Introduction to Composites Modelling in LS-DYNA
1. Introduction
Modelling across the length scales

Micro-scale

- Individual fibres + matrix + Interface

Meso-scale: Single Ply

- Material homogenisation at ply level

Meso-scale: Laminate

- Combine several homogenised layers

Macro-scale

- Assembly of structural elements (incl. BCs, loads)

Fig 1. Optical microscopy of a cross-ply CFRP laminate [0/90]_ns after out-of-plane impact damage (Olsson et al, 2012)

- Transverse tensile fracture: \( \sigma_{22} > 0 \) \( Y_T \)
- Transverse compressive fracture: \( \sigma_{22} < 0 \) \( Y_C \)
- Longitudinal tensile fracture: \( \sigma_{11} > 0 \) \( X_T \)
- Longitudinal compressive fracture: \( \sigma_{11} < 0 \) \( X_C \)
### Outline – FE Modelling of Composites

#### Intra-laminar behaviour

<table>
<thead>
<tr>
<th>Element</th>
<th>Material</th>
<th>Structure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shells (2D stress state)</td>
<td>*MAT_022: COMPOSITE DAMAGE</td>
<td>One Shell Layer</td>
</tr>
<tr>
<td>Thick shells (2D or 3D stress state)</td>
<td>*MAT_054/055: ENHANCED_COMPOSITE DAMAGE</td>
<td>Layers of Solid Elements</td>
</tr>
<tr>
<td>Solids (3D stress state)</td>
<td>*MAT_058/158: LAMINATED_COMPOSITE_FABRIC/RATE_SENSITIVE</td>
<td>Layers of Shell Elements</td>
</tr>
<tr>
<td></td>
<td>*MAT_059: COMPOSITE_FAILURE_MODEL</td>
<td></td>
</tr>
<tr>
<td></td>
<td>*MAT_261: LAMINATED_FRACTURE_DAIMLER_PINHO</td>
<td></td>
</tr>
<tr>
<td></td>
<td>*MAT_262: LAMINATED_FRACTURE_DAIMLER_CAMANHO</td>
<td></td>
</tr>
<tr>
<td></td>
<td>*MAT_161/162: COMPOSITE_MSC (Commercial)</td>
<td></td>
</tr>
</tbody>
</table>

#### Inter-laminar behaviour

<table>
<thead>
<tr>
<th>Material</th>
<th>Structure</th>
</tr>
</thead>
<tbody>
<tr>
<td>*MAT_138: COHESIVE_MIXED_MODE</td>
<td></td>
</tr>
<tr>
<td>*MAT_184: COHESIVE_ELASTIC</td>
<td></td>
</tr>
<tr>
<td>*MAT_185: COHESIVE_TH</td>
<td></td>
</tr>
<tr>
<td>*MAT_186: COHESIVE_GENERAL</td>
<td></td>
</tr>
<tr>
<td>*MAT_240: COHESIVE_MIXED_MODE_ELASTO_PLASTIC_RATE</td>
<td></td>
</tr>
<tr>
<td>*CONTACT_TIEBREAK: AUTOMATIC_ONE WAY_SURFACE_TO_SURFACE</td>
<td></td>
</tr>
<tr>
<td>*MAT_169: MAT_ARUP_ADHESIVE</td>
<td></td>
</tr>
</tbody>
</table>
2. Material & layup definitions
Shell Orientation

For shells of orthotropic/anisotropic materials, there are 3 coordinate systems to consider:

- Global \([X,Y,Z]\)
- Element (local) \([x,y,z]\)
- Material \([a,b,c]\)

The material coordinate system needs to be defined for each layer \(i\) of the element.
Shell Orientation

The orthotropic material models are formulated w. r. t. a local material coordinate system.

\[ \theta_i = \beta + \beta_i \]

The material orientation comes from two sources:

- **\( \beta \):** angle to define basic material coordinate system in the element. This is defined using:
  - AOPT (and BETA) within the *MAT-keyword cards
  - *ELEMENT_SHELL_BETA or *ELEMENT_SOLID_ORTHO (overwrites BETA in *MAT-keyword card)

- **\( \beta_i \):** relative angle to define material coordinate system in each layer (IP through-thickness). This is defined using:
  - *SECTION_SHELL/_TSHELL
  - *PART_COMPOSITE(_TSHELL)
  - *ELEMENT_SHELL_COMPOSITE
**Example.** The total offset \( (\theta_i) \) of any of the layers in the composite is the sum of the:

- ‘base angle-offset’ \( (\beta) \)+ ‘layer angle-offset’ \( (\beta_i) \)

\[
B_1 = 30^\circ \text{ (‘layer offset’, on *PART_COMPOSITE)}
\]

The material orientation for this particular layer is total angle of 55°

\[
\text{BETA} = 25^\circ \text{ (‘base offset’, on *ELEMENT_SHELL_BETA)}
\]
Local Coordinate Systems

- Definition of BETA in *MAT-keyword card ($a, b, c = 1, 2, 3$)

EQ.0.0: locally orthotropic with material axes determined by element nodes as shown in part (a) of Figure M2-1. The $a$-direction is from node 1 to node 2 of the element. The $b$-direction is orthogonal to the $a$-direction and is in the plane formed by nodes 1, 2, and 4. When this option is used in two-dimensional planar and axisymmetric analysis, it is critical that the nodes in the element definition be numbered counterclockwise for this option to work correctly.

- $a$-direction defined based on element coordinate system
- Can be changed with:
  - *ELEMENT SHELL_BETA
  - *ELEMENT SOLID_ORTHO

EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the $a$-direction. This option is for solid elements only.

AOPT = 0.0

AOPT = 1.0 (solid only)
Local Coordinate Systems

- Definition of BETA in *MAT-keyword card \((a,b,c = 1,2,3)\)

AOPT = 2.0 (shell)

\[
a = a_{\text{input}} - \frac{a_{\text{input}} \cdot n}{n \cdot n} n
\]

EQ.2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

Define:
- \(a\) (Shell)
- \(a\ & d\) (Solid)

AOPT = 2.0 (solid)

\[
c = a \times d
\]

\(c\) is orthogonal to the \(a,d\) plane

\(a,d\) are input. The computed axes do not depend on the element.

\[
b = c \times a
\]

\(b\) is orthogonal to the \(c,a\) plane
Local Coordinate Systems

- Definition of BETA in *MAT-keyword card \((a, b, c = 1, 2, 3)\)

EQ.3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector \(v\) with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.

EQ.4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector \(v\), and an originating point, \(P\), which define the centerline axis. This option is for solid elements only.

LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
Local Coordinate Systems

Composites Webinar

Figure M2-2. Flow chart showing how for each solid element LS-DYNA determines the vectors \((a, b, c)\) from the input.
Local Coordinate Systems

Figure M2-3. Flowchart for shells: (a) check for coordinate system ID; (b) process AOPT; (c) determine \( \beta \); and (d) for each layer determine \( \beta_f \).
In LS-DYNA the default output option writes stresses and strains into the PTF (d3plot) binary files using the global coordinate system.

