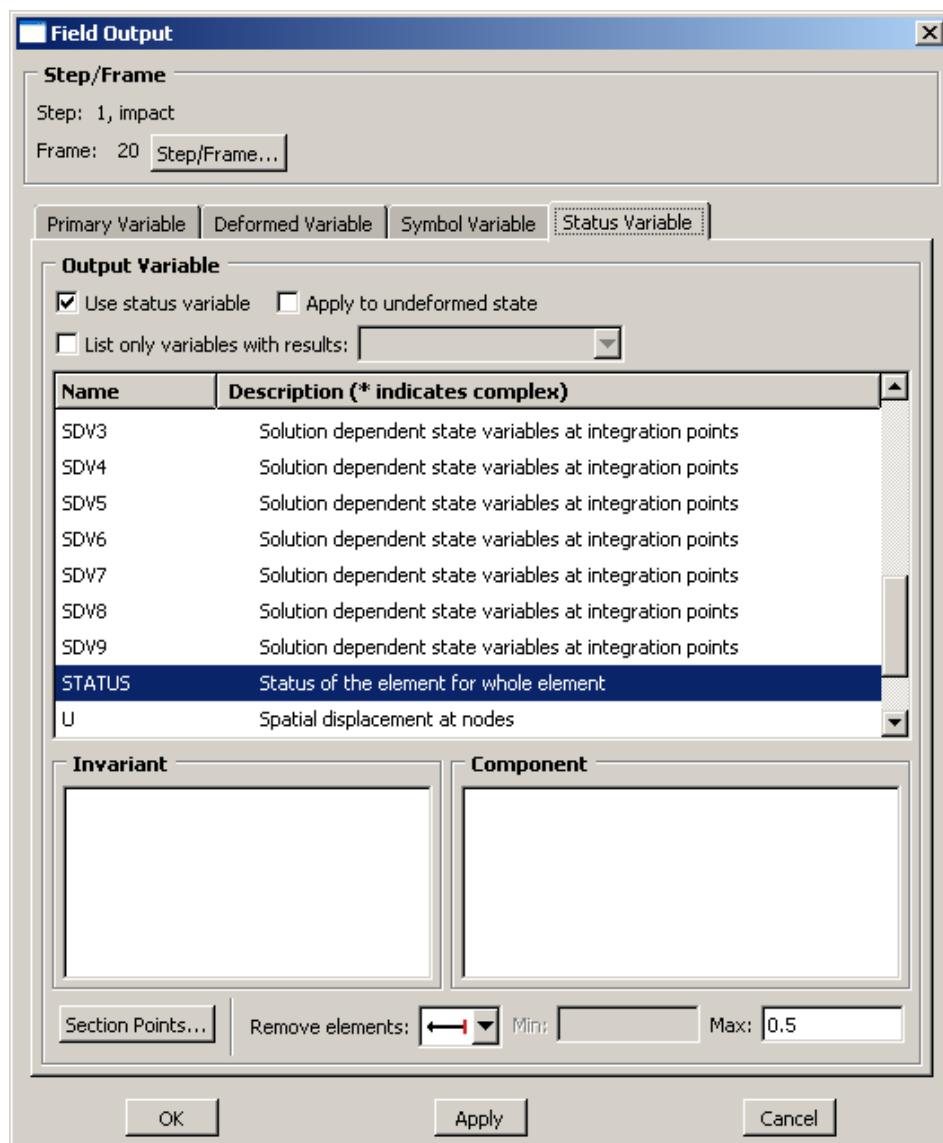


Figure W3–4. **Status Variable** options

3. Use the **Create Display Group** tool  in the toolbar to create a display group that includes a single layer of the composite plate.

In the **Create Display Group** dialog box, choose the item **Elements** and the method **Element sets**, as shown in Figure W3–5.

