Introduction to Composites Modelling in LS-DYNA

The Arup Campus, Blythe Gate, Blythe Valley Park, Solihull, West Midlands, B90 8AE

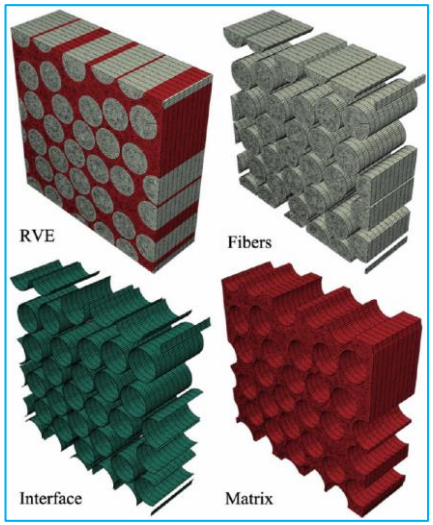
tel: +44 (0) 121 213 3399

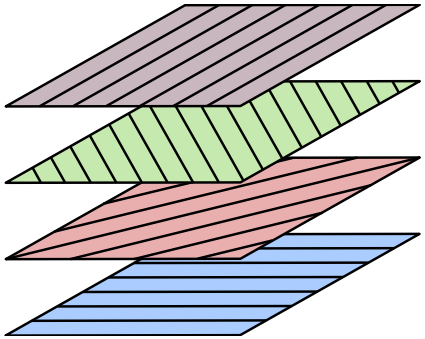
email: dyna.support@arup.com

1. Introduction

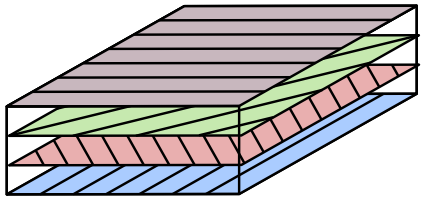
Modelling across the length scales

Micro-scale	Meso-scale: Single Ply	Meso-scale: Laminate	Macro-scale



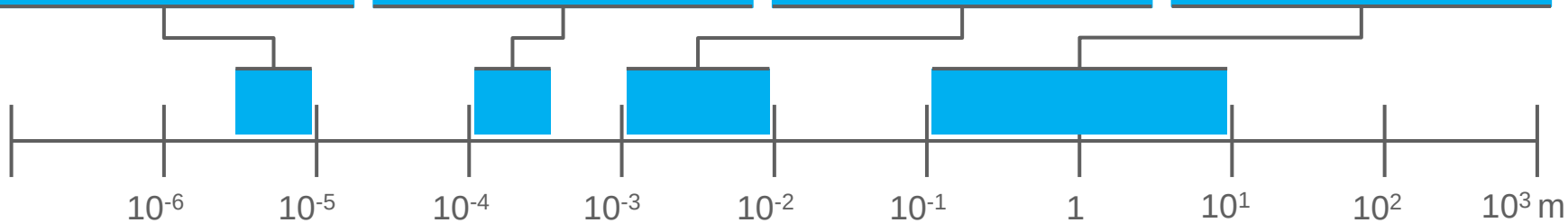


90/±45/0



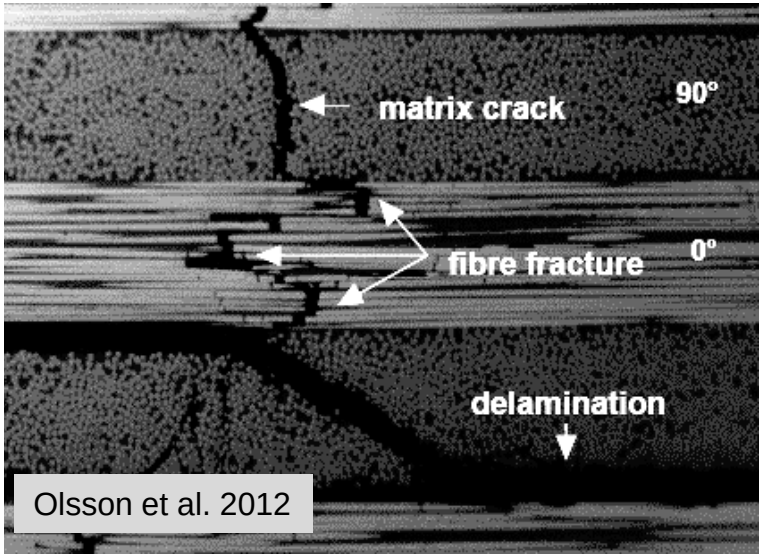
90/±45/0

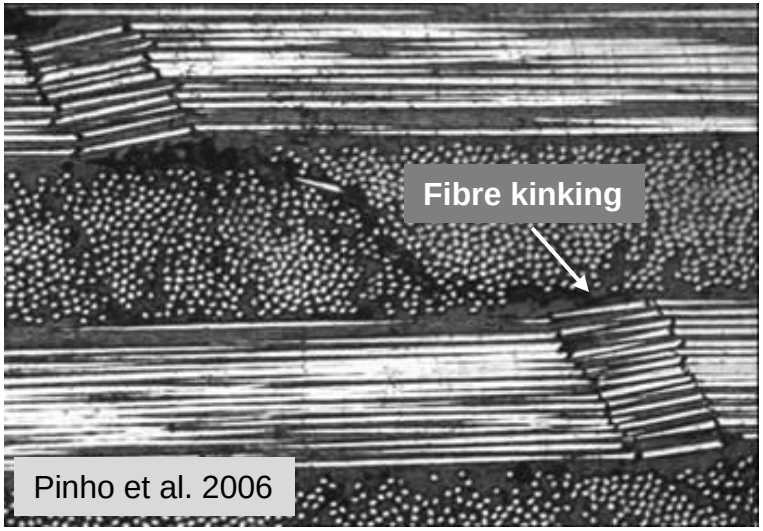
Individual fibres + matrix + Interface[*]	Material homogenisation at ply level	Combine several homogenised layers	Assembly of structural elements (incl. BCs, loads)

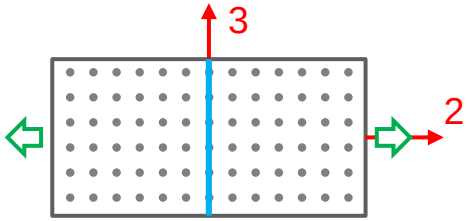


* Yang et al., Micro., Journal of Reinforced Plastic & Composites, 2015

Failure Mechanisms

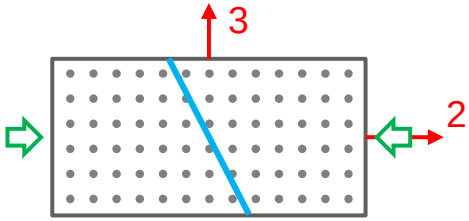






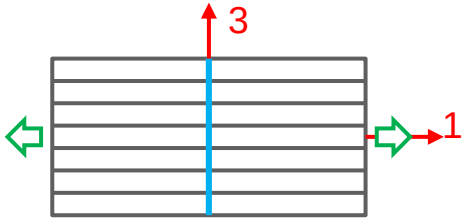
Transverse tensile fracture

$$\sigma_{22} > 0 \qquad Y_T$$



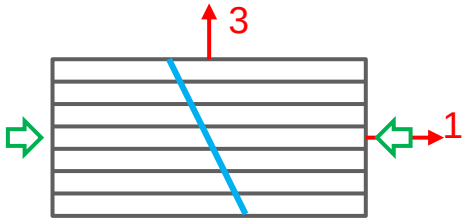
Transverse compressive fracture

$$\sigma_{22} < 0 \qquad Y_C$$



Longitudinal tensile fracture

$$\sigma_{11} > 0 \qquad X_T$$



Longitudinal compressive fracture

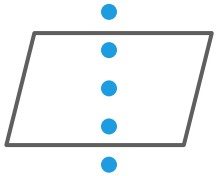
$$\sigma_{11} < 0 \qquad X_C$$

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

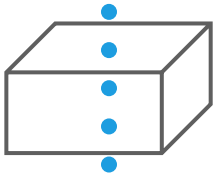
Outline – FE Modelling of Composites

Intra-laminar behaviour

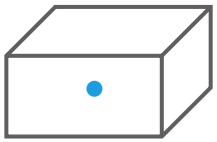
Element



Shells
(2D stress state)



Thick shells
(2D or 3D stress state)



Solids
(3D stress state)

Material

*MAT_022:
COMPOSITE DAMAGE

*MAT_054/055:
ENHANCED_ COMPOSITE DAMAGE

*MAT_058/158:
LAMINATED_COMPOSITE_ FABRIC/RATE_SENSITIVE

*MAT_059:
COMPOSITE_FAILURE_ MODEL

*MAT_261:
LAMINATED_FRACTURE_ DAIMLER_PINHO

*MAT_262:
LAMINATED_FRACTURE_ DAIMLER_CAMANHO

*MAT_161/162:
COMPOSITE_MSC
(Commercial)