D3PLOT can then transform these results from the global coordinate system into one of the other systems that it supports.

CMPFLG = 0
The default output can be changed on *DATABASE_EXTENT_BINARY by setting CMPFLG=1.

When this option is set the output for any elements defined using an orthotropic material model are changed so that they are written out using the material axes coordinate system.

Nothing else is changed in the PTF file so there is nothing to indicate this change to a post processor. Therefore by default the post processor will still assume that the results are in the global system.

Any coordinate system transformations or calculations will also assume that the results are starting in the global coordinate system.
To correctly interpret the results in a PTF when CMPFLG is used, the post processor needs to know the following information in addition to the value of CMPFLG:

1. Which elements use orthotropic materials.

2. The orientation of the material axes relative to the global axes throughout the analysis.

   * For a composite Shell or Thick Shell element this information is needed for all the through thickness integration points.

The rules in LS-DYNA for calculating the material axes are complex as they can be defined via a number of different options. Some of these options are additive while others overwrite each other.

- AOPT on *MAT cards – five options some apply to Shells and some to Solids.
- BETA and COMPOSITE options for *ELEMENT_SHELL / *ELEMENT_TSHELL.
- *PART_COMPOSITE – can vary the material and define orientation angles.
- PART ID on *INTEGRATION_SHELL – can vary the material.
PRIMER v15 knows about all of the different methods and rules for calculating the material coordinate system and can sketch them via the entities panel.

PRIMER v15 writes this information along with the value of `CMPFLG` to the ZTF file.

In addition to the material axes information PRIMER v15 also writes information to the ZTF file about what material is used for each element and integration point so D3PLOT can identify what coordinate system results are in
With a ZTF file D3PLOT can correctly identify **CMPFLG** and will automatically transform the stress and strain values to the global coordinate system.
When D3PLOT detects a model with orthotropic material properties a new option to plot results using the material axes coordinate system is available.
The method of modelling composites depends on the scope of the simulation. The main techniques include:

**One Shell Element**
- With *PART_COMPOSITE* or *ELEMENT_SHELL_COMPOSITE* all the layers can be defined using a single shell element.
- One integration point available for each layer.
- Can control with MAXINT amount of output in thickness.
- *Cannot* model delamination.

**Layers of Solid Elements**
- Each layer can be modelled separately.
- The connection between layers can be node-to-node, or using _TIEBREAK contact, or by physically modelling adhesive.
- *Can* model delamination.

**Layers of Shell Elements**
- Each layer can be modelled separately.
- The connection between layers can be done using a _TIEBREAK contact, or by physically modelling adhesive.
- *Can* model delamination.
**PART_COMPOSITE:** layers (plies) defined on *PART card.

*PART_COMPOSITE:* one Part per combination of plies
*ELEMENT_SHELL_COMPOSITE: layers (plies) defined on *ELEMENT card.

*ELEMENT_COMPOSITE: one Part per physical component
3. Material Models
The main difference between the LS-DYNA orthotropic and composite materials is that for the composite materials an option for failure of fibres and matrix is available.

The available material model in LS-DYNA for composites are the following:

* `*MAT_COMPOSITE_DAMAGE [*MAT_022]`
* `*MAT_ENHANCED_COMPOSITE_DAMAGE [*MAT_054/55]`
* `*MAT_LAMINATED_COMPOSITE_FABRIC [*MAT_058]`
* `*MAT_COMPOSITE_FAILURE_OPTION [*MAT_059]`
* `*MAT_RATE_SENSITIVE_COMPOSITE_FABRIC [*MAT_158]`
* `*MAT_LAMINATED_FRACTURE_DAIMLER_PINHO [*MAT_261]`
* `*MAT_LAMINATED_FRACTURE_DAIMLER_CAMANHO [*MAT_262]`
## Overview of composite material models

<table>
<thead>
<tr>
<th>Element</th>
<th>Failure criteria</th>
<th>Comments</th>
</tr>
</thead>
</table>
| *MAT_022: COMPOSITE_DAMAGE | Chang-Chang | No fibre compression failure  
Simple brittle model  
No crash front algorithm |
| *MAT_054/55: ENHANCED_COMPOSITE_DAMAGE | 54: Chang-Chang  
55: fibre:Chang-Chang  
matrix: Tsai-Wu | Improvement of MAT_022  
Crash front algorithm  
Minimum stress limit factor |
| *MAT_058: LAMINATED_COMPOSITE_FABRIC | Modified Hashin.  
Three different failure criteria:  
1. multi-surface, 2. smooth failure surface, 3. faceted failure | Smooth stress-strain relation  
Non-linear shear behaviour  
Minimum stress limit factor  
Exponential softening |
| *MAT_059: COMPOSITE_FAILURE_MODEL | Modified Hashin | Similar to MAT_054  
Crash front algorithm  
Minimum stress limit factor |
| *MAT_158: RATE_SENSITIVE_COMPOSITE_FABRIC | Modified Hashin | Same as MAT_058  
Rate sensitive |
| *MAT_261: LAMINATED_FRACTURE_DAIMLER_PINHO | Pinho: Considers the state of the art  
Puck’s criterion for inter-fibre failure | Physical based failure criteria  
Continuum damage model  
Linear softening evolution based on fracture toughness |
| *MAT_262: LAMINATED_FRACTURE_DAIMLER_CAMANHO | Camanho: Considers the state of the art  
Puck’s criterion for inter-fibre failure | Physical based failure criteria  
Continuum damage model  
Bi-linear/linear softening evolution based on fracture toughness |
## Orthotropic Material Strengths

<table>
<thead>
<tr>
<th>Loading Mode</th>
<th>Stress</th>
<th>Strength</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fibre</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tension</td>
<td>$\sigma_{11} &gt; 0$</td>
<td>$X_T$</td>
</tr>
<tr>
<td>Compression</td>
<td>$\sigma_{11} &lt; 0$</td>
<td>$X_C$</td>
</tr>
<tr>
<td><strong>Transverse</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tension</td>
<td>$\sigma_{22} &gt; 0$</td>
<td>$Y_T$</td>
</tr>
<tr>
<td>Compression</td>
<td>$\sigma_{22} &lt; 0$</td>
<td>$Y_C$</td>
</tr>
<tr>
<td><strong>Shear</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>In-Plane</td>
<td>$\sigma_{12} &gt; 0$</td>
<td>$S_{12}$</td>
</tr>
<tr>
<td><strong>Through the thickness (TT)</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tension</td>
<td>$\sigma_{33} &gt; 0$</td>
<td>$Z_T$</td>
</tr>
<tr>
<td>Compression</td>
<td>$\sigma_{33} &lt; 0$</td>
<td>$Z_C$</td>
</tr>
<tr>
<td><strong>Solid elements only</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TT Transverse</td>
<td>$\sigma_{23} &gt; 0$</td>
<td>$S_{23}$</td>
</tr>
<tr>
<td>TT Longitudinal</td>
<td>$\sigma_{13} &gt; 0$</td>
<td>$S_{13}$</td>
</tr>
</tbody>
</table>

