Modeling of Composites in LS-DYNA

- **Some Characteristics of Composites**
- **Orthotropic Material Coordinate System**
- **User-defined Integration Rule for Shells**
- **Output for Composites**
- **Some Characteristics of Several Composite Material Models in LS-DYNA**
- **Closing Recommendations**

Two Types of Composites

- **Advanced composites have stiff, high strength fibers bound in a matrix material.**
  - **Each layer/lamina/ply is orthotropic by nature as the fibers run in a single direction.**
  - **Usually, an advanced composite section will have multiple layers and each lamina within the stack will have the fibers running in a different direction than in the adjacent lamina.**

- **A sandwich composite section has laminae which may be individually isotropic but the material properties and thickness may vary from lamina to lamina.**
  - **A foam core composite is a particular type of sandwich composite where a thick, soft layer of foam is sandwiched between two thin, stiff plies.**

Orthotropic Materials in LS-DYNA

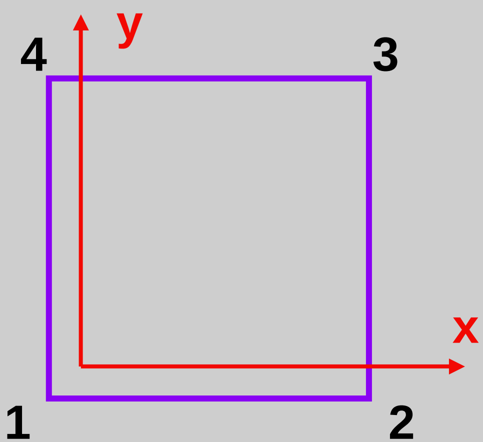
- **Orthotropic material constants are defined in the *material coordinate system*.**
- **The *material coordinate system* must be <u>initially</u> established for each orthotropic element and, in the case of shells, for each through-thickness integration point as well. This orientation comes from three sources.**
  - **In the material definition (*mat)**
    - **See description of "AOPT" in User's Manual under *mat_2 (orthotropic_elastic)**
  - **In the section definition (*section_shell)**
    - **A "beta" angle is given for each integration point**
  - **Optionally, in the element definition (*element_shell_beta, *element_solid_ortho)**

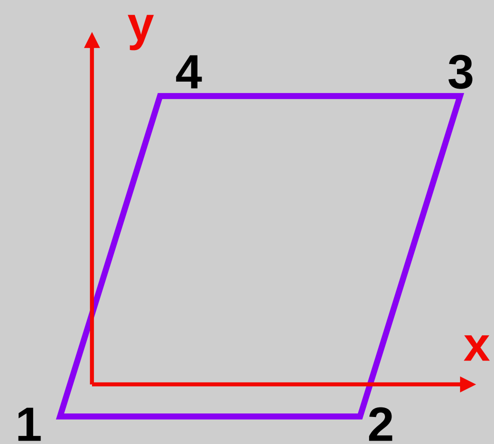
Orthotropic Materials in LS-DYNA

- As the solution progresses and the elements rotate and deform, the *material coordinate system* is automatically updated, following the rotation of the *element coordinate system*.
  - **The orientation of the *material coordinate system* and thus response of orthotropic shells can be very sensitive to in-plane shearing deformation and hourglass deformation, depending on how the *element coordinate system* is established.**
  - **To minimize this sensitivity, "Invarient Node Numbering", invoked by setting INN = 2 (shells) or 3 (solids) in *control_accuracy is highly recommended.**

Without Invarient Node Numbering (N1-to-N2 establishes element x-direction)