Structure

One Shell Layer

Layers of Solid Elements

Layers of Shell Elements

Inter-laminar behaviour

*MAT_138:
COHESIVE_MIXED_MODE

*MAT_184:
COHESIVE_ELASTIC

*MAT_185:
COHESIVE_TH

*MAT_186:
COHESIVE_GENERAL

*MAT_240:
COHESIVE_MIXED_MODE_ ELASTO_PLASTIC_RATE

*CONTACT_TIEBREAK:
AUTOMATIC_ONE_WAY_ SURFACE_TO_SURFACE

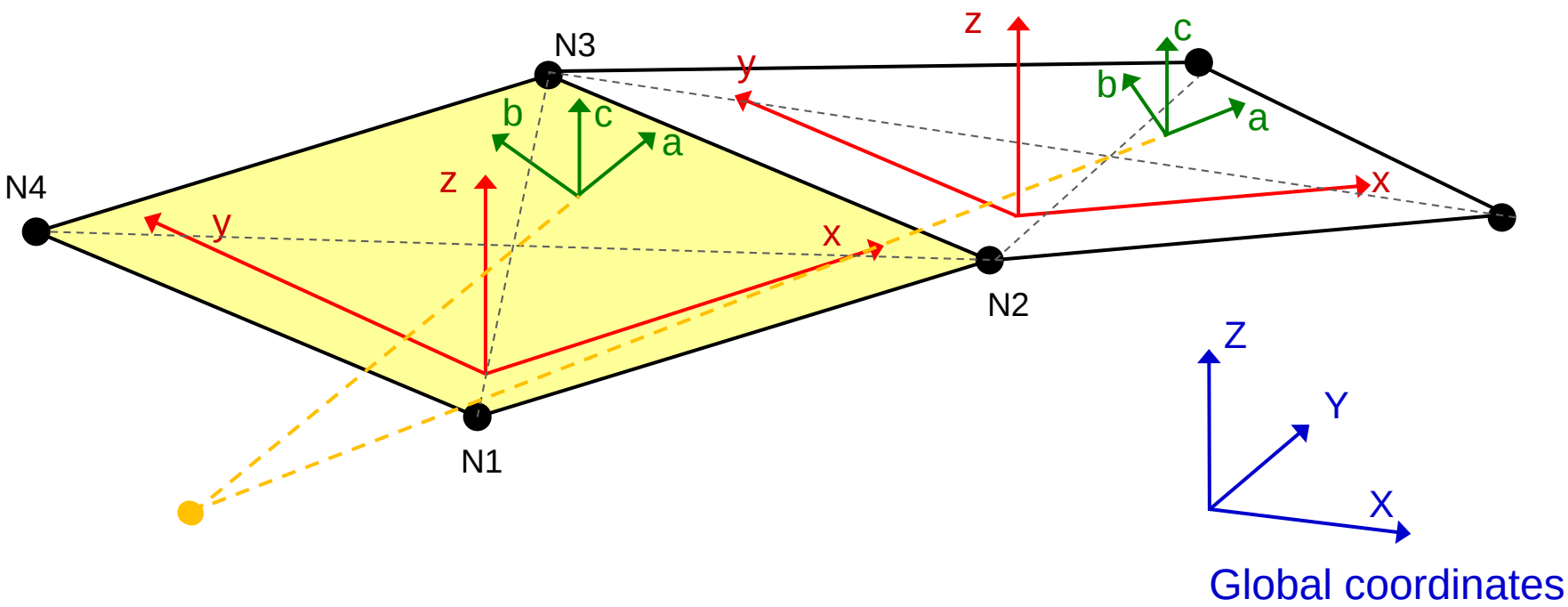
*MAT_169:
MAT_ARUP_ADHESIVE

2. Material & layup definitions

Shell Orientation

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

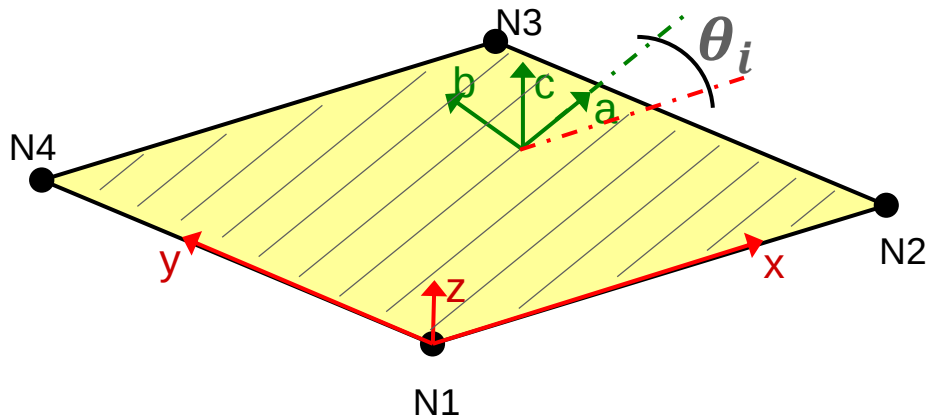
- Global [X,Y,Z]
- Element (local) [x,y,z]
- Material [a,b,c]



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:

- $\boldsymbol{\beta}$: 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)

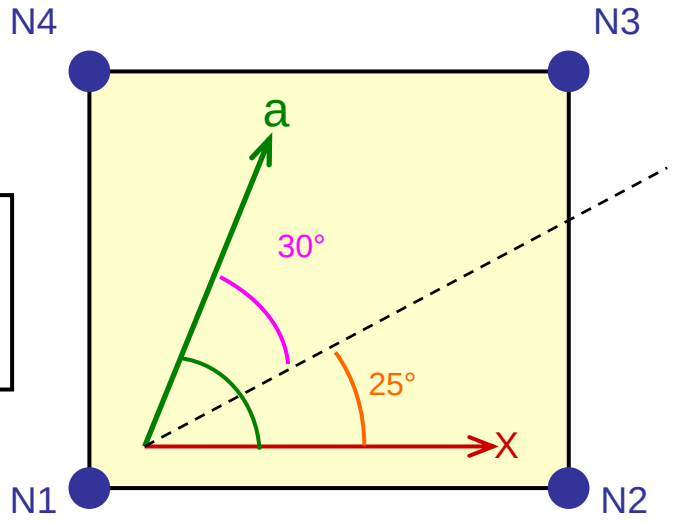
- $\boldsymbol{\beta_i}$: relative angle to define material coordinate system in each layer (IP through-thickness). This is defined using:
  - ***SECTION_SHELL/_TSHELL**
  - ***PART_COMPOSITE(_TSHELL)**
  - ***ELEMENT_SHELL_COMPOSITE**

Example. The total offset (θ_i) of any of the layers in the composite is the sum of the:

'base angle-offset' (β)+ 'layer angle-offset' (β_i)

B1 = 30° ('layer offset', on *PART_COMPOSITE)

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

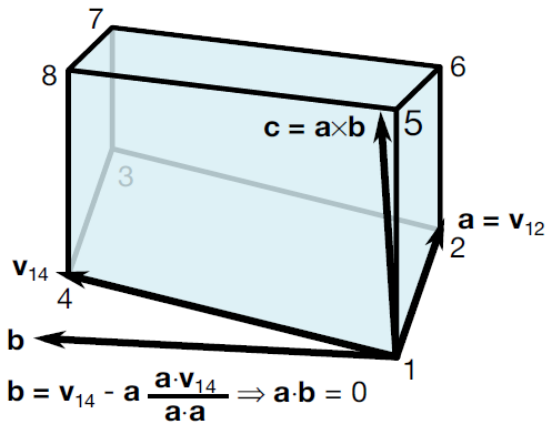


BETA = 25° ('base offset', on *ELEMENT_SHELL_BETA)

Local Coordinate Systems

- Definition of BETA in *MAT-keyword card (a,b,c = 1,2,3)
- See LS-DYNA Keyword User's Manual, *MAT_{OPTION}TROPIC_ELASTIC (*MAT_002)

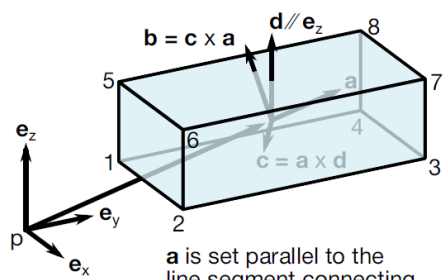
AOPT = 0.0



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**

AOPT = 1.0 (solid only)



a is set parallel to the line segment connecting p to the element center.

d is set parallel to $\mathbf{e}_z$.

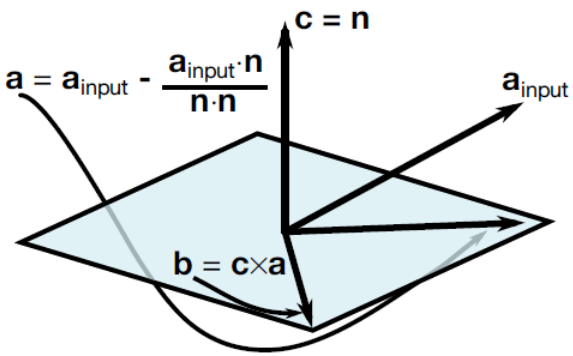
input(p) → {**a**} → {**c**} → {**b**}

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.

Local Coordinate Systems

- Definition of BETA in *MAT-keyword card (a,b,c = 1,2,3)
- See LS-DYNA Keyword User's Manual, *MAT_{OPTION}TROPIC_ELASTIC (*MAT_002)

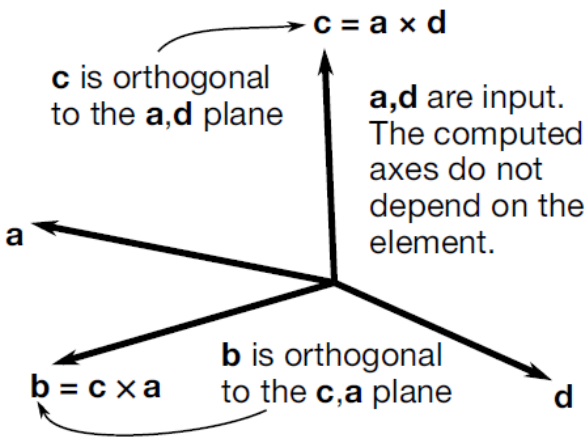
AOPT = 2.0 (shell)



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

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

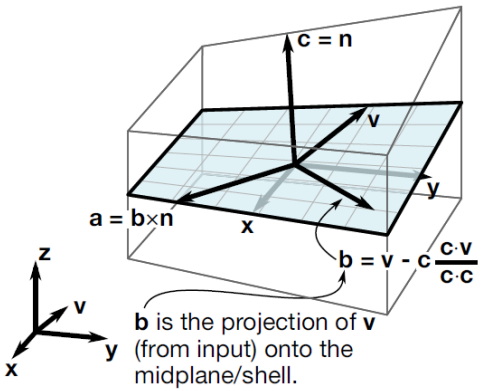
AOPT = 2.0 (solid)



Local Coordinate Systems

- Definition of BETA in *MAT-keyword card (a,b,c = 1,2,3)
- See LS-DYNA Keyword User's Manual, *MAT_{OPTION}TROPIC_ELASTIC (*MAT_002)

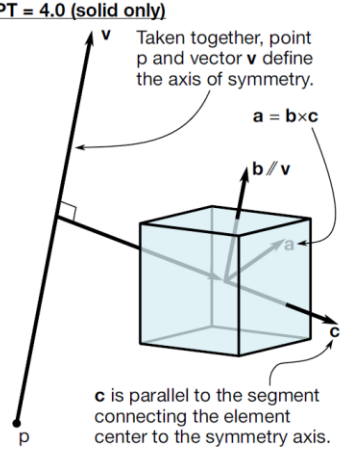
AOPT = 3.0



Define v

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.

AOPT = 4.0 (solid only)



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.