[Source: Kawashita, Bristol, 2017]
**Features**

- Enhanced version of *MAT_022
  - depicts a more sudden failure after exceeding the strength limits of the ply
- Chang-Chang failure criteria
- Elasto-plastic stress-strain relationship
- Parameters to control the element erosion in a more numerical manner
  - **DFAILT** for limiting the strain in fibre tension
  - **DFAILC** for limiting the strain of fibre compression
  - **DFAILM** for limiting the stress of matrix in tension and compression
  - **DFAILS** for the limiting the strain in shear
  - **SLIMxx** for reducing the stress level beyond the strength limit
  - **EFS** defines an effective strain to control element erosion
  - **TFAIL** for element timestep criteria for element erosion
- Crash front algorithm – **SOFT**
- 2-way fibre action (thin shells only) – **2WAY**
- Valid for thin/thick shells and solid elements
- Strain rate dependent strengths (**LCxx** – Load curve ID)
### Failure Mode

<table>
<thead>
<tr>
<th>Mode</th>
<th>Formula</th>
<th>Degradation after failure:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile Fibre Mode</td>
<td>Failure assumed when $e_f^2 &gt; 0$</td>
<td>$E_{11} = E_{22} = G_{12} = \nu_{12} = \nu_{21} = 0$</td>
</tr>
<tr>
<td>Compressive Fibre Mode</td>
<td>Failure assumed when $e_c^2 &gt; 0$</td>
<td>$E_{11} = \nu_{12} = \nu_{21} = 0$</td>
</tr>
<tr>
<td>Tensile Matrix Mode</td>
<td>Failure assumed when $e_m^2 &gt; 0$</td>
<td>$E_{22} = G_{12} = \nu_{21} = \nu_{12} = 0$</td>
</tr>
<tr>
<td>Compressive Matrix Mode</td>
<td>Failure assumed when $e_d^2 &gt; 0$</td>
<td>$E_{22} = G_{12} = \nu_{21} = \nu_{12} = 0$</td>
</tr>
</tbody>
</table>
## *MAT_054: ENHANCED_COMPOSITE_DAMAGE*

### Composites Webinar

<table>
<thead>
<tr>
<th>Card 1</th>
<th>Card 2</th>
<th>Card 3</th>
<th>Card 4</th>
<th>Card 5</th>
<th>Card 6</th>
<th>Card 7</th>
<th>Card 8</th>
<th>Card 9</th>
</tr>
</thead>
<tbody>
<tr>
<td>MID</td>
<td>GAB</td>
<td>GAB</td>
<td>V1</td>
<td>TFAIL</td>
<td>X</td>
<td>PFL</td>
<td>SLIMT1</td>
<td>LCXC</td>
</tr>
<tr>
<td>RO</td>
<td>GBC</td>
<td>GBC</td>
<td>V2</td>
<td>ALPH</td>
<td>T</td>
<td>EPSF</td>
<td>SLIMC2</td>
<td>LCXT</td>
</tr>
<tr>
<td>EA</td>
<td>GCA</td>
<td>GCA</td>
<td>V3</td>
<td>SOFT</td>
<td>Y</td>
<td>EPSR</td>
<td>SLIMT2</td>
<td>LCYX</td>
</tr>
<tr>
<td>EB</td>
<td>(EC)</td>
<td>(KF)</td>
<td>A1</td>
<td>FBRT</td>
<td>YT</td>
<td>TSMD</td>
<td>SLIMC2</td>
<td>LCYC</td>
</tr>
<tr>
<td>(EC)</td>
<td>PRBA</td>
<td>AOPT</td>
<td>A2</td>
<td>YCFAC</td>
<td>SC</td>
<td>SOFT2</td>
<td>SLIMS</td>
<td>LCSC</td>
</tr>
<tr>
<td>PRBA</td>
<td>PRCA</td>
<td>PRCB</td>
<td>A3</td>
<td>DFAILT</td>
<td>CRIT</td>
<td></td>
<td>NCYREF</td>
<td>DT</td>
</tr>
<tr>
<td>PRCA</td>
<td></td>
<td></td>
<td>MANGLE</td>
<td>DFAILC</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PRCB</td>
<td></td>
<td></td>
<td></td>
<td>EFS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Elastic, orthotropic parameters
- **RO**: Density
- **EA**: Young’s modulus in longitudinal direction (a)
- **EB**: Young’s modulus in transverse direction (b)
- **(EC)**: Young’s modulus in normal direction (c) – not used
- **PRBA**: Poisson’s ratio in ba plane
- **PRCA**: Poisson’s ratio in ca plane
- **PRCB**: Poisson’s ratio in cb plane
- **GAB**: Shear modulus in the ab plane
- **GBC**: Shear modulus in the bc plane
- **GCA**: Shear modulus in the ca plane

### Material coordinate system
- **AOPT**: Material axes option
  - **XP, YP, ZP**: Components of vector a for AOPT=1
  - **A1, A2, A3**: Components of vector a for AOPT=2
  - **V1, V2, V3**: Components of vector v for AOPT=3
  - **D1, D2, D3**: Components of vector d for AOPT=2
  - **BETA**: Material angle in degrees for AOPT=0 & 3

### Failure modelling parameters
- **TFAIL**: Failure modulus values
- **ALPH**: Failure strain
- **SOFT**: Softening parameter
- **FBRT**: Failure strain
- **YCFAC**: Failure strain
- **DFAILM**: Failure strain
- **DFAILS**: Failure strain
- **TSMD**: Softening parameter
- **SC**: Softening parameter
- **CRIT**: Critical parameter
- **BETA**: Material angle

### Parameters related to material coordinate definition
- **Card 1**: MID, RO, EA, EB, (EC), PRBA, PRCA, PRCB
- **Card 2**: GAB, GBC, GCA, (KF), AOPT
- **Card 3**: A1, A2, A3, MANGLE
- **Card 4**: D1, D2, D3, DFAILM, DFAILS
- **Card 5**: TFAIL, ALPH, SOFT, FBRT, YCFAC, DFAILT, DFAILC, EFS
- **Card 6**: X, XT, Y, YT, SC, CRIT, BETA
- **Card 7**: PFL, EPSF, EPSR, TSMD, SOFT2
- **Card 8**: SLIMT1, SLIMC2, SLIMT2, SLIMC2, SLIMS, NCYREF, SOFTG
- **Card 9**: LCXC, LCXT, LCYX, LCYC, LCSC, DT

Parameters related to elastic, orthotropic material properties

---

**Slide 29**
# *MAT_054: ENHANCED_COMPOSITE_DAMAGE*

- **Elastic, orthotropic parameters**
  - Card 1: MID, RO, EA, EB, (EC), PRBA, PRCA, PRCB
  - Card 2: GAB, GBC, GCA, (KF), AOPT
  - Card 3: A1, A2, A3, MANGLE
  - Card 4: V1, V2, V3, D1, D2, D3, DFAILM, DFAILS
  - Card 5: TFAIL, ALPH, SOFT, FBRT, YCFAC, DFAILT, DFAILC, EFS
  - Card 6: XC, XT, YC, YT, SC, CRIT, BETA
  - Card 7: PFL, EPSF, EPSR, TSMD, SOFT2
  - Card 8: SLIMT1, SLIMC2, SLIMT2, SLIMC2, SLIMS, NCYREF, SOFTG
  - Card 9: LCXC, LCXT, LCYX, LCYC, LCSC, DT