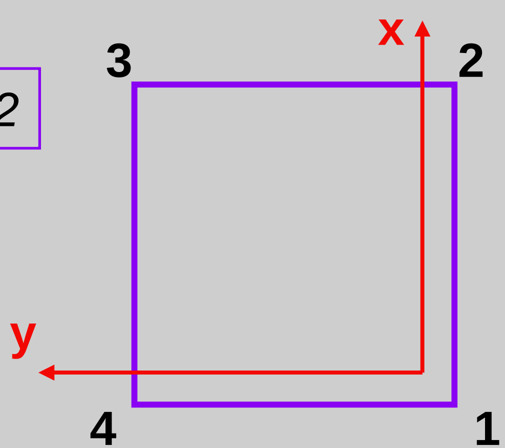
Case 1

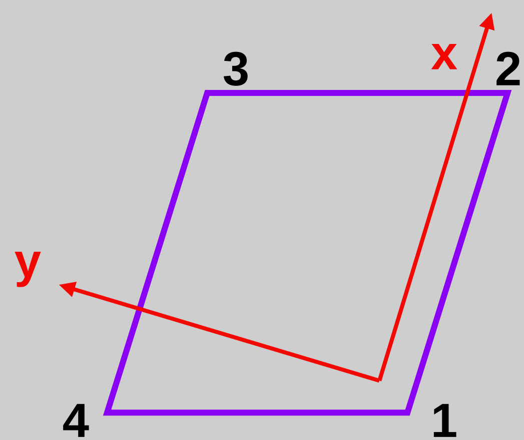




Element rotation = 0

Case 2



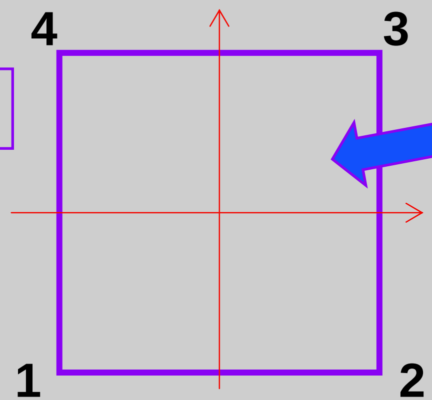


Element rotation = - 20^{o}

Undeformed *Deformed*

With Invarient Node Numbering
(based on element bisectors)