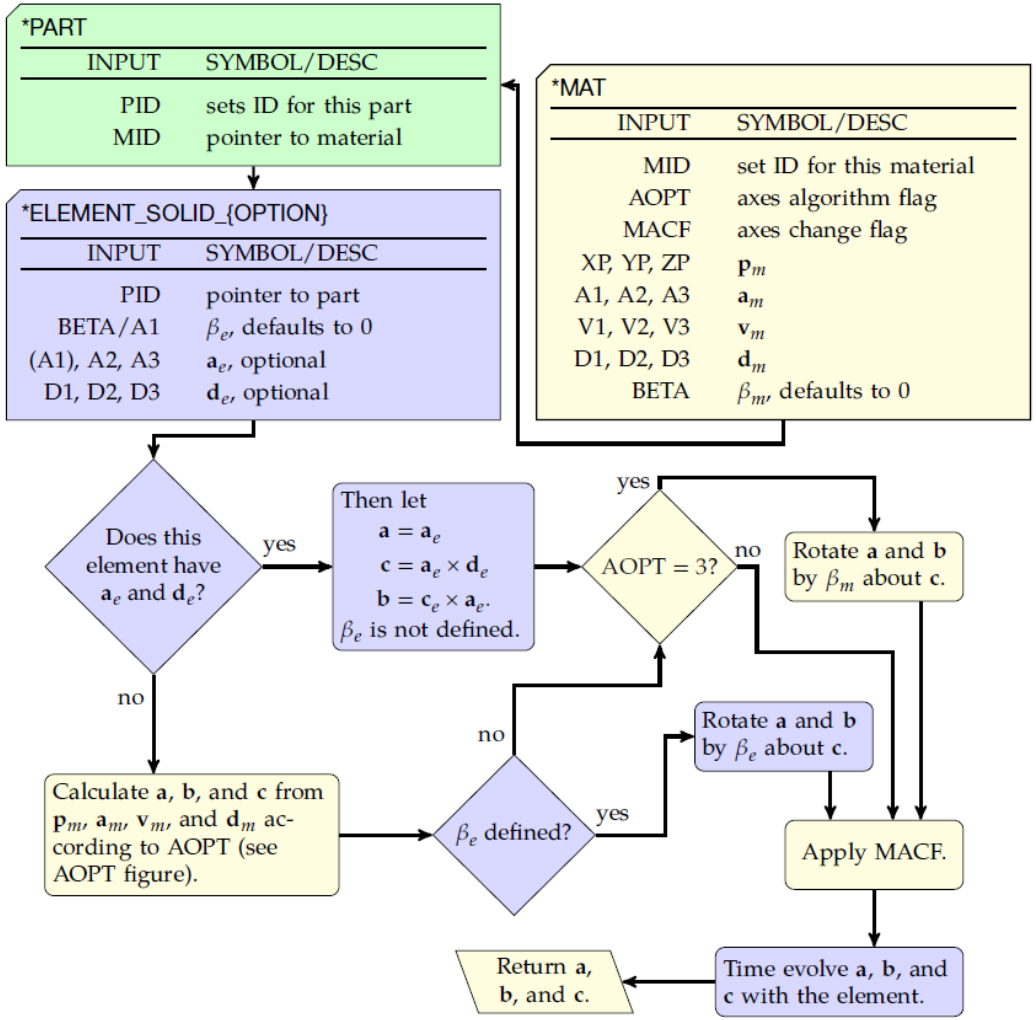


Figure M2-2. Flow chart showing how for each solid element LS-DYNA determines the vectors $\{a, b, c\}$ from the input.

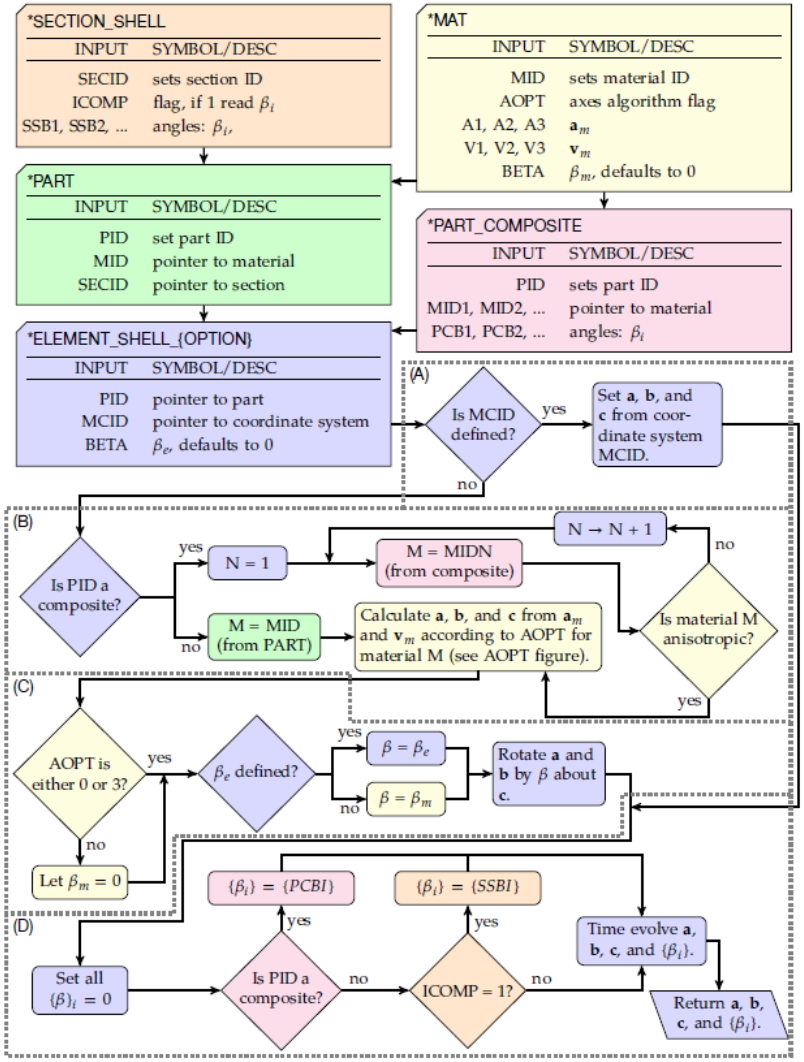
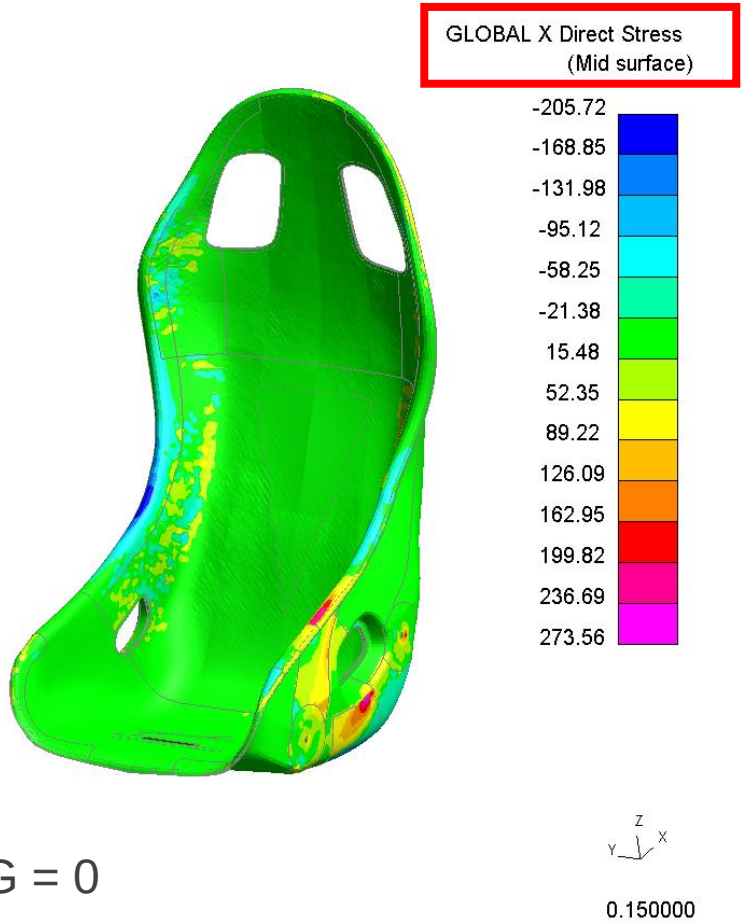


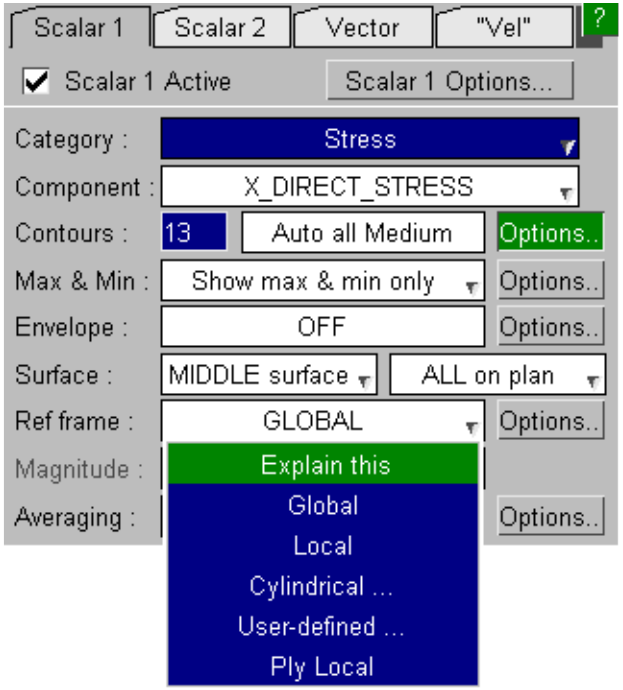
Figure M2-3. Flowchart for shells: (a) check for coordinate system ID; (b) process AOPT; (c) determine β; and (d) for each layer determine β_i.

Material Axes

In LS-DYNA the default output option writes stresses and strains into the PTF (d3plot) binary files using the global coordinate system.



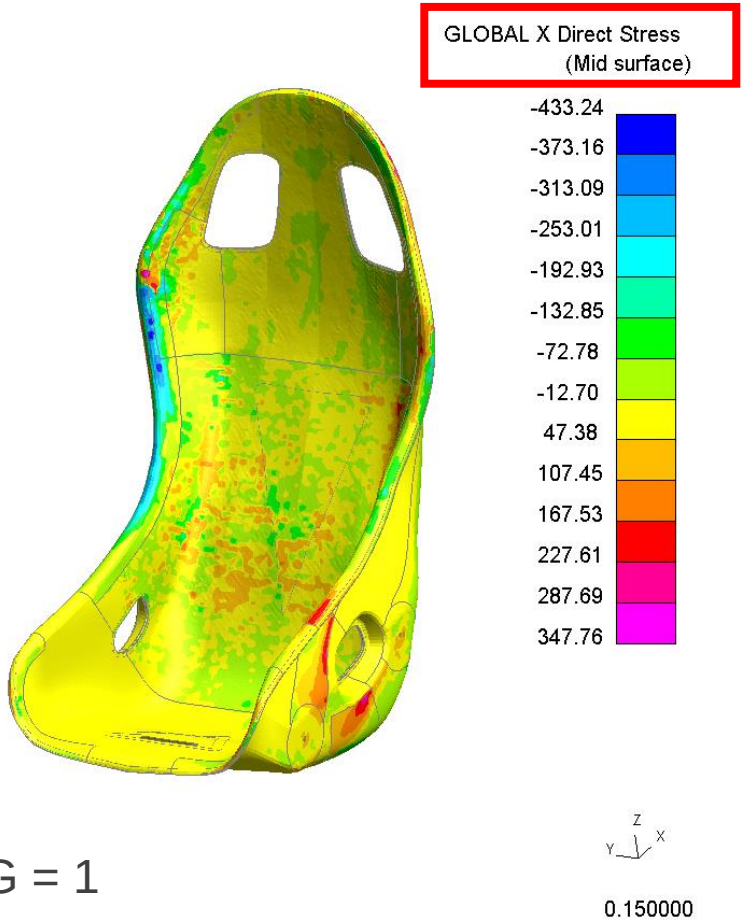
D3PLOT can then transform these results from the global coordinate system into one of the other systems that it supports.



CMPFLG = 0

Material Axes

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



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.

Material Axes

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.