### Parameters related to elastic, orthotropic material properties
- **XC**: Longitudinal compressive strength
- **XT**: Longitudinal tensile strength
- **YC**: Transverse compressive strength
- **YT**: Transverse tensile strength
- **SC**: Shear strength
- **CRIT**: Failure criterion (54/55)
- **BETA**: Weighting factor for shear term in tensile fibre mode
- **ALPH**: Shear stress parameter for the non-linear term

### Parameters related to element deletion
- **DFAILM**: Maximum strain for matrix straining in tension or compression
- **DFAILS**: Maximum tensorial shear strain ($\varepsilon_{12} = \gamma_{12}/2$)
- **DFAILT**: Maximum strain for fibre tension
- **DFAILC**: Maximum strain for fibre compression
- **EFS**: Effective failure strain
- **TFAIL**: Time step size criteria for element deletion
- **2WAY**: Flag to turn on 2-way fibre action
- **TI**: Flag for transversal isotropic behaviour (Solids)

### Parameters related to post-matrix compressive failure
- **FBRT**: Softening for fibre tensile strength
- **YCFAC**: Reduction factor for compressive fibre strength

### Parameters related to crashfront
- **SOFT**: Softening reduction factor for material strength in crashfront elements
- **PFL**: Percentage of layers that must fail until crashfront
- **SOFT2**: Optional ‘orthogonal’ softening reduction factor
- **SOFTG**: Softening reduction factor for transverse stiffness

---

Slide 30
**MAT_054: ENHANCED_COMPOSITE_DAMAGE**

Composites Webinar

1-Element Test, Single Layer (Shell, ELFORM = 16)

Fibre Direction

Transverse Direction

In-plane Shear Direction

- XT = 1.5
- Perfectly plastic until DFAILT = 0.1
- XC = 1.0

- YT = 180
- YC = 240

DFAILxx = 0.1
EA = 141 GPa
EB = 9.34 GPa
GAB = 4.5 GPa

S = 150

**Slide 31**
**MAT_054: ENHANCED_COMPOSITE_DAMAGE**

Composites Webinar

---

**Fibre Direction**

- $X_T = 1.5$
- $X_C = 1.0$
- $SLIM1*XT$
- $SLIMC1*XC$

**Transverse Direction**

- $Y_T = 180$
- $Y_C = 240$
- $SLIM2*YT$
- $SLIMC2*YC$

**In-plane Shear Direction**

- $SC = 150$
- $SLIMS*SC$

---

DFAIL_xx = 0.1
EA = 141 GPa
EB = 9.34 GPa
GAB = 4.5 Gpa
SLIM_xx = 0.5
**Extra History Variables**

- Information about the status of each integration point and each element can be plotted using the 'Extra history variables' available for this material, as showed in the table below.
- These extra variables will be output to the .ptf files if the **NEIPS** (Shell) /**NEIPH** (Solid) is set in the ***DATABASE_EXTENT_BINARY**

<table>
<thead>
<tr>
<th>SHELLS</th>
<th>SOLIDS</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1 ef</td>
<td>#1 ef</td>
</tr>
<tr>
<td>#2 ec</td>
<td>#2 ec</td>
</tr>
<tr>
<td>#3 em</td>
<td>#3 em</td>
</tr>
<tr>
<td>#4 ed</td>
<td>#4 ed</td>
</tr>
<tr>
<td>#5 efail</td>
<td>#5 efail</td>
</tr>
<tr>
<td>#6 dam</td>
<td>#6 dam</td>
</tr>
<tr>
<td>#8 q1</td>
<td>#15 εaa</td>
</tr>
<tr>
<td>#9 q2</td>
<td>#16 εbb</td>
</tr>
<tr>
<td>#10 εaa</td>
<td>#17 εab</td>
</tr>
<tr>
<td>#11 εbb</td>
<td></td>
</tr>
<tr>
<td>#12 εab</td>
<td></td>
</tr>
<tr>
<td>#16 d_{56}</td>
<td>Transverse shear damage</td>
</tr>
</tbody>
</table>

**SHELLS**

- #1 ef: Failure flag tensile fibre mode
- #2 ec: Failure flag compressive fibre mode
- #3 em: Failure flag tensile matrix mode
- #4 ed: Failure flag compressive matrix mode
- #5 efail: Integration point status
- #6 dam: Damage parameter (SOFT)
- #8 q1: Material direction cosine (alpha)
- #9 q2: Material direction sine (alpha)
- #10 εaa: Local strain a-direction
- #11 εbb: Local strain b-direction
- #12 εab: Local shear strain
- #16 d_{56}: Transverse shear damage

**SOLIDS**

- #1 ef: Failure flag tensile fibre mode
- #2 ec: Failure flag compressive fibre mode
- #3 em: Failure flag tensile matrix mode
- #4 ed: Failure flag compressive matrix mode
- #5 efail: Integration point status
- #6 dam: Damage parameter (SOFT)
- #15 εaa: Local strain a-direction
- #16 εbb: Local strain b-direction
- #17 εab: Local shear strain

**Legend**

- ef, ec, em, ed, efail: 1-intact; 0-failed
- dam: -1-intact; 0-failed
- d_{56}: 0-intact; 1-full damaged
Support for extra variables has been enhanced in D3PLOT v15.

1. New “Material Properties” category with descriptive component names for extra variables, organised by material and element type if applicable.
D3PLOT requires additional information to support “Material Properties” contouring:

1. A ZTF file is required to provide the link between parts and materials.

   ```
   $ Material type code,material name (including the leading '*'),data slot (intg pt),variable name, short name (alias),element type applicability (3 - shell/1 - solid/2 - both)
   MAT_002,*MATORTHOTROPICELASTIC,1,C12_STIFFNESS_COMP,C12 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,2,C13_STIFFNESS_COMP,C13 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,3,C14_STIFFNESS_COMP,C14 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,4,C22_STIFFNESS_COMP,C22 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,5,C23_STIFFNESS_COMP,C13 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,6,C24_STIFFNESS_COMP,C24 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,7,C33_STIFFNESS_COMP,C33 Stiffness,3
   MAT_002,*MATORTHOTROPICELASTIC,8,C34_STIFFNESS_COMP,C34 Stiffness,3
   ```

2. A new pre-populated CSV file (mat_prop.csv) provides material extra variable information – this can be edited by the user to allow new materials to be added or existing component names to be modified.
4. Delaminations
Delamination is a critical failure mechanism in highly loaded laminated fibre-reinforced composites. Caused by high interlaminar stresses in conjunction with the typically very low through-thickness strength due to:

- Out-of-plane loading
- Curved or tapered geometries
- Discontinuities owing to cracks
- Ply-drops or free-edges

Olsson et al. 2012
Skin-stiffener debonding under 4-point bend loading

[Clegg et al., Evaluation of the effects of tufting on the performance of composite T-joints, ECCM17, 2016]
Matrix dominated failure: delamination is a fracture process.

