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 Common 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 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

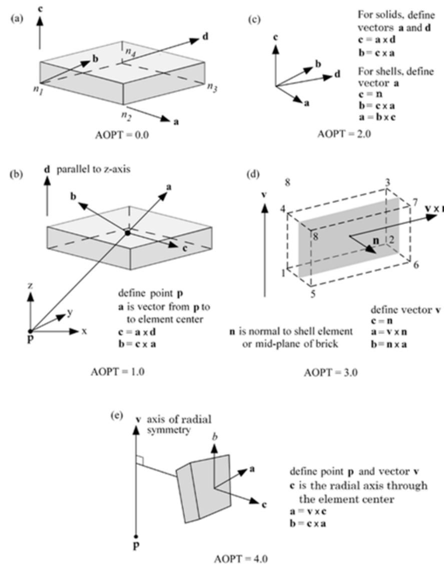
- Brief Definitions
  - Isotropic Material: Material's properties do not depend on direction. In other words, there is no strong or weak direction.
  - Orthotropic Material: Material's properties are different along each orthogonal axis (e.g., a fiber reinforced composite).
- Orthotropic materials need a *material coordinate system* to track the orientation of the orthogonal material axes.

Orthotropic Materials in LS-DYNA

- From data in the input deck, the *material coordinate system* is established at t=0 for each element integration point. This initial orientation can come from three sources:
  - From the material definition (*MAT)
  - For shells and tshells, also from *PART_COMPOSITE(_TSHELL)
    - A "BETA" angle is given for each though-thickness layer
  - If certain *ELEMENT options are invoked, from the element definition. Data given here will supercede conflicting orientation data from *MAT and *PART_COMPOSITE.
    - *ELEMENT_SOLID_ORTHO
    - *ELEMENT_SHELL_BETA, *ELEMENT_SHELL_COMPOSITE
    - *ELEMENT_TSHELL_COMPOSITE

Material Coordinate System for Orthotropic Materials

- Shown to the right are figures from *MAT_002 in the User's Manual. These figures help to illustrate the various methods of establishing a material coordinate system at the material level. Depending on the choice of AOPT, certain vector(s) may have to be defined in the *MAT input.
- BETA angles in *PART_COMPOSITE and/or optional input in the *ELEMENT definition also contribute toward the initial orientation of the material coordinate system for each element integration point
- Orientation of material orientation for each integration point can be shown (and thus confirmed) using LS-PrePost
  - Ident > Element > Mat Dir



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,tshells), 3 (solids), or 4 (shells, tshells, solids) in *CONTROL_ACCURACY, is highly recommended.

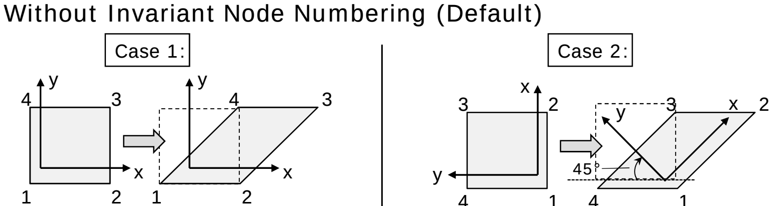
Invarient Node Numbering

- Invarient node numbering is invoked with *CONTROL_ACCURACY (INN)
  - Always recommended and particularly important for orthotropic materials
  - By default, local element x-direction aligns with N1-N2 edge
  - Changing element connectivity without invoking invarient node numbering option (default condition) shifts local coordinate system based on element shape >> results can be sensitive to connectivity!!
  - Changing element connectivity and invoking invariant node numbering option shifts local system by a 90-degree increment so that regardless of element shape, results are insensitive to connectivity.

Why should we use Invariant Node Numbering?
→ *CONTROL_ACCURACY (INN=2/3/4)

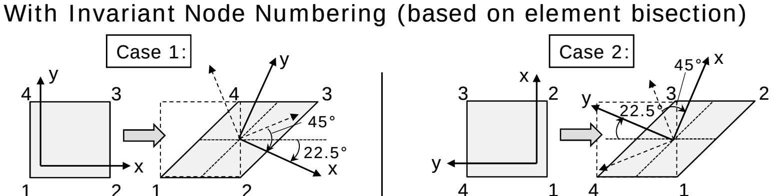
Change of element coordinate system (ECS) during deformation (Example – Shells)

- Without Invariant Node Numbering (Default)



Rotation of ECS: 0° | Rotation of ECS: 45°

- With Invariant Node Numbering (based on element bisection)



Rotation of ECS: 22.5° | Rotation of ECS: 22.5°

Slide courtesy of Stefan Hartmann, Dynamore GmbH

Steps to define local element coordinate system for shells with invariant node numbering:

- Create vectors $\mathbf{a}_1$ and $\mathbf{a}_2$ through midpoints of element sides
- Create shell normal vector $\mathbf{n}$ with:

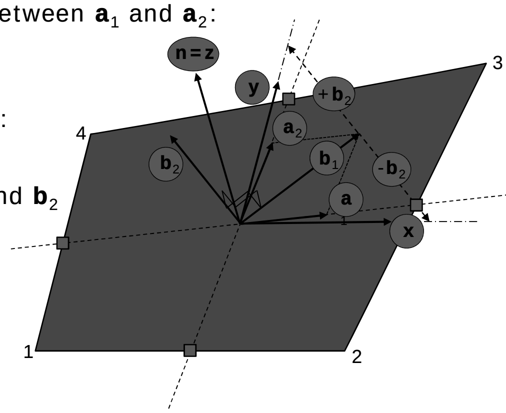
  $\mathbf{n} = \mathbf{a}_1 \times \mathbf{a}_2$
- Define vector $\mathbf{b}_1$ as middle between $\mathbf{a}_1$ and $\mathbf{a}_2$:

  $$\mathbf{b}_1 = \frac{\mathbf{a}_1}{\|\mathbf{a}_1\|} + \frac{\mathbf{a}_2}{\|\mathbf{a}_2\|}$$
- Create orthogonal vector $\mathbf{b}_2$:

  $\mathbf{b}_2 = \mathbf{n} \times \mathbf{b}_1$
- Rotate back (45°) from $\mathbf{b}_1$ and $\mathbf{b}_2$ to get $\mathbf{x}$ and $\mathbf{y}$:

  $$\mathbf{x} = \frac{\sqrt{2}}{2}\left(\frac{\mathbf{b}_1}{\|\mathbf{b}_1\|} - \frac{\mathbf{b}_2}{\|\mathbf{b}_2\|}\right)$$

  $$\mathbf{y} = \frac{\sqrt{2}}{2}\left(\frac{\mathbf{b}_1}{\|\mathbf{b}_1\|} + \frac{\mathbf{b}_2}{\|\mathbf{b}_2\|}\right)$$



Slide courtesy of Stefan Hartmann, Dynamore GmbH

User-Defined (Through-Thickness) Integration

- Gaussian or Lobatto integration rules have pre-established integration point locations and weights
  - These rules are good for up to 10 integraton points
  - 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 material for each layer/integration point