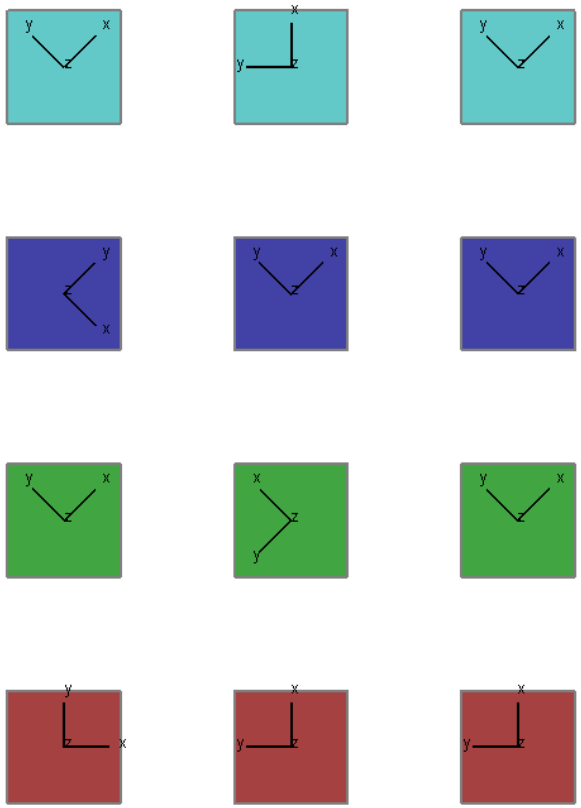
   * For a composite Shell or Thick Shell element this information is needed for all the through thickness integration points.

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.

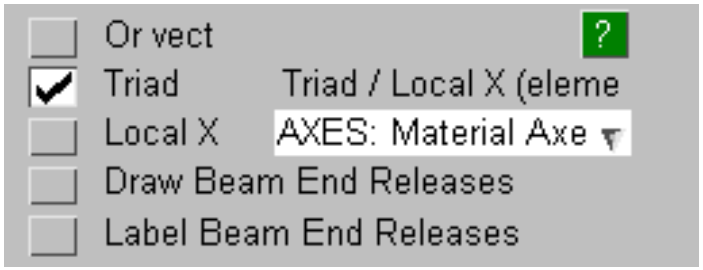
- AOPT on *MAT cards – five options some apply to Shells and some to Solids.
- BETA and COMPOSITE options for ***ELEMENT_SHELL** / ***ELEMENT_TSHELL**.
- ***PART_COMPOSITE** – can vary the material and define orientation angles.
- PART ID on ***INTEGRATION_SHELL** – can vary the material.

Material Axes

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.



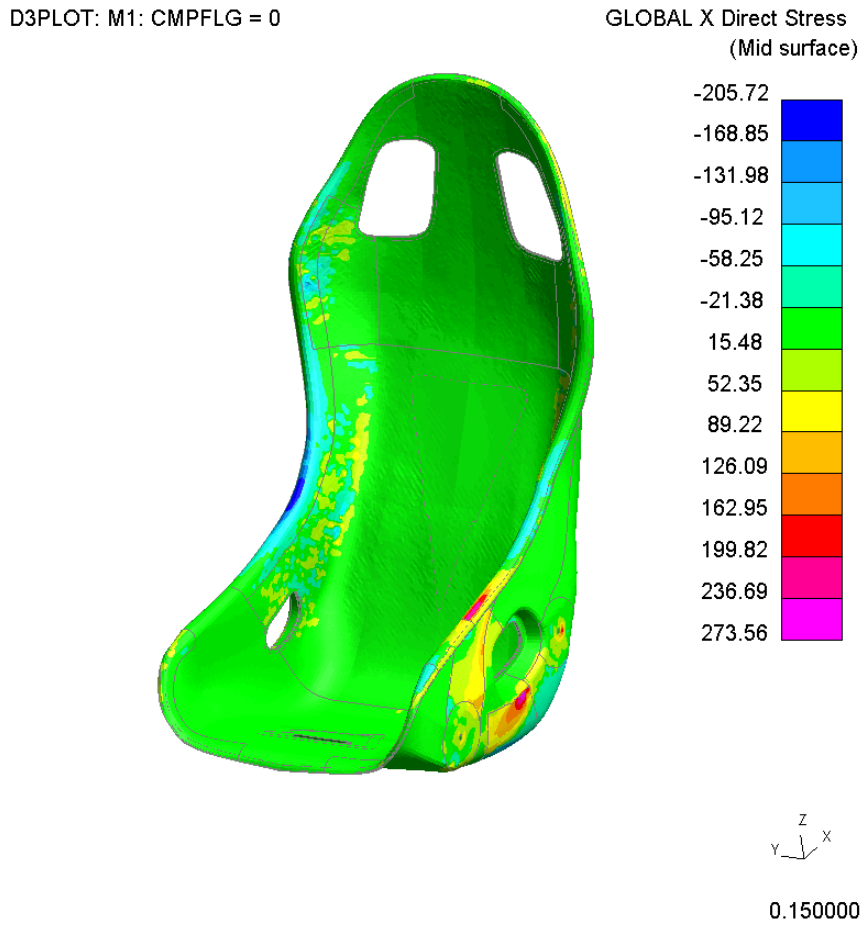
Display - Entities

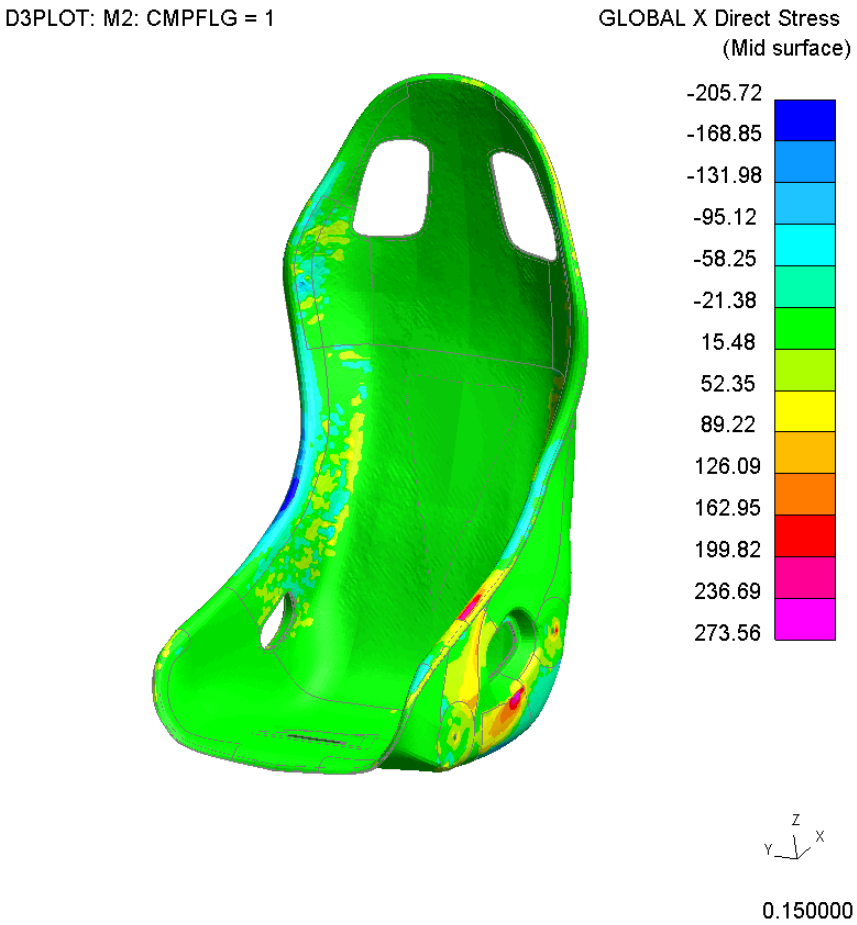


PRIMER v15 writes this information along with the value of **CMPFLG** to the ZTF file.

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

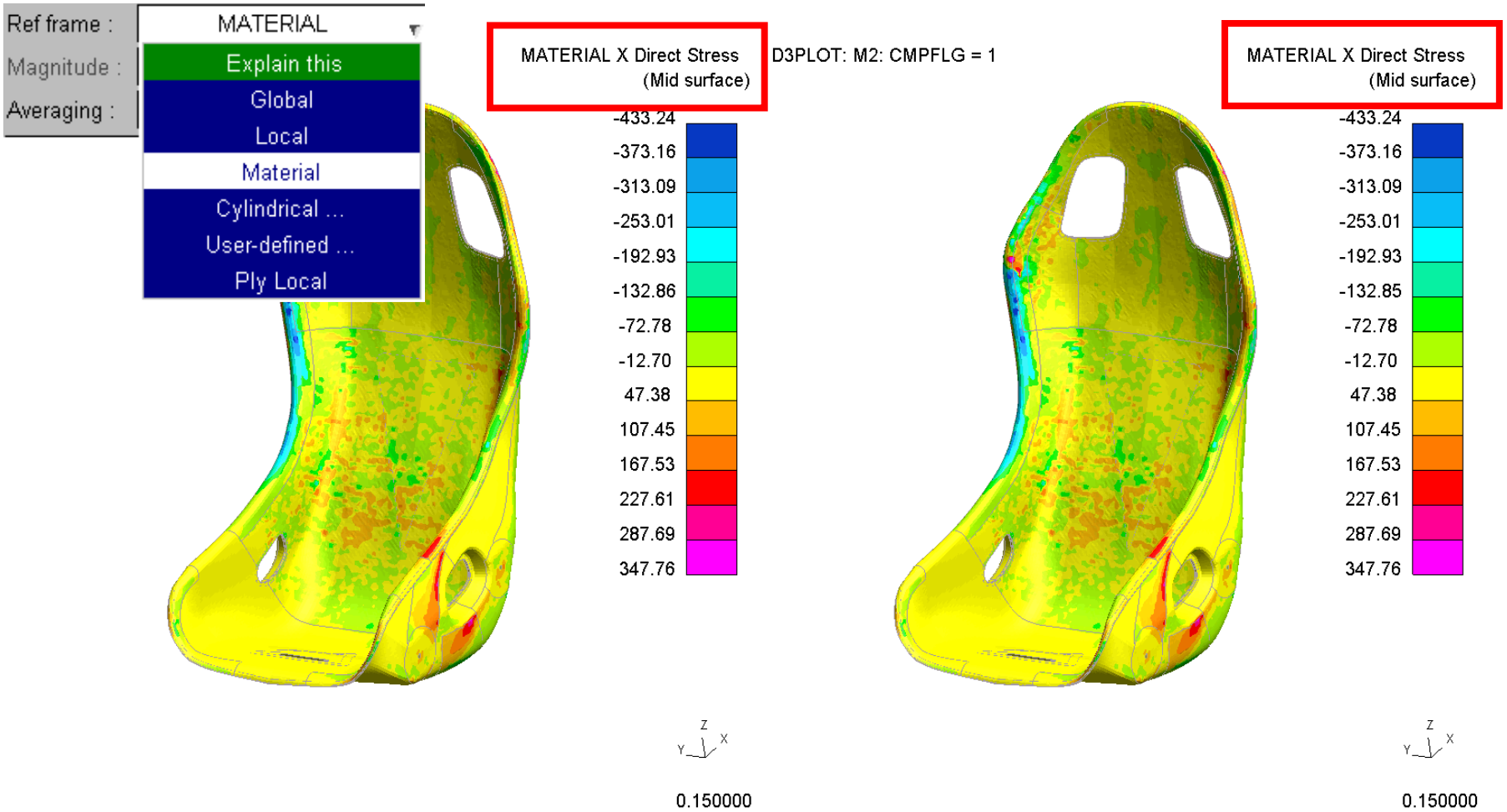
With a ZTF file D3PLOT can correctly identify **CMPFLG** and will automatically transform the stress and strain values to the global coordinate system.