**Cohesive Zone Model (CZM):** describes the separation of crack surfaces, which takes place across an extended crack tip or cohesive zone, and is resisted by cohesive tractions.

**Cohesive Zone Length (CZL)** is the length of the process zone ahead of the crack tip.

The fracture process is ‘lumped’ along the length of the **CZL** where a softening ‘traction separation’ law exists.
Traction-Separation Law

Double Cantilever Beam (DCB) Mode I

Mixed-Mode Bend (MMB) Mode I/II

End-Notched Flexure (ENF) Mode II

\[ \eta = 2.284 \]

Double Cantilever Beam (DCB)

Mode I

Mixed-Mode Bend (MMB)

Mode I/II

End-Notched Flexure (ENF)

Mode II

\[ \Delta \]

\[ a_0 \]

\[ 2h \]

\[ L \]

\[ P \]
Fracture toughness vs. Mode-mixity

\[ \eta = 2.284 \]

**Experimental**

**B-K law**
Inter-laminar Material Modelling in LS-DYNA

LS-DYNA Option 1: Cohesive Zone Elements
- Cohesive element
- *MAT_COHESIVE...
- Finite or zero thickness elements
- Composite shells, tshells or solid elements allowed

LS-DYNA Option 2: Tiebreak Contacts
- Tiebreak contacts
- *CONTACT_ONE WAY_SURFACE_TO_SURFACE_TIEBREAK
- Composite shells, tshells or solid elements allowed

LS-DYNA Option 3: Volume Elements.
*MAT_ARUP_ADHESIVE for structural bonding
Cohesive Zone Elements

- **Cohesive elements** typically formulated in terms of a **traction vs. relative displacement** relationship instead of the traditional stress vs. strain relation.
- Traction on the mid-surface are functions of the relative displacements between nodal pairs 1-5, 2-6, 3-7, and 4-8, interpolated to the four (2x2) in-plane integration gauss points.
- **INTFAIL**: Number of integration points required for element deletion.
8-node Type 19 Elements

- Connect to volume elements (moment are not transferred)
- ‘zero thickness’ elements supported
  - ROFLG = 1, density per unit area
- Critical time-step governed by the characteristic stiffness and mass of the element rather than characteristic length

8-node Type 20 Elements

- Identical to Type 19 but with offsets for use with shells
- Cohesive element is assumed to be centred between two layers of shells
- Offset distances for both shells are one half the initial thicknesses of the nodal pairs (1-5, 2-6, 3-7, and 4-8) separating the two shells.
- Moments applied to shells = force x offset
- If used with tied contacts, use TIED_SHELL_EDGE_TO_SURFACE
Displacements at an integration point are:

\[ \Delta x = \sum_{i=1}^{4} N_i(s,t)\Delta x_{i+4,i} \]

\[ \Delta X = \sum_{i=1}^{4} N_i(s,t)\Delta X_{i+4,i} \]

- \( R(t) \): transformation matrix from local to global coordinate system at time \( t \);
- \( X, x \): the initial and current coordinates, respectively.

Integrate tractions over the mid-surface and rotate back to global system:

\[ F_i = R(t) \sum_{g=1}^{4} T_g N_i(s_g, t_g) \det(J_g), \text{ for } 1 \leq i \leq 4 \]
### Cohesive Material Models: Overview

<table>
<thead>
<tr>
<th>LS-DYNA Material Model</th>
<th>Traction-Separation Law/Shape</th>
<th>Initiation &amp; Propagation Criteria</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MAT_138:</strong> COHESIVE_MIXED_MODE</td>
<td>Bilinear (linear softening)</td>
<td>Quadratic mixed-mode. Supports power law ($XMU &gt; 0$) and B-K law ($XMU &lt; 0$)</td>
<td>Simplification of <em>MAT_COHESIVE_GENERAL.</em></td>
</tr>
<tr>
<td><strong>MAT_184:</strong> COHESIVE_ELASTIC</td>
<td>Linear</td>
<td>Stress based criteria governed by normal and/or tangential strengths</td>
<td>Simple cohesive elastic model</td>
</tr>
<tr>
<td><strong>MAT_185:</strong> COHESIVE_ELASTIC_TH</td>
<td>Tri-linear</td>
<td>Quadratic mixed-mode governed by dimensionless separation parameter $\lambda$ (i.e. traction drops to zero when $\lambda = 1$)</td>
<td>Cohesive model by Tvergaard and Hutchinson. Same loading and unloading path; completely reversible</td>
</tr>
<tr>
<td><strong>MAT_186:</strong> COHESIVE_GENERAL</td>
<td>Arbitrary normalised traction-separation law given by a load curve (TSLC)</td>
<td>Three general irreversible mixed-mode interaction cohesive: 1. TES = 0: Power-law ($XMU$) 2. TES = 1.0: B-K law ($XMU$) 3. TES = 2.0: Dimensionless separation parameter</td>
<td></td>
</tr>
<tr>
<td><strong>MAT_240:</strong> COHESIVE_MIXED_MODE ELASTO_PLASTIC_RATE</td>
<td>Tri-linear</td>
<td>Quadratic yield and damage criterion in mixed-mode loading. Damage evolution is governed by a power-law</td>
<td>Rate-dependent, elastic-ideal plastic</td>
</tr>
</tbody>
</table>
MAT_COHESIVE_MIXED_MODE: MAT_138

- Simplification of *MAT_COHESIVE_GENERAL
- Restricted to linear softening
- Bilinear traction-separation law with quadratic mixed mode delamination criterion and a damage formulation
- Can be used only with cohesive element formulations

The total mixed-mode relative displacement is defined as:

\[ \delta_m = \sqrt{\delta_I^2 + \delta_{II}^2} \]

The mixed-mode damage initiation displacement is defined as:

\[ \delta_I^0 = T / EN \]
\[ \delta_{II}^0 = S / ET \]
\[ \beta = \frac{\delta_{II}}{\delta_I} \]

Mode I  Mode II

\[ \delta_I = \delta_3 \]
\[ \delta_{II} = \sqrt{\delta_I^2 + \delta_2^2} \]

Composites Webinar
MAT_COHESIVE_MIXED_MODE: MAT_138

- Simplification of *MAT_COHESIVE_GENERAL
- Restricted to linear softening
- Bilinear traction-separation law with quadratic mixed mode delamination criterion and a damage formulation
- Can be used only with cohesive element formulations