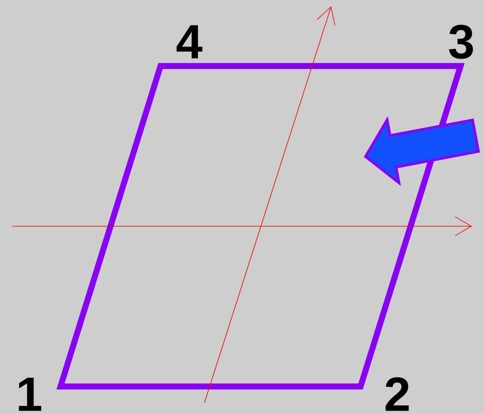
Case 1



Local x

= (0+90)/2 – 45

= 0^{o}



Local x

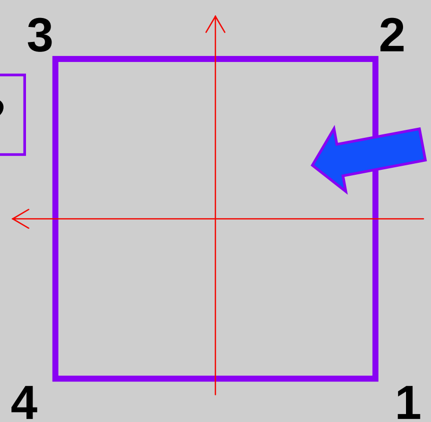
= (0+70)/2 – 45

= -10^{o}

Element rotation

= -10 – 0 = -10^{o}

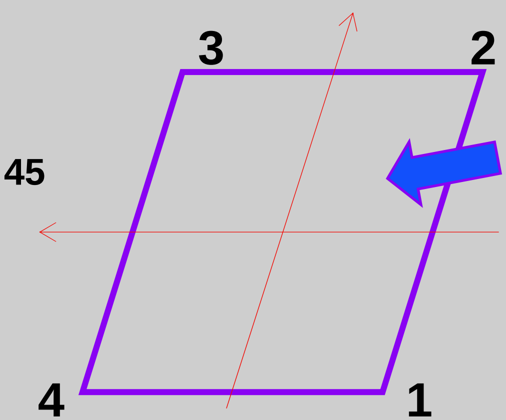
Case 2



Local x

= (90+180)/2 – 45

= 90^{o}



Local x

= (70+180)/2 – 45

= 80^{o}

Element rotation

= 80 - 90 = -10^{o}

Undeformed

Deformed

User-Defined (Through-Thickness) Integration

- Gaussian or Lobatto integration rules have pre-established integration point locations and weights (NIP <= 10).
  - **Lobatto includes integration points on the outside surfaces**
- Trapezoidal integration has equally spaced integration points.
- For composites, the user may need to define his/her own integration point locations and weights (corresponding to ply thicknesses) and may need to reference a different set of material constants for each integration point.

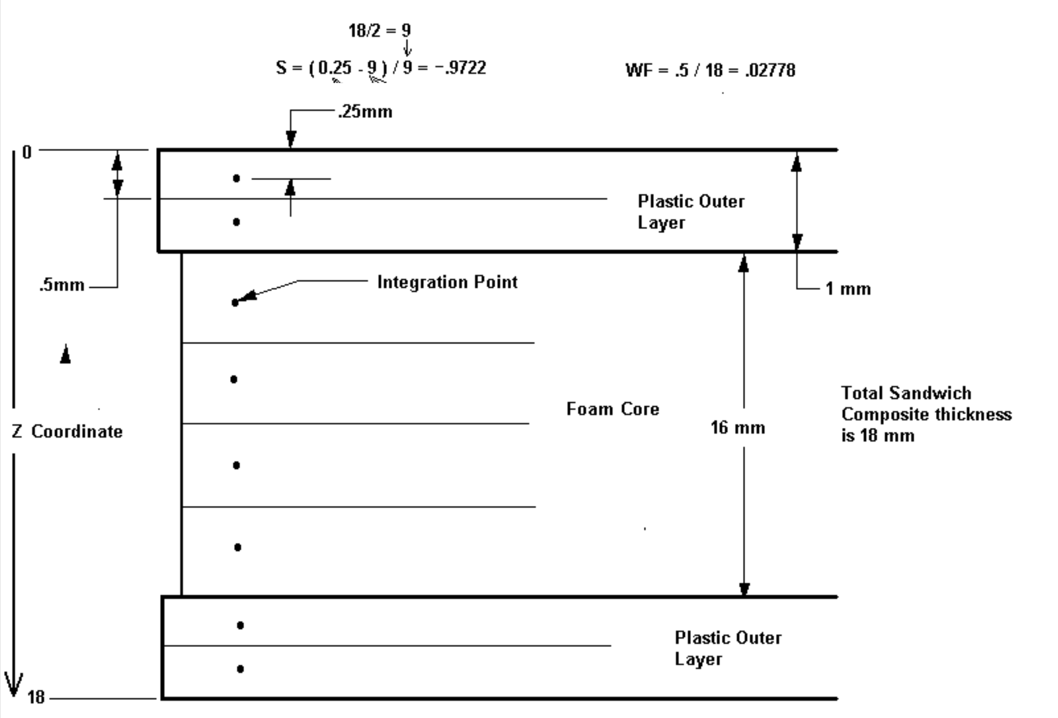
User-Defined Integration (970)

```
*PART
material 1
         1         1        11
*PART
material 2
         2         1        12
$---+----1----+----2----+----+----3----+----4----+----5----+----6-
*SECTION_SHELL
         1         2                                              -20
 18.000000 18.000000 18.000000 18.000000
*mat_layered_linear_plasticity
11, 2.7e-6, 73.4, 0.32, 1e9


*mat_layered_linear_plasticity
12, 6.3e-7, 0.286, 0.3, 1e9


*INTEGRATION_SHELL
20,8,0
-.9722, .02778, 1
-.9167, .02778, 1
-.6667, .22222, 2
-.2222, .22222, 2
 .2222, .22222, 2
 .6667, .22222, 2
 .9167, .02778, 1
 .9722, .02778, 1
*ELEMENT_SHELL
       1       1       1       2      33
       2       1       2       3      34
```