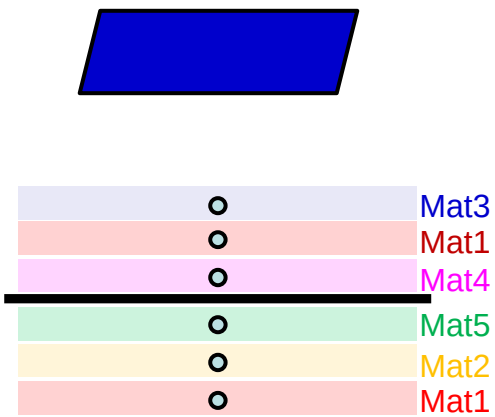
Material Axes

When D3PLOT detects a model with orthotropic material properties a new option to plot results using the material axes coordinate system is available.



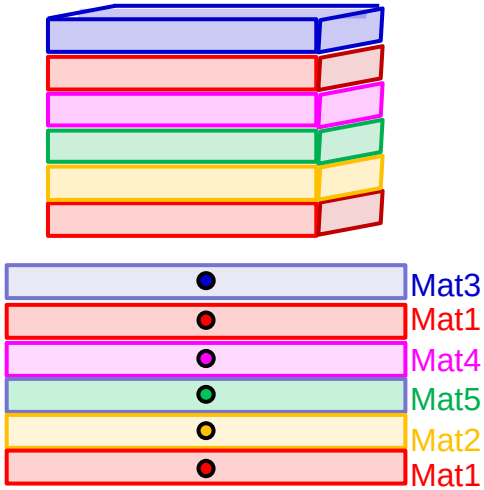
The method of modelling composites depends on the scope of the simulation. The main techniques include:

One Shell Element



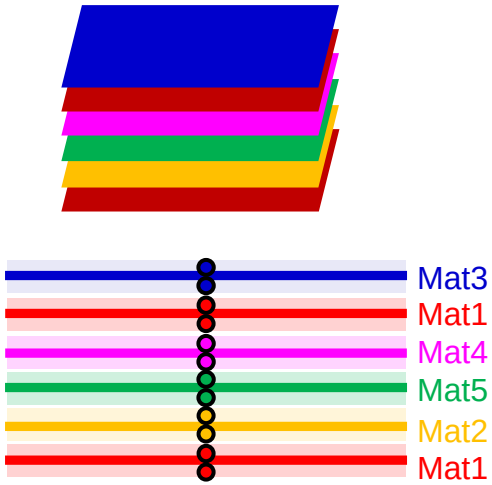
- With *PART_COMPOSITE or *ELEMENT_SHELL_COMPOSITE all the layers can be defined using a single shell element.
- One integration point available for each layer.
- Can control with MAXINT amount of output in thickness.
- *Cannot* model delamination.

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.

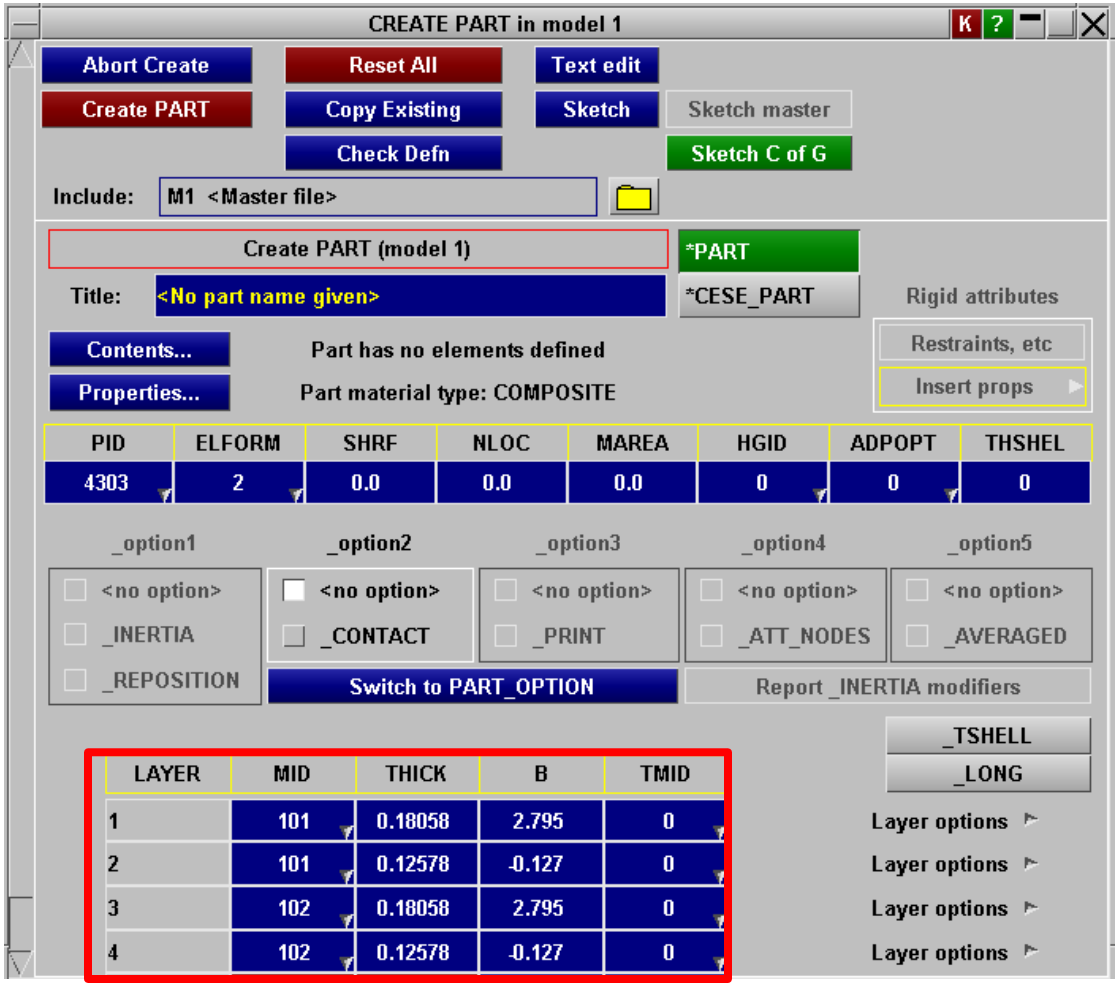
Layers of Shell Elements



- 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.

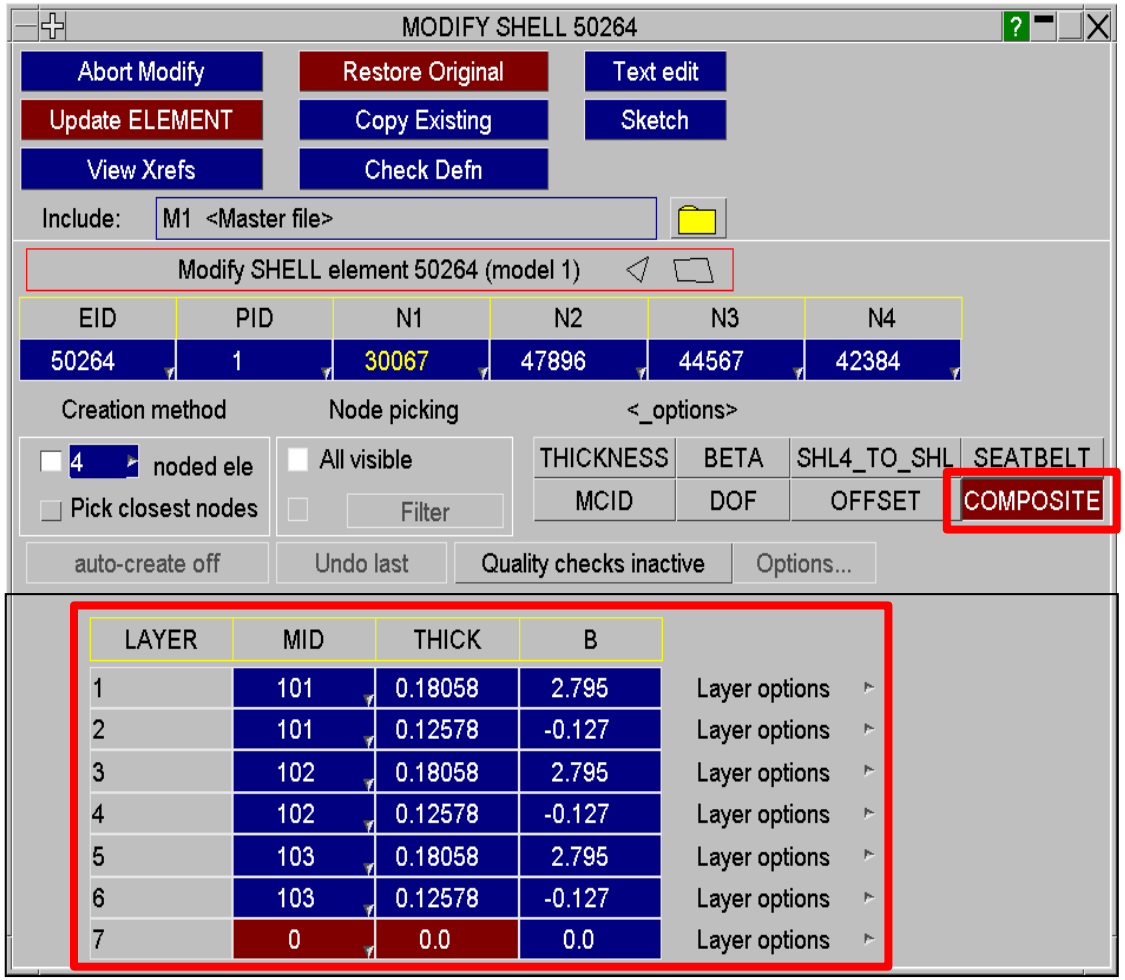
***PART_COMPOSITE:** layers (plies) defined on *PART card.

*PART_COMPOSITE: one Part per combination of plies



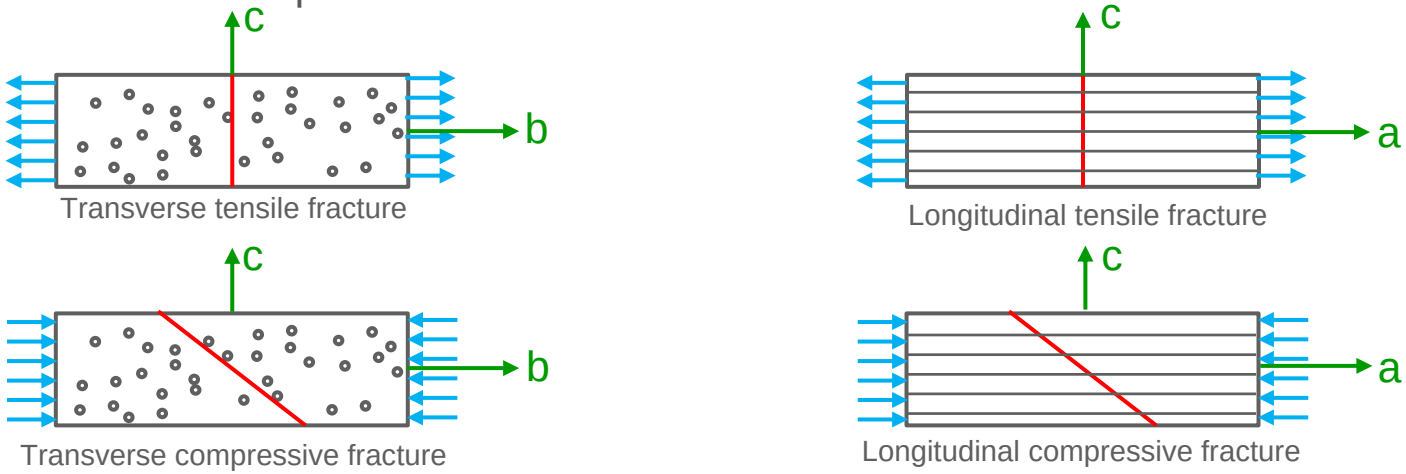
***ELEMENT_SHELL_COMPOSITE:** layers (plies) defined on *ELEMENT card.