**Formulation 1:** Power law \( (XMU > 0) \):

- Ultimate mixed-mode displacement \( \delta^F \) (total failure):

\[
\delta^F = \frac{2(1 + \beta^2)}{\delta^0} \left[ \left( \frac{EN}{GIC} \right)^{XMU} + \left( \frac{ET}{GIIC} \right)^{XMU} \right]^{-\frac{1}{XMU}}
\]

\[
\left( \frac{GI}{GIC} \right)^{\alpha} + \left( \frac{GII}{GIIC} \right)^{\alpha} = 1
\]

**Formulation 2:** Benzegagh-Kenane \( (XMU < 0) \):

\[
\delta^F = \frac{2}{\delta^0 \left[ \frac{1}{1 + \beta^2 EN} + \frac{\beta^2}{1 + \beta^2 ET} \right]} \left[ GIC + (GIIC - GIC) \left( \frac{\beta^2 \cdot ET}{EN + \beta^2 \cdot ET} \right)^{XMU} \right]^{XMU} \]

\[
GC = GIC + (GIIC - GIC) \left( \frac{GII}{GI + GII} \right)^{\eta}
\]
<table>
<thead>
<tr>
<th>Card 1</th>
<th>Card 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>MID</td>
<td>XMU</td>
</tr>
<tr>
<td>RO</td>
<td>T</td>
</tr>
<tr>
<td>ROFLG</td>
<td>S</td>
</tr>
<tr>
<td>INTFAIL</td>
<td>UND</td>
</tr>
<tr>
<td>EN</td>
<td>UTD</td>
</tr>
<tr>
<td>ET</td>
<td></td>
</tr>
<tr>
<td>GIC</td>
<td></td>
</tr>
<tr>
<td>GIIC</td>
<td></td>
</tr>
</tbody>
</table>

**MID:** Material identification.

**RO:** Mass density

**ROFLG:** Flag for whether density is specified per unit area or volume

**INTFAIL:** The number of integration points required for the cohesive element to be deleted

**EN:** Stiffness normal to the plane of the cohesive element

**ET:** Stiffness in the plane of the cohesive element

**GIC:** Fracture toughness / energy release rate for mode I

**GIIC:** Fracture toughness / energy release rate for mode II

**XMU:** Exponent for mixed mode criterion

**T:** Peak traction in normal direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

**S:** Peak traction in tangential direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

**UND:** Ultimate displacement in the normal direction

**UTD:** Ultimate displacement in the tangential direction
Double Cantilever Beam: Mode I

Explicit Analysis: Quasi-Static

- Comparison of element formulations

### Table

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Card 1</strong></td>
<td>MID</td>
<td>RO</td>
<td>ROFLG</td>
<td>INTFAIL</td>
<td>EN</td>
<td>ET</td>
<td>GIC</td>
<td>GIIC</td>
</tr>
<tr>
<td></td>
<td>138</td>
<td>1.0E-5</td>
<td>0.0</td>
<td>0.0</td>
<td>1E+5</td>
<td>1E+5</td>
<td>0.969</td>
<td>1.719</td>
</tr>
<tr>
<td><strong>Card 2</strong></td>
<td>XMU</td>
<td>T</td>
<td>S</td>
<td>UND</td>
<td>UTD</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>-2.284</td>
<td>30.0</td>
<td>60.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

**Figure:**
- **SHELL** (ELFORM = 16)
- **SOLID** (ELFORM = 2)
- **TSHELL** (ELFORM = 2)
Double Cantilever Beam: Mode I

Composites Webinar

Implicit Non-linear Analysis

- Explicit vs implicit non-linear static analysis

SOLID (ELFORM = 2)

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>MID</td>
<td>RO</td>
<td>ROFLG</td>
<td>INTFAIL</td>
<td>EN</td>
<td>ET</td>
<td>GIC</td>
<td>GIIC</td>
</tr>
<tr>
<td></td>
<td>138</td>
<td>1.0E-5</td>
<td>0.0</td>
<td>0.0</td>
<td>1E+5</td>
<td>1E+5</td>
<td>0.969</td>
<td>1.719</td>
</tr>
<tr>
<td>Card 2</td>
<td>XMU</td>
<td>T</td>
<td>S</td>
<td>UND</td>
<td>UTD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>-2.284</td>
<td>30.0</td>
<td>60.0</td>
<td>-</td>
<td>-</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Mixed-Mode Bend: Mode I/II

Composites Webinar

Graph showing force (N) vs. displacement (mm) for different mixed-mode bends with GII/GT values of 20%, 50%, and 80%.

- MMB - 20% (Implicit)
- MMB - 50% (Implicit)
- MMB - 80% (Implicit)

Lines indicate max load for each mode as follows:
- Max Load - Beam Theory (MMB 80%)
- Max Load - Beam Theory (MMB 50%)
- Max Load - Beam Theory (MMB 20%)
3-Point End Notched Flexure: Mode II

S = 60 MPa

First row of crack front element enter the damage regime

Damage flag prior to crack propagation (PS = 1)

Crack propagation (PS = 1)

\[ P \quad L \]

\[ a_0 \]

Base

\[ \text{Corrected Beam Theory} \quad \text{Crack Propagation} \quad \text{ENF - 0.5mm} \]
Mesh Dependency – Strength

Element size < 2.0mm:
- Stable crack propagation

Element size > 2.0mm
- Clear transition from stable to unstable crack propagation, resulting in non-physical spurious oscillations
- Significant reduction in the number of elements spanning the fracture process zone as the element size increases
- Stable crack growth characterised by the gradual loading of elements ahead of the crack front

\[ S = 60 \text{ MPa} \]
Current mesh design guidelines recommend at least three elements within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.

Engineering solution for mesh size effects

$$l_{cz} = ME \frac{G_c}{(\sigma_{\text{max}})^2}$$

Young’s modulus of the material

Critical energy release rate

Cohesive zone length

Scaling factor

Maximum strength

$$N_e = \frac{l_{cz}}{l_e}$$

Number of elements in FPZ

Element size

$$\sigma_{\text{max}} = \sqrt{\frac{MEG_c}{N_ek_e}}$$

(Turon et al. 2007)
Mesh Dependency – Strength

Current mesh design guidelines recommend at least three elements within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>MID</td>
<td>RO</td>
<td>ROFLG</td>
<td>INTFAIL</td>
<td>EN</td>
<td>ET</td>
</tr>
<tr>
<td>138</td>
<td>1.0E-5</td>
<td>0.0</td>
<td>0.0</td>
<td>1E+5</td>
<td>1E+5</td>
<td>0.969</td>
</tr>
<tr>
<td>Card 2</td>
<td>XMU</td>
<td>T</td>
<td>S</td>
<td>UND</td>
<td>UTD</td>
<td></td>
</tr>
<tr>
<td>-2.284</td>
<td>-30.0</td>
<td>-60.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**T:** Peak traction in normal direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