User-Defined Shell Integration using *PART_COMPOSITE

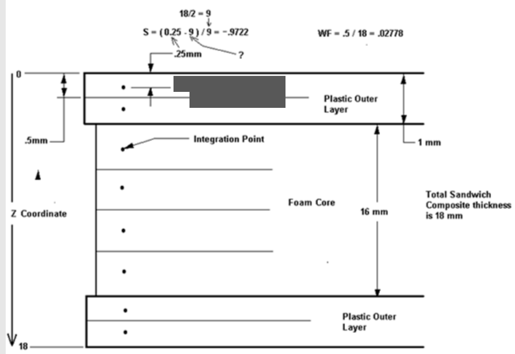
```
$ no *section command needed
$ thickness is sum of thick values given in *PART_COMPOSITE
$ no need for multiple *PART commands
$
*PART_COMPOSITE
$ pid, elform
1, 2
$ mid, thick, beta,,mid,thick,beta
   11,   0.5,,,      11,  0.5
   12,   4.0,,,      12,  4.0
   12,   4.0,,,      12,  4.0
   11,   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) are 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 (cont'd)

- Some composite material models have "extra history variables" that help to track modes of failure in each integration point
  - 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 tensile matrix failure status (3rd extra history variable) of each integration point in MAT_054 shells, set NEIPS to 3 (or greater) and MAXINT to the number of through-thickness integration points

Composite Material Models

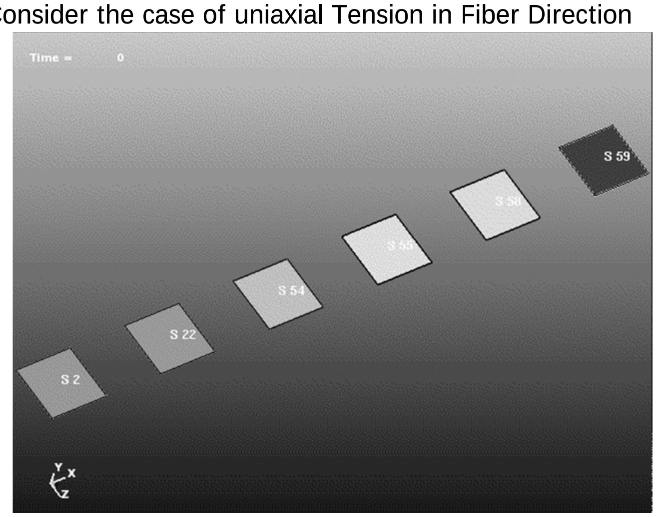
- *MAT_002 (ELASTIC_ORTHOTROPIC)
  - 9 elastic constants (EA,EB,EC, PRBA,PRCA,PRCB, GAB,GBC,GCA)
  - Total Lagrangian formulation (hyperelastic)
    - P-K stress vs. Green strain
  - No failure (unless added via *MAT_ADD_EROSION)
- Each of the following orthotropic material models differ in their fiber/matrix damage and failure criteria.
  - *MAT_022 (COMPOSITE_DAMAGE)
  - *MAT_054,055 (ENHANCED_COMPOSITE_DAMAGE)
  - *MAT_058 (LAMINATED_COMPOSITE_FABRIC)
  - *MAT_158 like 58 but includes strain rate effects
  - *MAT_059 (COMPOSITE_FAILURE(_SHELL, _SOLID)_MODEL)
  - Mats 22, 54, 55, and 59 can be used with shells, tshells, and <u>solids</u>

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 mats 54, 58, and 59.
  - URL for paper: ftp://ftp.lstc.com/outgoing/jday/composites/crash_composites_paper.pdf
- Other notes and examples related to composites are also available in ftp://ftp.lstc.com/outgoing/jday/composites

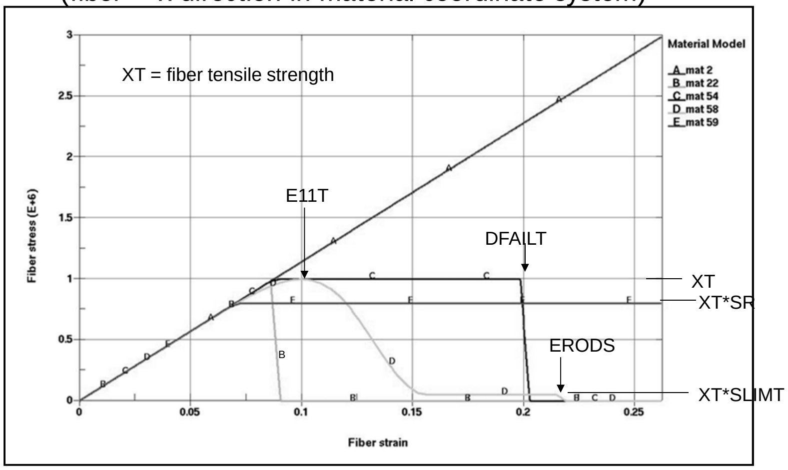
Comparison of Several Composite Material Models

- Consider the case of uniaxial Tension in Fiber Direction



Comparison of Several Composite Material Models

- Consider case of uniaxial Tension in Fiber Direction (fiber = x direction in material coordinate system)



- Similar input parameters for fiber compression, matrix tension, matrix compression, and shear

Laminated Shell Theory

- Use of Laminated Shell Theory (LST) is important if a composite shell has layers of dissimilar materials
  - LST corrects for the default assumption of uniform transverse shear strain through the thickness of a shell or tshell element
  - Without LST, a sandwich composite will generally be much too stiff
  - LST is activated using parameter LAMSHT in *PART_COMPOSITE
    - LAMSHT=3: LST active for shells
    - LAMSHT=4: LST active for tshells
    - LAMSHT=5: LST active for shells and tshells
- When LST is active, shear correction factor SHRF in *PART_COMPOSITE should be set to the default value of 1.0

Composite Material Models (cont'd)

- *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 incorporate Laminated Shell Theory (not good for foam core/sandwich composites)

Composite Material Models (cont'd)

- *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 coordinate system
    - Stiffness coefficients in 118 given in element coordinate system (less storage req'd)
  - Shell thickness is inherent in stiffness matrix. Thus uniform thickness of part is mandatory

Composite Material Models (cont'd)

- *MAT_161 (COMPOSITE_MSC)
  - Proprietary model from Materials Sciences (requires license add-on, contact keys@lstc.com)
  - Technical support provided by Bazle Haque at University of Delaware (gama@udel.edu)
  - Available for solids only
  - *MAT_162 like *MAT_161 but adopts damage mechanics approach for softening after damage initiation
- *MAT_219 (CODAM2) New in LS-DYNA
  - 2nd Generation UBC Composite Damage Model
  - Continuum damage mechanics approach for fiber-reinforced laminates where nonlinear behavior arises out of damage evolution in each sub-laminate
  - Optional nonlocal averaging in strain-based failure criterion
  - For shells, tshells, solids

A Few Words about Delamination

- Delamination prediction requires multiple layers of elements
- Layers can be tied with *CONTACT_AUTOMATIC_..._TIEBREAK in which failure of contact represents delamination
  - See OPTIONs 6 – 11 in the tiebreak contact (See section on CONTACT)
- Thin cohesive elements (solid ELFORM 19, 20) representing the bond material between composite layers is another alternative
  - Small or zero thickness of cohesive element does not affect time step
  - Cohesive material is modeled with *MAT_138, 184, 185, or 186
  - Eroding contact needed to account for erosion of cohesive elements

Closing Recommendations

- Most composites do not stretch significantly before breaking. To promote numerical stability, shell thinning option should NOT be invoked
  - Leave parameter ISTUPD in *CONTROL_SHELL set to zero
- 'Noise' in response can be mitigated by stiffness damping in some cases
  - *DAMPING_PART_STIFFNESS, COEF=0.1
- Shell bulk viscosity may aid stability in compressive modes of response
  - *CONTROL_BULK_VISCOSITY, TYPE=-2