*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]

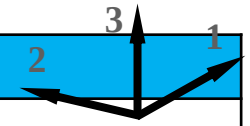
*MAT_LAMINATED_FRACTURE_DAIMLER_PINHO [*MAT_261]

*MAT_LAMINATED_FRACTURE_DAIMLER_CAMANHO [*MAT_262]

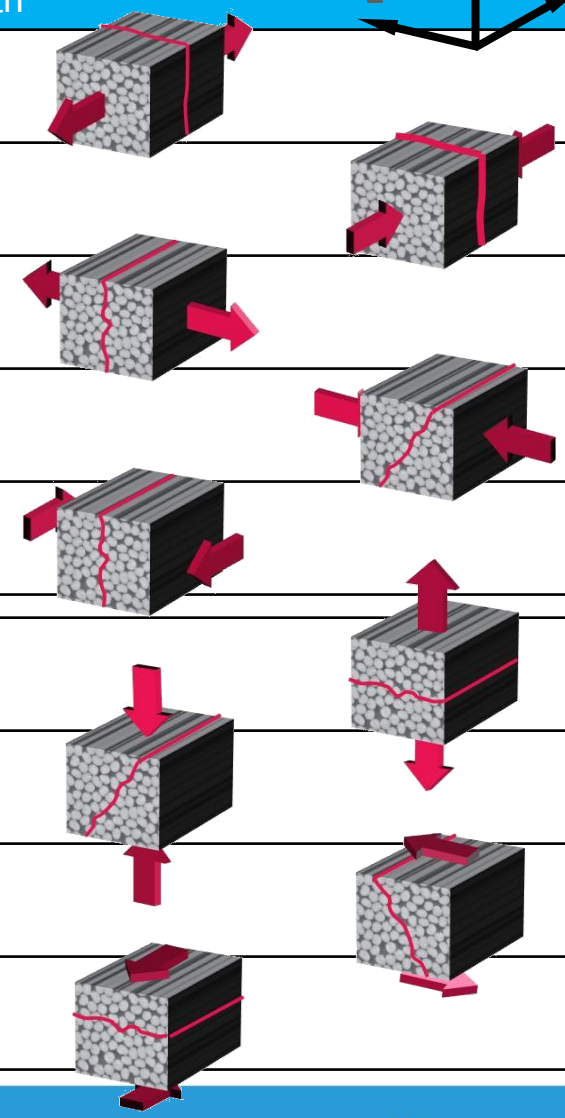
Overview of composite material models

	Element	Failure criteria	Comments
***MAT_022:** COMPOSITE_DAMAGE	Shell, Tshell, Solid	Chang-Chang	No fibre compression failure Simple brittle model No crash front algorithm
***MAT_054/55:** ENHANCED_COMPOSITE_DAMAGE	Shell, Tshell, Solid	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	Shell, Tshell (1,2)	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	Shell, Tshell, Solid, SPH	Modified Hashin	Similar to MAT_054 Crash front algorithm Minimum stress limit factor
***MAT_158:** RATE_SENSITIVE_COMPOSITE_FABRIC	Shell, Tshell	Modified Hashin	Same as MAT_058 Rate sensitive
***MAT_261:** LAMINATED_FRACTURE_DAIMLER_PINHO	Shell, Tshell, Solid	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	Shell, Tshell, Solid	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



	Loading Mode		Stress	Strength
Solid and shell elements	Fibre	Tension	$\sigma_{11} > 0$	X_T
		Compression	$\sigma_{11} < 0$	X_C
	Transverse	Tension	$\sigma_{22} > 0$	Y_T
		Compression	$\sigma_{22} < 0$	Y_C
	Shear	In-Plane	$\sigma_{12} > 0$	S_{12}
Solid elements only	Through the thickness (TT)	Tension	$\sigma_{33} > 0$	Z_T
		Compression	$\sigma_{33} < 0$	Z_C
	Shear	TT Transverse	$\sigma_{23} > 0$	S_{23}
		TT Longitudinal	$\sigma_{13} > 0$	S_{13}

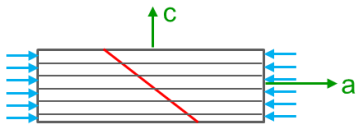
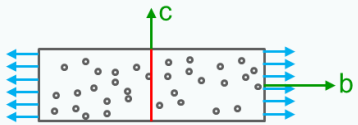
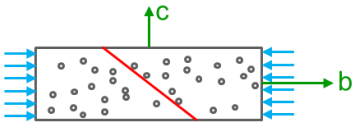


[Kawashita, Bristol, 2017]

*MAT_054: ENHANCED_COMPOSITE_DAMAGE

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	54: Chang-Change	
 Tensile Fibre Mode	Failure assumed when $e_f^2 > 0$ $e_f^2 = \left(\frac{\sigma_{11}}{X_T}\right)^2 + \beta\left(\frac{\sigma_{12}}{S_C}\right)^2 - 1$	Degradation after failure: $E_{11} = E_{22} = G_{12} = \nu_{12} = \nu_{21} = 0$
 Compressive Fibre Mode	Failure assumed when $e_c^2 > 0$ $e_c^2 = \left(\frac{\sigma_{11}}{X_C}\right)^2 - 1$	Degradation after failure: $E_{11} = \nu_{12} = \nu_{21} = 0$
 Tensile Matrix Mode	Failure assumed when $e_m^2 > 0$ $e_m^2 = \left(\frac{\sigma_{22}}{Y_T}\right)^2 + \beta\left(\frac{\sigma_{12}}{S_C}\right)^2 - 1$	Degradation after failure: $E_{22} = G_{12} = \nu_{21} = \nu_{12} = 0$
 Compressive Matrix Mode	Failure assumed when $e_d^2 > 0$ $e_d^2 = \left(\frac{\sigma_{22}}{2S_C}\right)^2 + \left[\left(\frac{Y_C}{2S_C}\right)^2 - 1\right]\frac{\sigma_{22}}{Y_T} + \left(\frac{\sigma_{12}}{S_C}\right)^2 - 1$	Degradation after failure: $E_{22} = G_{12} = \nu_{21} = \nu_{12} = 0$

*MAT_054: ENHANCED_COMPOSITE_DAMAGE

		1	2	3	4	5	6	7	8
Elastic, orthotropic parameters	Card 1	MID	RO	EA	EB	(EC)	PRBA	PRCA	PRCB
	Card 2	GAB	GBC	GCA	(KF)	AOPT			
Material coordinate system	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
Failure modelling parameters	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		

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 ration in ba plane
PRCA	Poisson's ration in ca plane
PRCB	Poisson's ration in cb plane
GAB	Shear modulus in the ab plane
GBC	Shear modulus in the bc plane
GCA	Shear modulus in the ca plane

Parameters related to elastic, orthotropic material properties

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

Parameters related to material coordinate definition

*MAT_054: ENHANCED_COMPOSITE_DAMAGE

		1	2	3	4	5	6	7	8
Elastic, orthotropic parameters	Card 1	MID	RO	EA	EB	(EC)	PRBA	PRCA	PRCB
	Card 2	GAB	GBC	GCA	(KF)	AOPT			
Material coordinate system	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
Failure modelling parameters	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		

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 elastic, orthotropic material properties

FBRT	Softening for fibre tensile strength
YCFAC	Reduction factor for compressive fibre strength

Parameters related to post-matric compressive failure

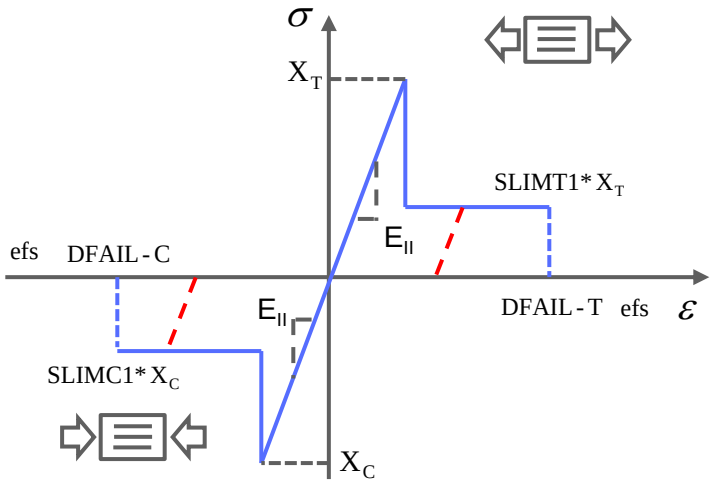
DFAILM	Maximum strain for matrix straining in tension or compression
DFAILS	Maximum tensorial shear strain (ε12 = γ12/2)
DFAILT	Maximum strain for fibre tension
DFAILC	Maximum strain for fibre compression
EFS	Effective failure strain
TFAIL	Time step size criteria for element deletion
2WAY	Flag to turn on 2-way fibre action
TI	Flag for transversal isotropic behaviour (Solids)

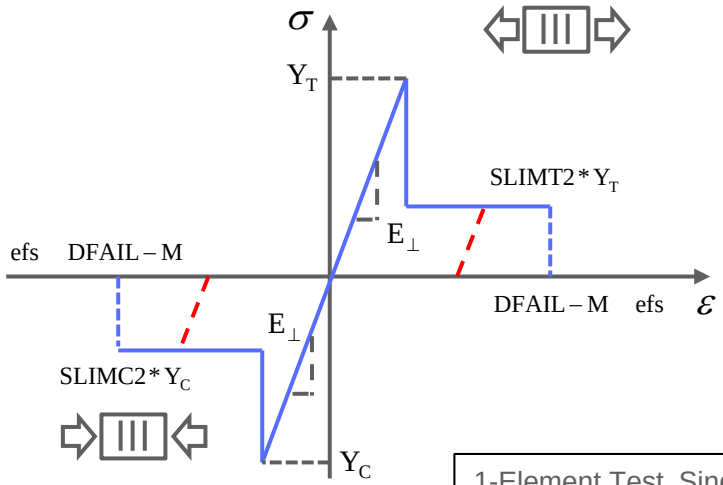
Parameters related to element deletion

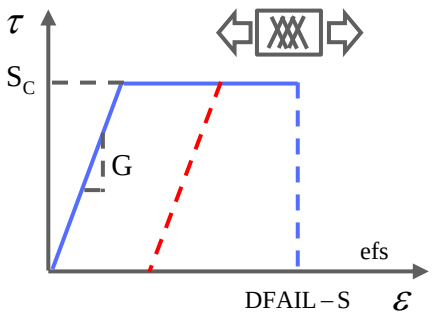
SOFT	Softening reduction factor for material strength in crashfront elements
PFL	Percentage of layers that must fail until crashfront
SOFT2	Optional ‘orthogonal# softening reduction factor
SOFTG	Softening reduction factor for transverse stiffness