**S:** Peak traction in tangential direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

Mesh Dependency – Strength

Current mesh design guidelines recommend at least three elements within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>MID</td>
<td>RO</td>
<td>ROFLG</td>
<td>INTFAIL</td>
<td>EN</td>
<td>ET</td>
</tr>
<tr>
<td>138</td>
<td>1.0E-5</td>
<td>0.0</td>
<td>0.0</td>
<td>1E+5</td>
<td>1E+5</td>
<td>0.969</td>
</tr>
<tr>
<td>Card 2</td>
<td>XMU</td>
<td>T</td>
<td>S</td>
<td>UND</td>
<td>UTD</td>
<td></td>
</tr>
<tr>
<td>-2.284</td>
<td>-30.0</td>
<td>-60.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**T:** Peak traction in normal direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

**S:** Peak traction in tangential direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

Mesh Dependency – Strength

Current mesh design guidelines recommend at least three elements within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>MID</td>
<td>RO</td>
<td>ROFLG</td>
<td>INTFAIL</td>
<td>EN</td>
<td>ET</td>
</tr>
<tr>
<td>138</td>
<td>1.0E-5</td>
<td>0.0</td>
<td>0.0</td>
<td>1E+5</td>
<td>1E+5</td>
<td>0.969</td>
</tr>
<tr>
<td>Card 2</td>
<td>XMU</td>
<td>T</td>
<td>S</td>
<td>UND</td>
<td>UTD</td>
<td></td>
</tr>
<tr>
<td>-2.284</td>
<td>-30.0</td>
<td>-60.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Mesh Dependency – Strength

Current mesh design guidelines recommend at least three elements within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.

<table>
<thead>
<tr>
<th>Card 1</th>
<th>MID</th>
<th>RO</th>
<th>ROFLG</th>
<th>INTFAIL</th>
<th>EN</th>
<th>ET</th>
<th>GIC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>138</td>
<td>1.0E-5</td>
<td>0.0</td>
<td>0.0</td>
<td>1E+5</td>
<td>1E+5</td>
<td>0.969</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Card 2</th>
<th>XMU</th>
<th>T</th>
<th>S</th>
<th>UND</th>
<th>UTD</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-2.284</td>
<td>-30.0</td>
<td>-60.0</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
Current mesh design guidelines recommend at least three elements within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.
Tiebreak Contacts

- TIEBREAK contacts allow the modelling of interlaminar debonding, which transmits both compressive and tensile forces with optional failure criteria.
- TIEBREAK contacts are penalty based. TIED contacts are constraint based.
- Analogous to cohesive elements:

  Analogous to cohesive elements:

  Three components of relative displacements between slave node and master segment:

  \[
  \delta = \begin{bmatrix}
  \delta_1 \\
  \delta_2 \\
  \delta_3 
  \end{bmatrix}
  \]

  \[
  \begin{align*}
  \delta_1 &= \text{Shear} \\
  \delta_2 &= \text{Normal}
  \end{align*}
  \]

  Nodal force output = Contact stresses \( \times \) area of the segment.

- **Segment:**

  3-noded or 4-noded connectivity references a shell element or one of the faces of a solid element.
Tiebreak Contacts

- To detect a node-segment pair, the projection of the node onto the master segment, along the master segment normal, must lie within the segment area.
- Segment area is scaled by an additional 2% to collect nodes that lie near edges.
- Contact point is calculated just once (at initialisation t=0).
- **Projection distance** is the orthogonal distance of the slave node from the master segment along the master segment normal.
- Projection distance is computed in a local coordinate system embedded in the master segment.

- $d < 0$: 
  - Penetration
- $d > 0$: 
  - Zero penetration or positive offset
- $d = 0$: 
  - Save node is on the master segment surface
Recommended Tiebreak Contacts

- Non-Automatic
  - *CONTACT_TIEBREAK_NODES_TO_SURFACE
  - *CONTACT_TIEBREAK_NODES_ONLY

- Automatic
  - *CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK

- Automatic TIEBREAKS recommended:
  - Allows stress based failure criterion
  - Automatic shell normal orientation
  - Shell thickness offsets considered in post-failure interaction
  - Damage options based on _OPTION parameter.
### Tiebreak Contacts

**CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_OPTION**

<table>
<thead>
<tr>
<th>Card 1</th>
<th>Card 2</th>
<th>Card 3</th>
<th>Card 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>SSID</td>
<td>FS</td>
<td>SFS</td>
<td>OPTION</td>
</tr>
<tr>
<td>MSID</td>
<td>FD</td>
<td>SFM</td>
<td></td>
</tr>
<tr>
<td>SSTYP</td>
<td>DC</td>
<td>SST</td>
<td></td>
</tr>
<tr>
<td>MSTYP</td>
<td>VC</td>
<td>MST</td>
<td></td>
</tr>
<tr>
<td>SBOXID</td>
<td>VDC</td>
<td>SFST</td>
<td></td>
</tr>
<tr>
<td>MBOXID</td>
<td>PENCHK</td>
<td>SFMT</td>
<td></td>
</tr>
<tr>
<td>SPR</td>
<td>BT</td>
<td>FSF</td>
<td></td>
</tr>
<tr>
<td>MPR</td>
<td>DT</td>
<td>VSF</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **OPTION 9/11** follows same damage initiation and evolution criteria as ***MAT_COHESIVE_MIXED_MODE (*MAT_138)**

- 14 options available with up to 7 parameters
- Some options apply to offset shell elements (moments are calculated between shell elements)

---

**-TIEBREAK <option>**

-3: As option 3, but moments are transferred. SMP only
-2: As option 2, but moments are transferred. SMP only
-1: As option 1, but moments are transferred. SMP only
1: Nodes making contact stick permanently, no tangential movement
2: Nodes initially in contact stick, can fail, no tangential movement
3: Nodes making contact stick, can fail, no tangential movement
4: Nodes initially in contact stick, tangential movement permitted
5: Nodes initially in contact stick, damage is func<crack width>
6: Nodes initially in contact stick, failure based on <param>
7: Dycoss Discrete Crack Model
8: As option 6, but applies to offset shell elements
9: As option 7, but with power law and B-K damage
10: As option 7, but applies to offset shell elements
11: As option 9, but applies to offset shell elements
Tiebreak Contacts

*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_OPTION

<table>
<thead>
<tr>
<th>Card 1</th>
<th>SSID</th>
<th>MSID</th>
<th>SSTYP</th>
<th>MSTYP</th>
<th>SBOXID</th>
<th>MBOXID</th>
<th>SPR</th>
<th>MPR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 2</td>
<td>FS</td>
<td>FD</td>
<td>DC</td>
<td>VC</td>
<td>VDC</td>
<td>PENCHK</td>
<td>BT</td>
<td>DT</td>
</tr>
<tr>
<td>Card 3</td>
<td>SFS</td>
<td>SFM</td>
<td>SST</td>
<td>MST</td>
<td>SFST</td>
<td>SFMT</td>
<td>FSF</td>
<td>VSF</td>
</tr>
<tr>
<td>Card 4</td>
<td>OPTION</td>
<td>NFLS</td>
<td>SFLS</td>
<td>PARAM</td>
<td>ERATEN</td>
<td>ERATES</td>
<td>CT2CN</td>
<td>CN</td>
</tr>
</tbody>
</table>