Negative value indicates user integration rule



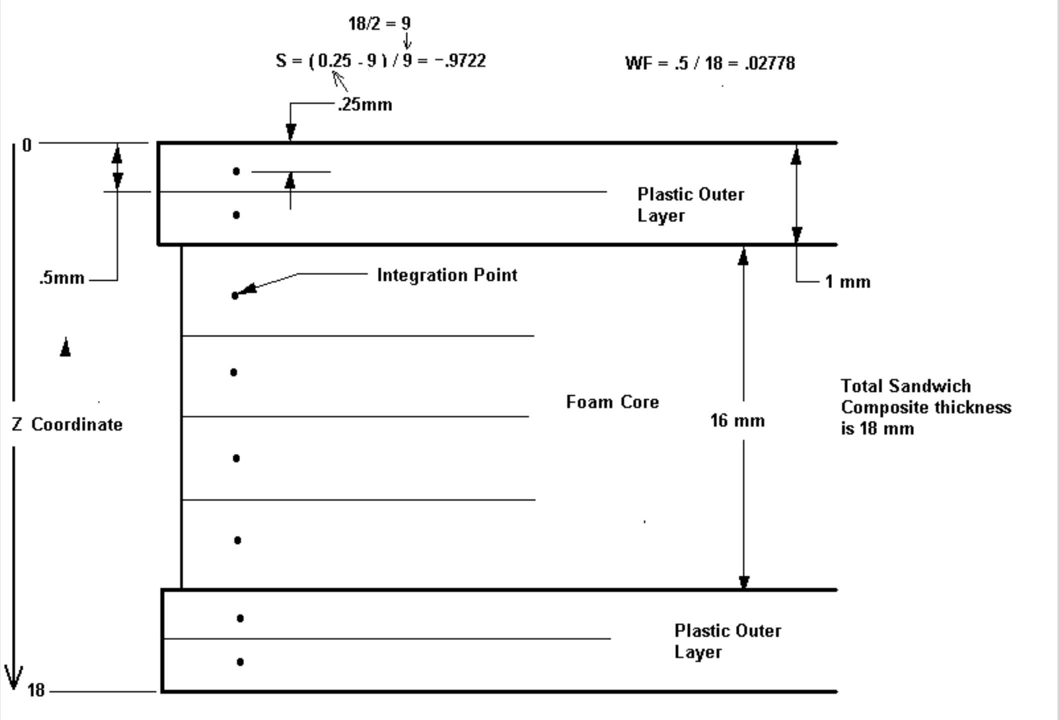
Sandwich Composite Material Cross-Section Example

User-Defined Integration (971)

```
$ no *section command needed
$ thickness is sum of thick values given in *PART_COMPOSITE
$ no need for multiple *PART commands
$
*PART_COMPOSITE
$ pid, elform
1, 2
$ mid, thick, beta,,mid,thick,beta
   11,   0.5,,,       11,  0.5
   12,   4.0,,,       12,  4.0
   12,   4.0,,,       12,  4.0
   11,   0.5,,,       11,  0.5
*mat_layered_linear_plasticity
11, 2.7e-6, 73.4, 0.32, 1e9


$  NOTE: foam core could use a different
$        material model (971)
*mat_layered_linear_plasticity
12, 6.3e-7, 0.286, 0.3, 1e9


*ELEMENT_SHELL
       1       1       1       2      33
       2       1       2       3      34
```



Sandwich Composite Material Cross-Section Example

Output for Composites

- For composite material models, stresses (and strains) will be written in the material coordinate system rather than the global coordinate system *if* CMPFLG (and STRFLG) is set to 1 in *database_extent_binary.
  - **Useful option for postprocessing of fiber and matrix stresses.**

- Set MAXINT in *database_extent_binary to the total number of through-thickness integration points in your composite shell. By default, stresses only at the top, bottom, and middle integration points are written.