Parameters related to crashfront

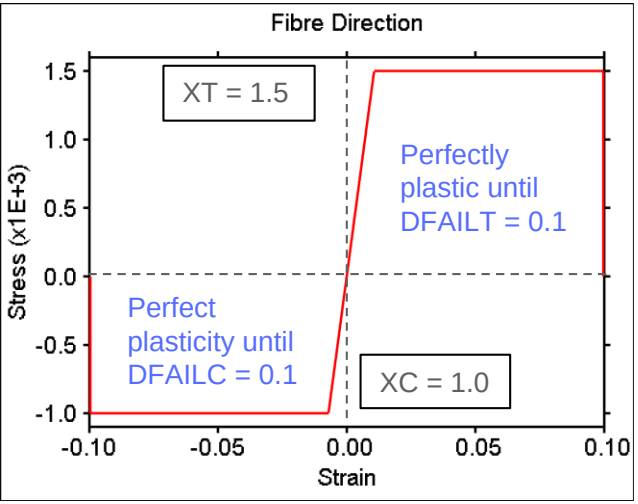
*MAT_054: ENHANCED_COMPOSITE_DAMAGE

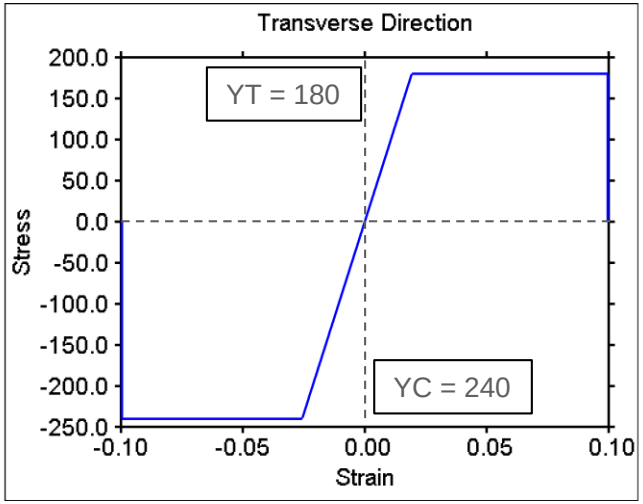


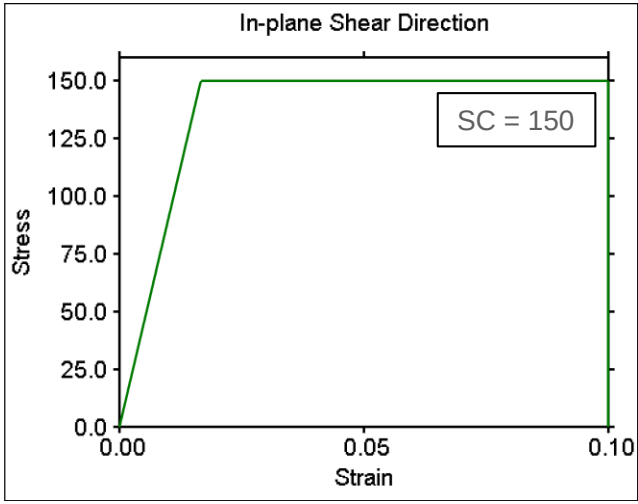


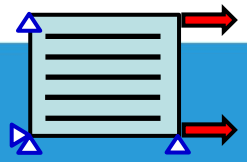


1-Element Test, Single Layer (Shell, ELFORM = 16)



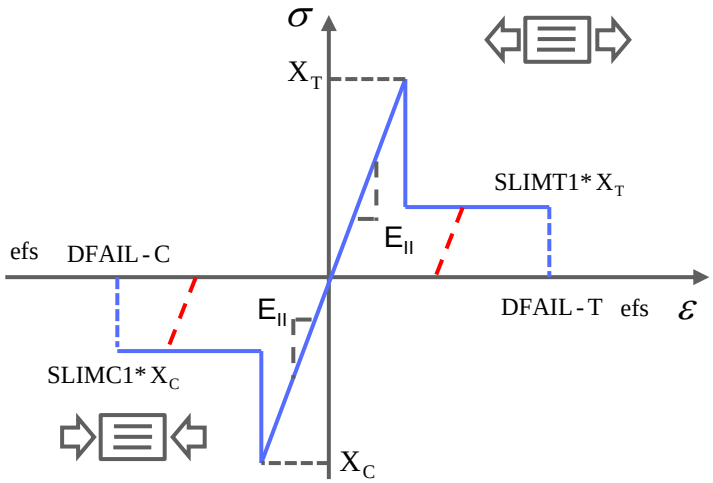


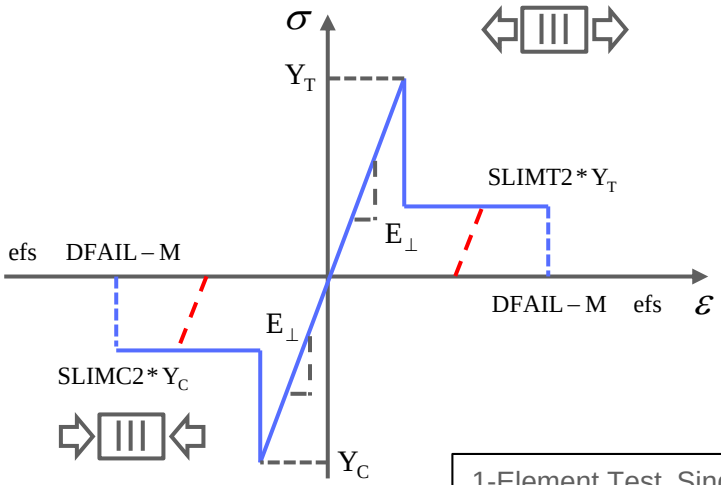


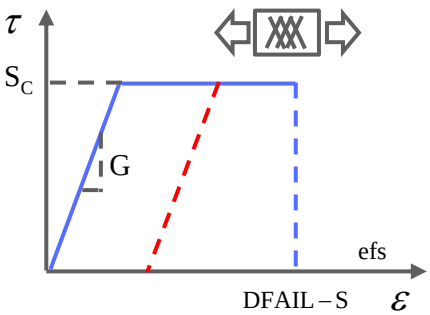


DFAILxx = 0.1
EA = 141 GPa
EB = 9.34 GPa
GAB = 4.5 GPa

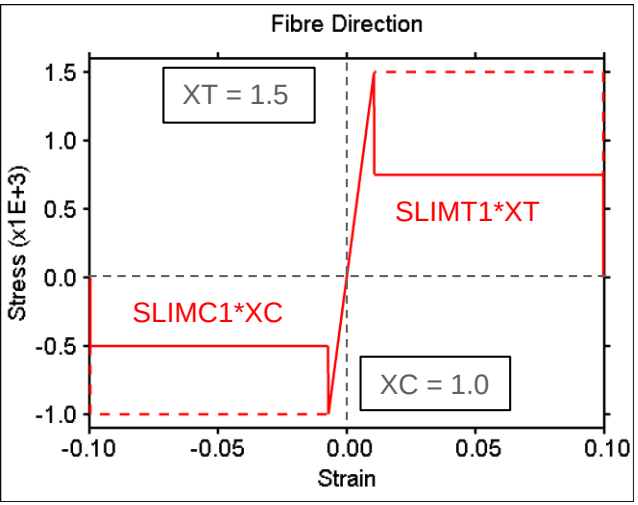
*MAT_054: ENHANCED_COMPOSITE_DAMAGE

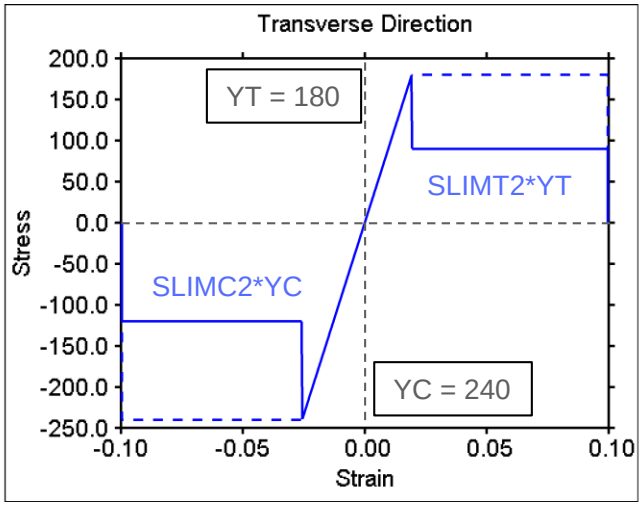


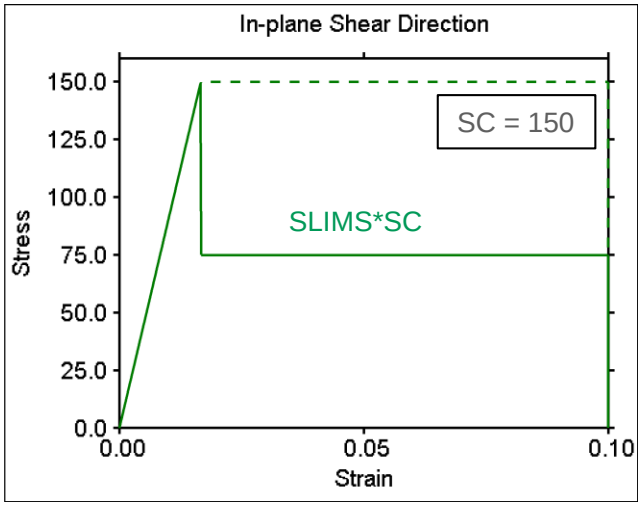


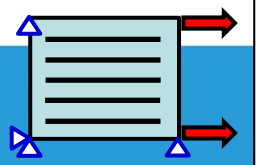


1-Element Test, Single Layer (Shell, ELFORM = 16)









DFAILxx = 0.1
EA = 141 GPa
EB = 9.34 GPa
GAB = 4.5 Gpa
SLIMxx = 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**

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	ε_{aa}	Local strain a-direction
#11	ε_{bb}	Local strain b-direction
#12	ε_{ab}	Local shear strain
#16	d_{56}	Transverse shear damage

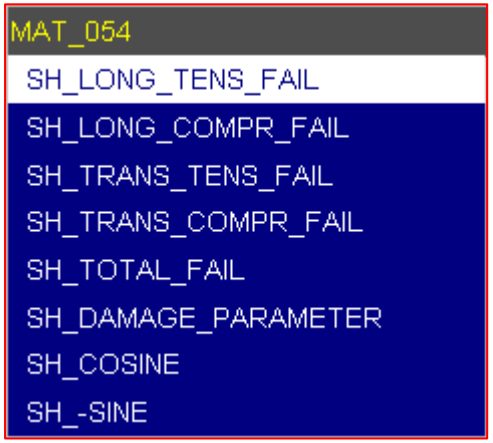
SOLIDS		
#1	*ef*	Failure flag tensile fibre mode
#2	ec	Failure flag compressive fibre mode
#3	em	Failure flag tensile matrix mode
#4	ed	Failure flag compressive matrix mode
#5	*efail*	Integration point status
#6	*dam*	Damage parameter (SOFT)
#15	ε_{aa}	Local strain a-direction
#16	ε_{bb}	Local strain b-direction
#17	ε_{ab}	Local shear strain