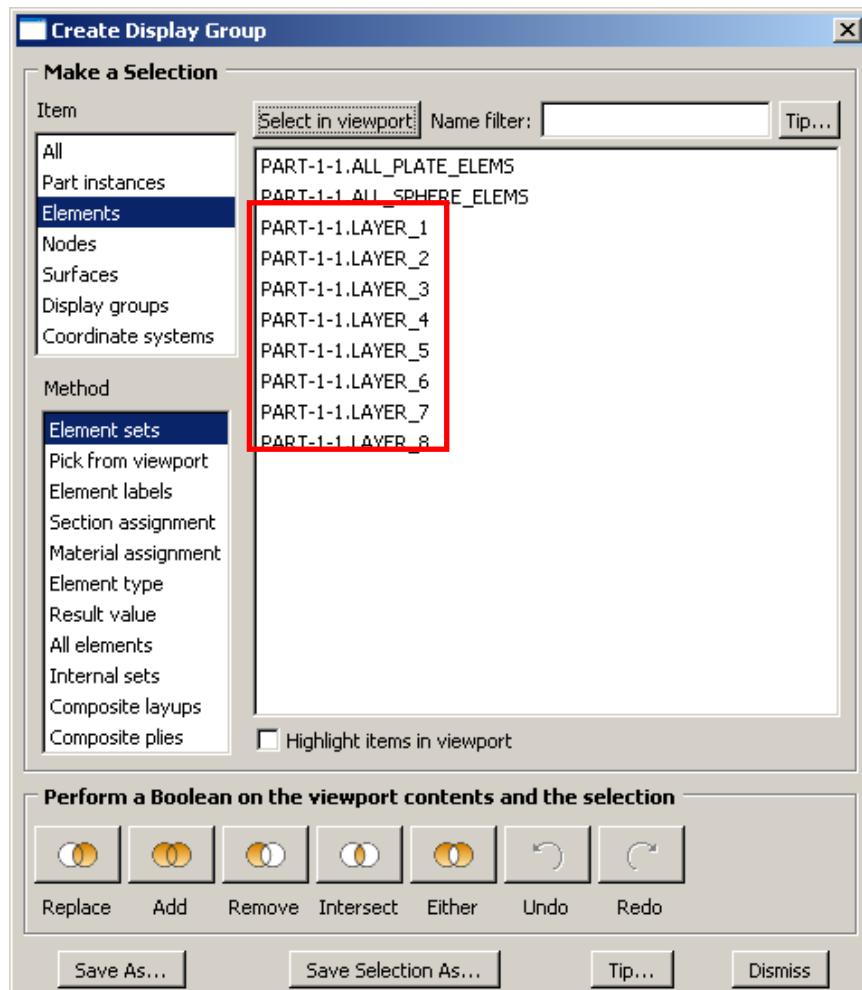


Figure W3–5. **Create Display Group** dialog box.

4. Select the element set **PART-1-1.LAYER\_1** and click **Replace** .

Note that the contour plot displays the stress contour in the first layer.

5. Click the **Plot Material Orientations on Deformed Shape** tool  in the toolbox.

The material orientation plot appears and displays the material orientations of the first layer.

Using a similar procedure, display the stress contour plots and material orientations of the other layers.

6. Click **Dismiss** to close the **Create Display Group** dialog box.

7. Click the **Replace All** tool  in the toolbar to display the whole model.
8. Plot the deformed model shape, as shown in Figure W3–6.