Output for Composites

- Some composite material models have “extra history variables” that help to track modes of failure in each integration point. (See material documentation in the User’s Manual for details.)
  - **NEIPS (shells) or NEIPH (solids) in *database_extent_binary should be set to the number of extra history variables needed. For example, if you want to track the damage parameter in mat_054, set NEIPS to 6.**

Composite Material Models

- ***mat_2 (elastic_orthotropic)**
  - **9 elastic constants (solids); 6 elastic constants (shells).**
  - **Total Lagrangian formulation (okay for large elastic deformations).**
  - **No failure criteria.**

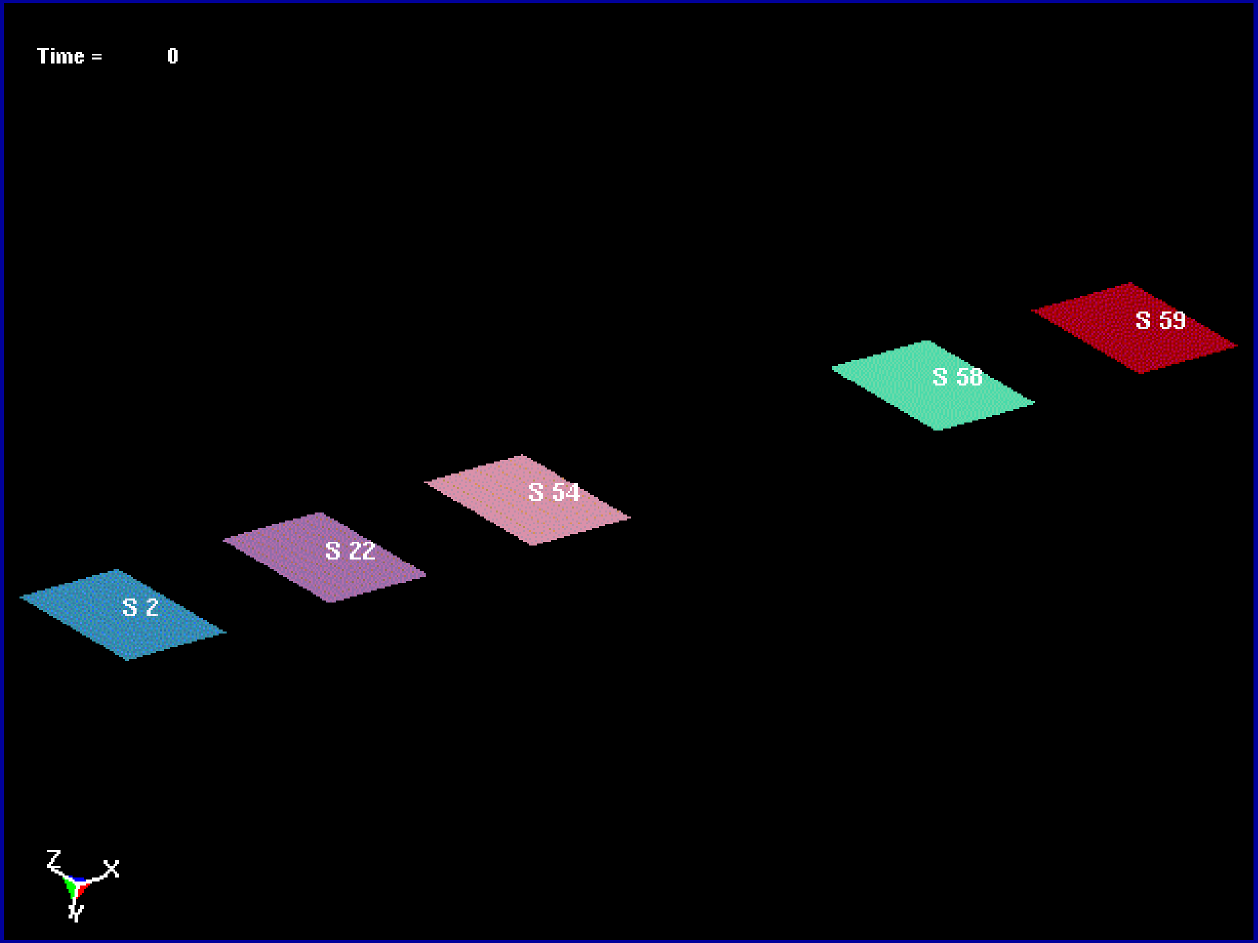
- **Each of the following orthotropic materials offer a particular brand of fiber/matrix damage and failure criteria. Up to 5 strength values are given (XT, XC, YT, YC, SC).**
  - ***mat_22 (composite_damage)**
  - ***mat_54,55 (enhanced_composite_damage)**
  - ***mat_58 (laminated_composite_fabric)**
    - *mat_158 includes strain rate effects
  - ***mat_59 (composite_failure(_shell, _solid)_model)**
    - Can be used with shells or solids