- **OPTION = 9/11 ~ *MAT_138 (*MAT_COHESIVE_MIXED_MODE)
  - **OPTION 9**: For use between solids and thick shells only
  - **OPTION 11**: Moments are calculated between shell elements
  - Only option will fully mixed-mode behaviour

NFLS  Normal failure stress
SFLS  Shear failure stress
PARAM Exponent in the damage model
ERATEN Normal energy release rate
ERATES Shear energy release rate
CT2CN Ratio of tangential to normal stiffness
CN Normal stiffness
Tiebreak Contacts

Double Cantilever Beam - Mode I

- Corrected Beam Theory
- Crack Propagation
- Solid (Explicit)
- Shell (Explicit)
- Tshell (Explicit)

3-Point End Notched Flexure - Mode II

- Corrected Beam Theory
- Crack Propagation
- Solid (Explicit)
## Tiebreak Contact-Related Outputs

- **Binary output file**
  - *DATABASE_BINARY_INTFOR* – contact forces and stresses (can be used for fringe plotting)
    - Set print flag(s) on card 1 of *CONTACT_SPR =1 and MPR=1*
    - Include *s=filename* on execution line
  - Binary file can be read by Oasys D3PLOT or LS-Prepost

- **DATABASE_OPTION_ID**

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>CID</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Card 2</td>
<td>SSID</td>
<td>MSID</td>
<td>SSTYP</td>
<td>SBOXID</td>
<td>MBOXID</td>
<td>SPR</td>
<td>MPR</td>
</tr>
</tbody>
</table>

- **DATABASE_NCFORC**

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>DT</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **DATABASE_BINARY_INTFOR**

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card 1</td>
<td>DT</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Slave/Master side information is written**

- **Text file**
  - CID – Contact surface ID
  - Contact nodes and pressure printed

- **Binary output file s=filename**
  - Contact force
  - Contact pressure
  - Contact gap
Fringe Plot: Cohesive Elements vs. Tiebreaks

\[ \delta = 2\text{mm} \]
\[ \delta = 3.6\text{mm} \]
\[ \delta = 4.7\text{mm} \]

**Cohesive** Element Model  
(Plastic Strain Flag)

**TIEBREAK** Contact Model  
*DATABASE_BINARY_INTFOR*
### Comparison of Methods*

<table>
<thead>
<tr>
<th><strong>Cohesive Elements</strong></th>
<th><strong>Pre-processing</strong></th>
<th><strong>Constitutive models</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Element numbering and orientation, Coincident nodes or tied-contact mesh</td>
<td></td>
<td>Elasto-damage, mixed-mode</td>
</tr>
<tr>
<td>Elasto-damage, mixed-mode, user-defined material models (umatXXc in dyn21b.f)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Higher resolution: more smooth results</td>
<td></td>
<td>Lower resolution: More coarse results</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Tiebreak Contacts</strong></th>
<th><strong>Post-failure response</strong></th>
<th><strong>Post-processing</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Automatically</td>
<td>Only nodal values, Intfor: Contact gap</td>
</tr>
</tbody>
</table>

*Liebold et al., *DYNAmore, 2018*
Other recommendations

- Highly recommended to invoke invariant node numbering by setting **INN = 2** (shells), **INN = 3** (solids) or **INN = 4** (shells/thick shells/solids) in the ***CONTROL_ACCURACY** card
  - Material coordinate system is automatically updated following the rotation of the element coordinate system
  - Orientation of the material coordinate system is very sensitive to in-plane shearing deformation and zero-energy hourglass modes
- ***CONTROL_SHELL** – Laminated Shell Theory (**LAMSHT**)  
  - **LAMSHT** corrects for the incorrect assumption of uniform constant shear strain through the thickness of the shell  
  - Particularly important if composite shells has layers of dissimilar materials (sandwich panel)
- Compressive modes of failure (stability):
  - ***CONTROL_BULK_VISCOSITY**, TYPE = 2
  - **DATABASE_EXTENT_BINARY**
    - **MAXINT** = number of through-thickness IPs
### Composites Webinar

#### Outline – FE Modelling of Composites

<table>
<thead>
<tr>
<th>Intra-laminar behaviour</th>
<th>Inter-laminar behaviour</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Element</strong></td>
<td><strong>Material</strong></td>
</tr>
<tr>
<td><img src="image" alt="Shells" /></td>
<td><em>MAT_022: COMPOSITE DAMAGE</em></td>
</tr>
<tr>
<td>Shells</td>
<td><em>MAT_054/055: ENHANCED_COMPOSITE_DAMAGE</em></td>
</tr>
<tr>
<td>(2D stress state)</td>
<td><em>MAT_058/158: LAMINATED_COMPOSITE_FABRIC/RATE_SENSITIVE</em></td>
</tr>
<tr>
<td><img src="image" alt="Thick shells" /></td>
<td><em>MAT_059: COMPOSITE.FAILURE_MODEL</em></td>
</tr>
<tr>
<td>Thick shells</td>
<td><em>MAT_261: LAMINATED_FRACTURE_DAIMLER_PINHO</em></td>
</tr>
<tr>
<td>(2D or 3D stress state)</td>
<td><em>MAT_262: LAMINATED_FRACTURE_DAIMLER_CAMANHO</em></td>
</tr>
<tr>
<td><img src="image" alt="Solids" /></td>
<td><em>MAT_161/162: COMPOSITE_MSC (Commercial)</em></td>
</tr>
<tr>
<td>Solids</td>
<td><em>MAT_138: COHESIVE_MIXED_MODE</em></td>
</tr>
<tr>
<td>(3D stress state)</td>
<td><em>MAT_184: COHESIVE_ELASTIC</em></td>
</tr>
<tr>
<td></td>
<td><em>MAT_185: COHESIVE_TH</em></td>
</tr>
<tr>
<td></td>
<td><em>MAT_186: COHESIVE_GENERAL</em></td>
</tr>
<tr>
<td></td>
<td><em>MAT_240: COHESIVE_MIXED_MODE_ELASTO_PLASTIC_RATE</em></td>
</tr>
<tr>
<td></td>
<td><em>CONTACT_TIEBREAK: AUTOMATIC.ONE WAY_SURFACE_TO_SURFACE</em></td>
</tr>
<tr>
<td></td>
<td><em>MAT_169: MAT_ARUP_ADHESIVE</em></td>
</tr>
</tbody>
</table>
Contact Information

For more information please contact the following:

**UK:**
**Arup**
The Arup Campus
Blythe Valley Park
Solihull, West Midlands
B90 8AE
UK
T +44 (0)121 213 3399
F +44 (0)121 213 3302
dyna.support@arup.com

**China:**
**Arup**
39/F-41/F Huai Hai Plaza
Huai Hai Road (M)
Shanghai
China 200031
T +86 21 6126 2875
F +86 21 6126 2882
china.support@arup.com

**India:**
**nHance Engineering Solutions Pvt. Ltd (Arup)**
Ananth Info Park
HiTec City
Madhapur
Hyderabad - 500081
India
T +91 (0) 40 44369797 / 8
india.support@arup.com

www.arup.com/dyna