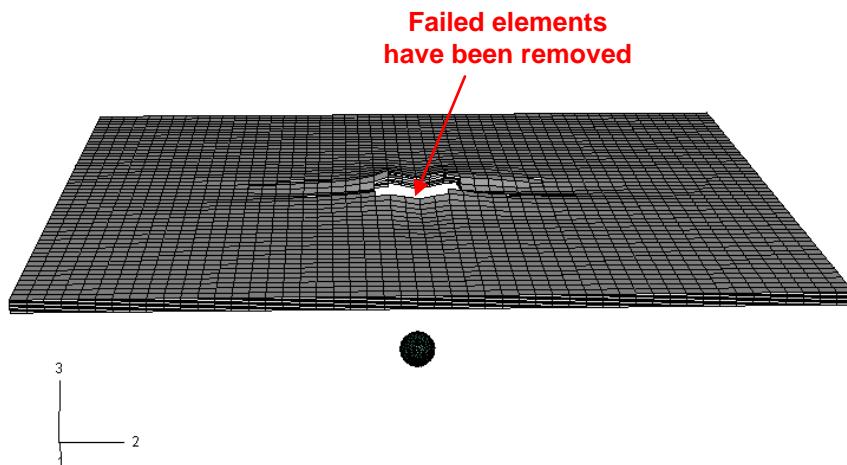


Figure W3–6. Deformed shape with failed elements removed.

9. Plot the velocity history of the rigid ball to evaluate the residual velocity of the projectile.
  - a. In the Results Tree, expand the **History Output** container underneath the output database named **w\_plate\_impact.odb**.
  - b. Double-click **Spatial velocity: V3 at Node 24311 in NSET SPHERE\_RP**. An X–Y plot appears and displays the velocity of the projectile, as shown in Figure W3–7.

The plot clearly shows that the projectile decelerates during impact and then remains a constant velocity after perforating the plate. If this were a model of debris impact on an aircraft panel or a ballistic impact on body armor, the design under consideration would be inadequate to keep the fragment from penetrating.

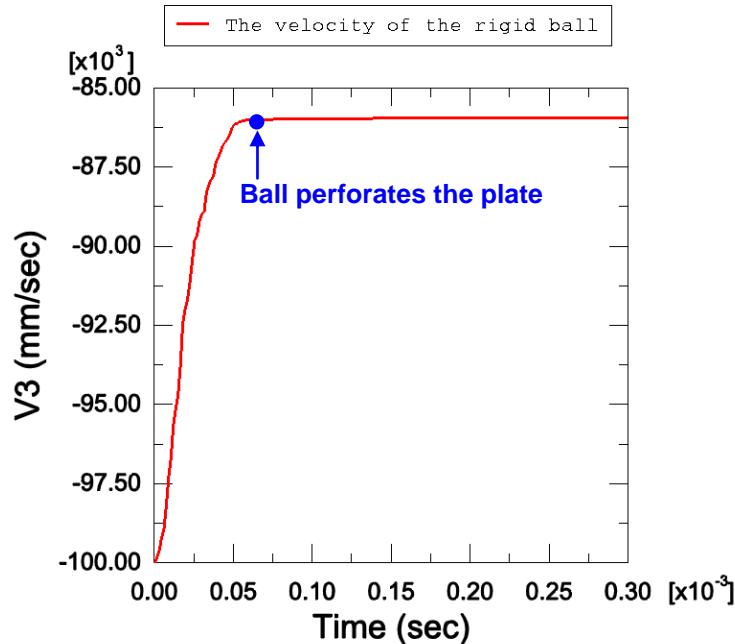


Figure W3–7. The velocity history of the projectile.

### Re-running the job without mass scaling

You may re-run the job without mass scaling to obtain results that are more physically meaningful. Note that this analysis may take approximately 2 hours to complete. If you are interested in running it, it is recommended you run the job outside of the course. The following discusses the results obtained without mass scaling.

1. The deformed model shape without the failed elements is shown in Figure W3–8. The plot shows that the deformation and failure of the composite plate are completely different from those with mass scaling (see Figure W3–6). Unlike the model with mass scaling, the projectile does not fully perforate the plate; instead it partially perforates the plate then rebounds. In addition, elements near the clamped edges also fail due to stress wave effects.