Composite Material Models

- The paper "Crashworthiness Analysis with Enhanced Composite Material Models in LS-DYNA - Merits and Limits", Schweizerhof et al, 5th International LS-DYNA User's Conference (1998) provides some insight into several composite material models in LS-DYNA, including mat_54, mat_58, and mat_59. This paper is available as a PDF file.

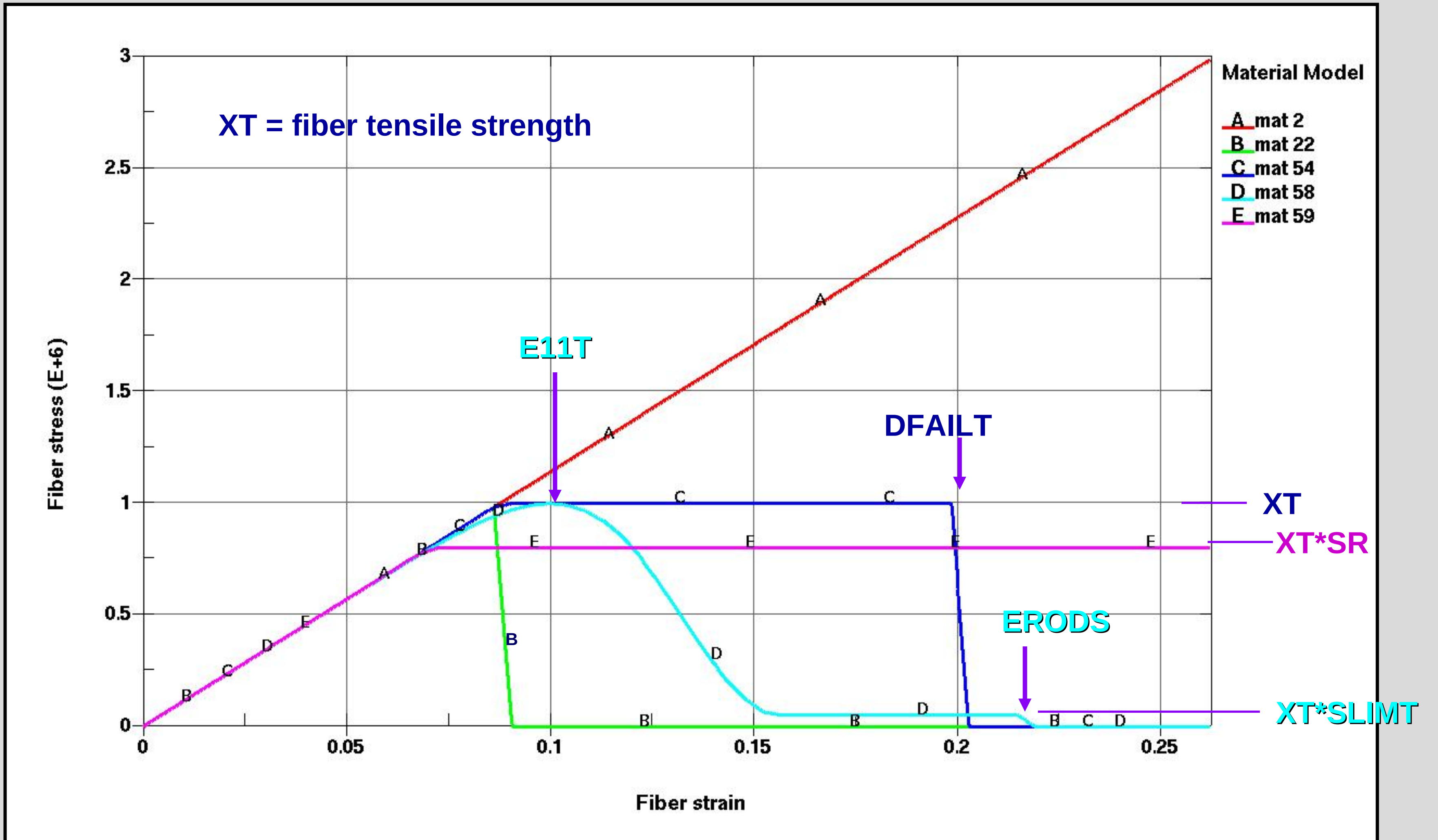
Comparison of Several Composite Material Models