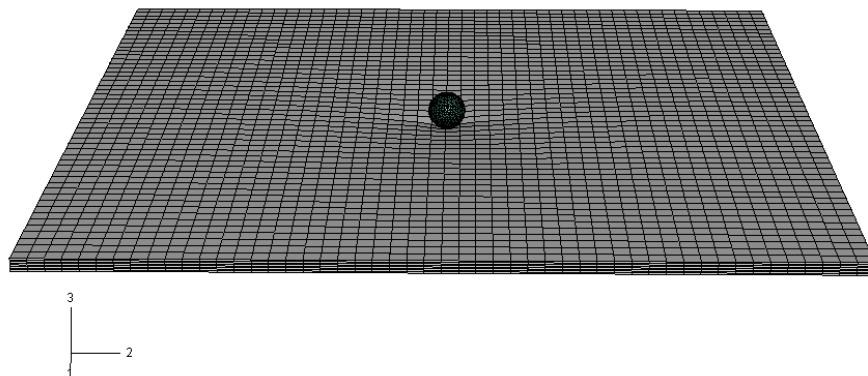


Figure W3-8. Deformed shape of the model without mass scaling.

2. The velocity history of the rigid ball is shown in Figure W3-9.

The plot shows that the projectile decelerates during impact until its velocity becomes zero, and then accelerates to a constant positive value after rebounding from the plate. Therefore we see the dangers of using mass scaling in a model where kinetic effects are significant. When modeled correctly, this plate layup indicates an ability to withstand the impact of the steel ball. Some damage to the plate occurs, but penetration is prevented.

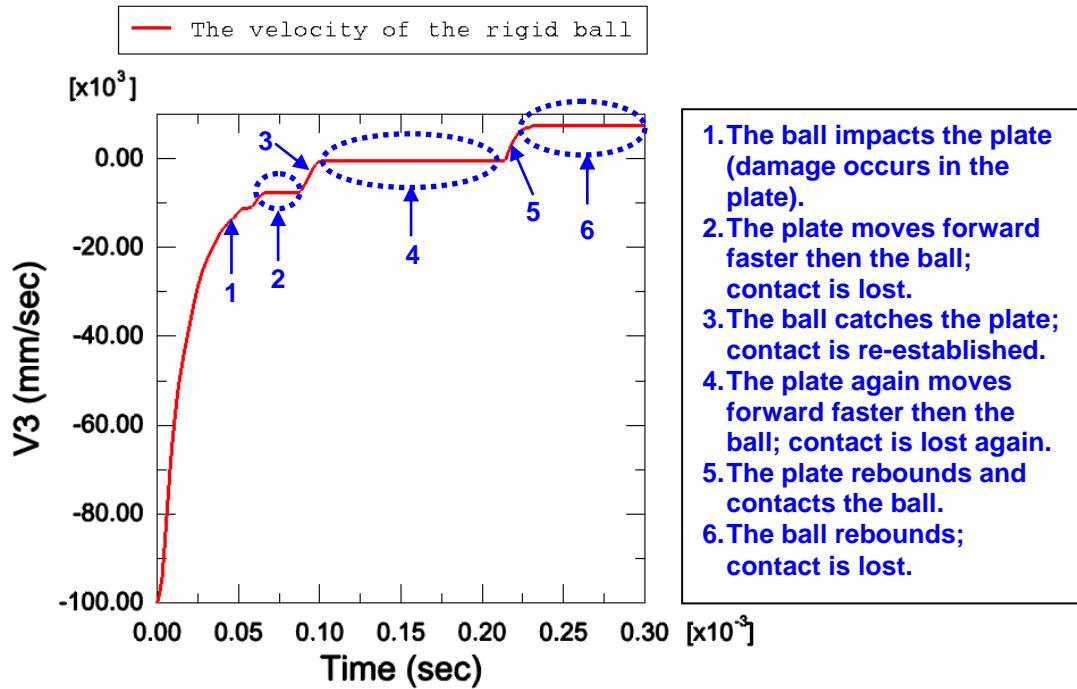


Figure W3-9. The velocity history of the projectile without mass scaling.

**Note: A complete input file is available for your convenience. You may consult this file if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input file is named**

**w\_plate\_impact\_complete.inp**

**and is available using the Abaqus fetch utility.**



# Notes

# Notes