- Uniaxial Tension in Fiber Direction



Comparison of Several Composite Material Models

- Uniaxial Tension in Fiber Direction



Laminated Shell Theory

- Use of Laminated Shell Theory (LST) is important if a composite shell has layers of dissimilar materials.
  - LST corrects for the incorrect assumption of uniform constant shear strain through the thickness of the shell.
  - Without LST, a sandwich composite will generally be much too stiff.
  - LAMSHT=1 in *control_shell invokes LST for material models 22, 54, 55, 76
  - *Mat_layered_linear_plasticity (114) is a plasticity model much like mat_024 but which includes LST.

Composite Material Models

- ***mat_116 (composite_layup)**
  - **Orthotropic elastic *resultant* formulation (no stresses calculated)**
  - **Very efficient for large number of layers**
  - **Requires *integration_shell**
    - Material constants can vary from layer to layer
  - **Does NOT use laminated shell theory (not good for foam core/sandwich composites)**

Composite Material Models

- ***mat_117 (composite_matrix)**
- ***mat_118 (composite_direct)**
  - **Resultant formulation (no stresses calculated)**
  - **21 coefficients of symmetric stiffness matrix are input directly**
    - Stiffness coefficients in 117 given in material coord system
    - Stiffness coefficients in 118 given in element coord system (less storage req'd)
  - **Shell thickness is inherent in stiffness matrix. Thus uniform thickness of part is mandatory.**

Composite Material Models

- ***mat_161 (composite_msc)**
  - **Proprietary model from Materials Sciences (requires license add-on)**
  - **Available for solids only**
  - **Offers fiber shear and fiber crush failure criteria**
  - **Can predict delamination**
  - ***mat_162 like *mat_161 but adopts damage mechanics approach for softening after damage initiation**

A Word about Delamination

- Shells do not have σ_{zz} component of stress and thus are not well-suited to rigorous study of composite delamination.

- Delamination may be approximated using multiple layers of shells tied with *CONTACT_..._TIEBREAK in which failure of contact represents delamination.

Closing Recommendations

- Most composites do not stretch significantly before breaking. To promote numerical stability, shell thinning option should NOT be invoked. Leave ISTUPD in *control_shell set to zero.
- 'Noise' in response can be mitigated by stiffness damping in some cases. See *damping_part_stiffness.
- Shell bulk viscosity (*hourglass, ITYPE=-1) may aid stability in compressive modes of response.