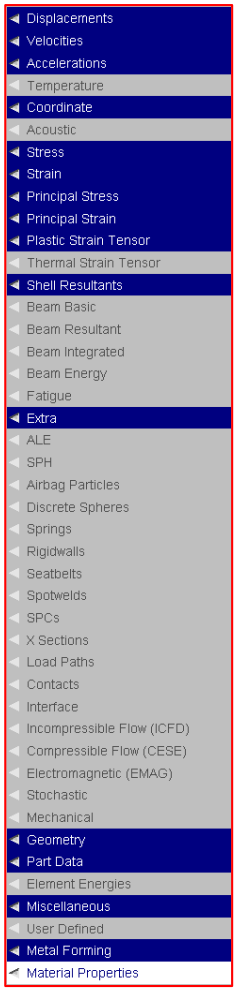
ef, ec, em, ed, efail: 1-intact; 0-failed
dam: -1-intact; 0-failed
d56: 0-intact; 1-full damaged

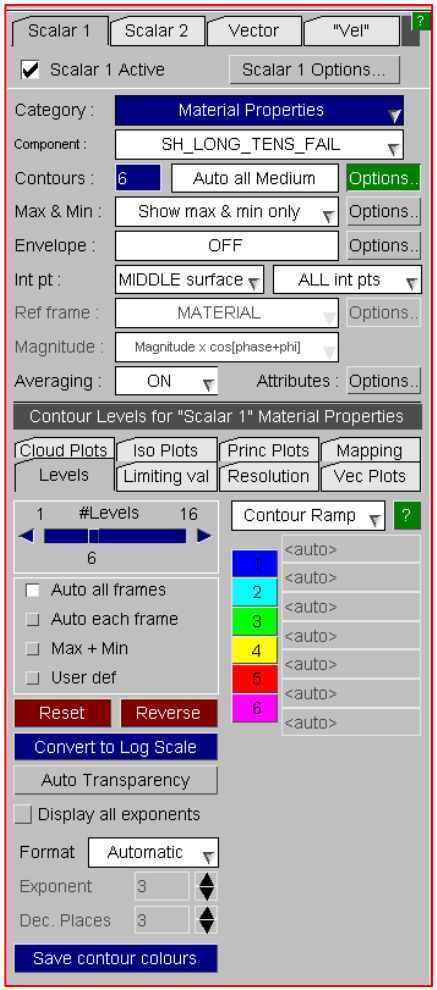
*MAT_054: ENHANCED_COMPOSITE_DAMAGE

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:

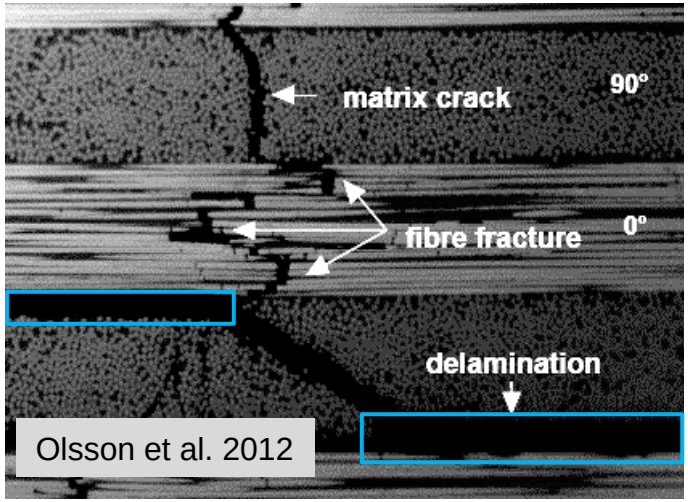
1. A ZTF file is required to provide the link between parts and materials.

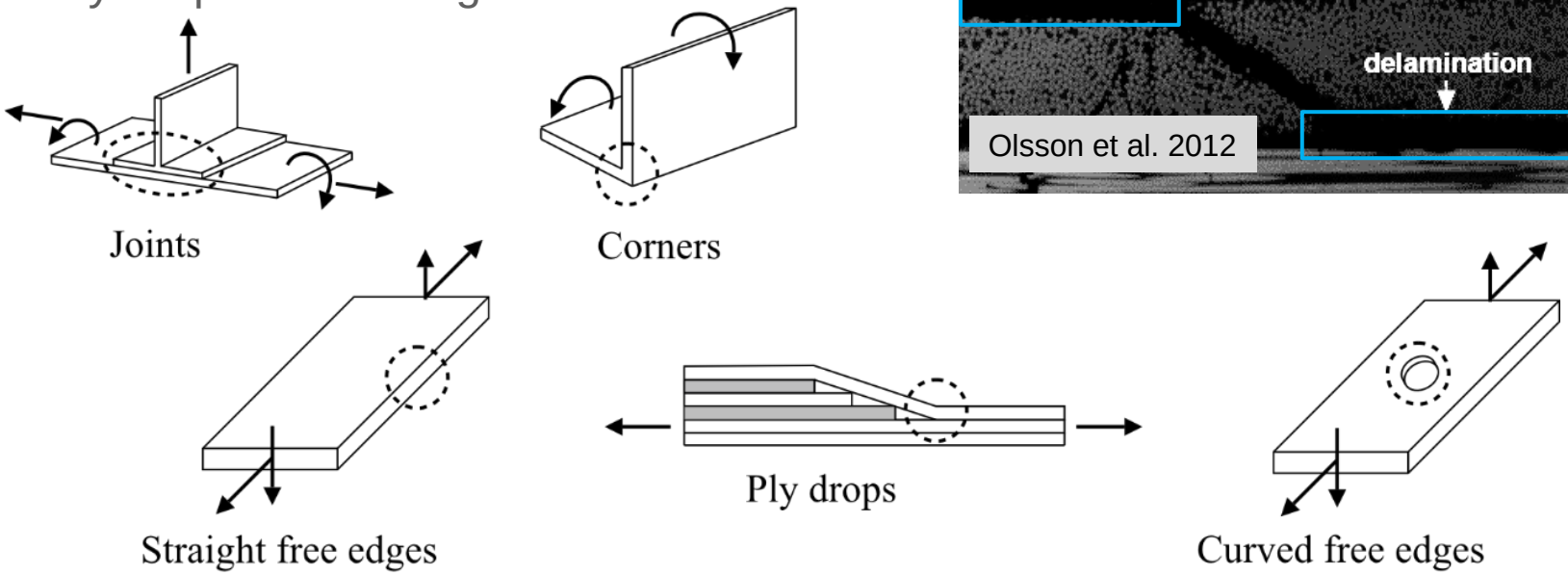
```
$ Material type code,material name (including the leading '*'),data slot (intg pt),variable
name, short name (alias),element type applicability (3 - shell/1 - solid/2 - both)
MAT_002,*MAT_ORTHOTROPIC_ELASTIC,1,C12_STIFFNESS_COMP,C12 Stiffness,3
MAT_002,*MAT_ORTHOTROPIC_ELASTIC,2,C13_STIFFNESS_COMP,C13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

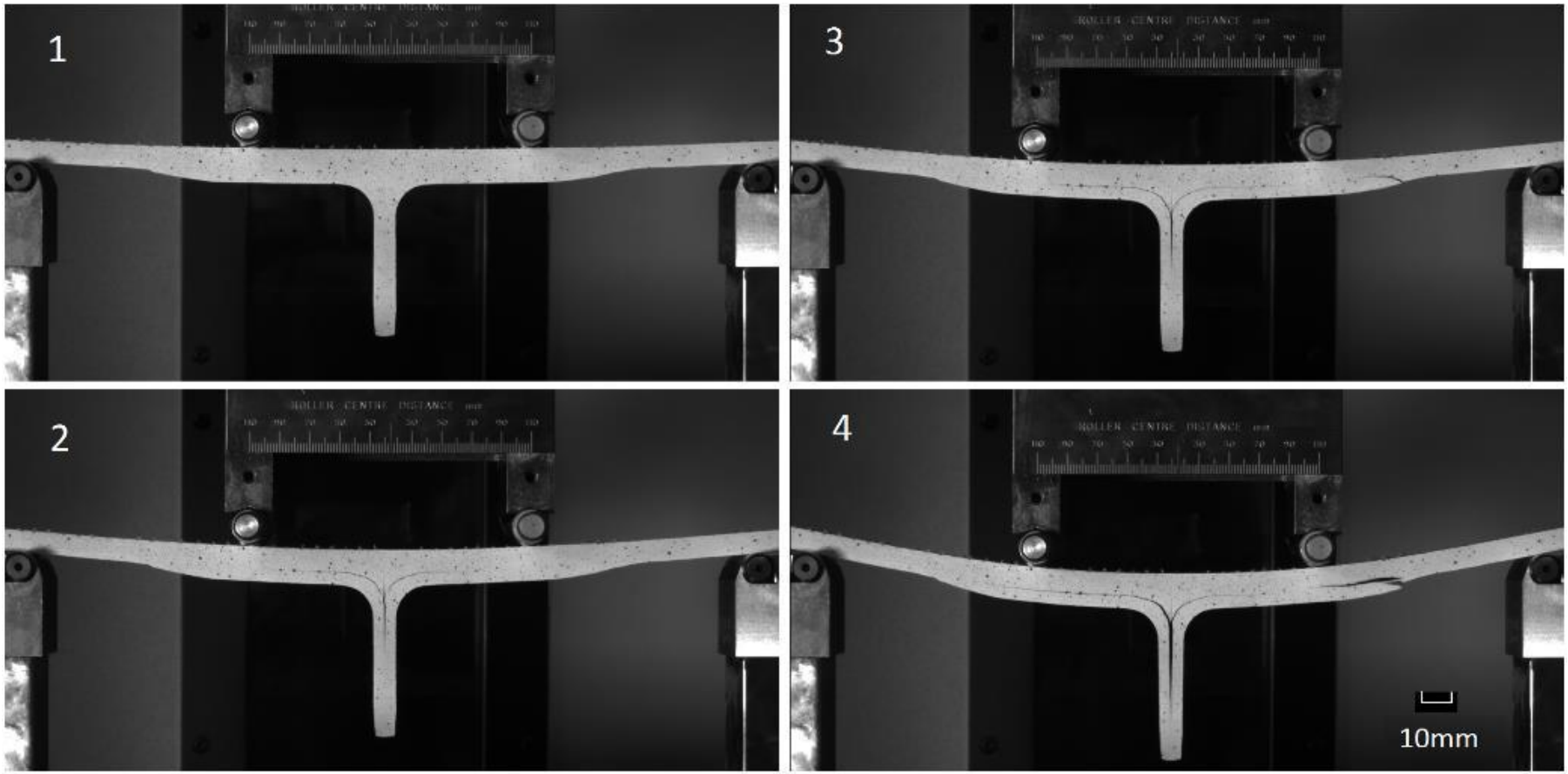
2. A new pre-populated CSV file (mat_prop.csv) provides material extra variable information – this can be edited by the user to allow new materials to be added or existing component names to be modified.

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



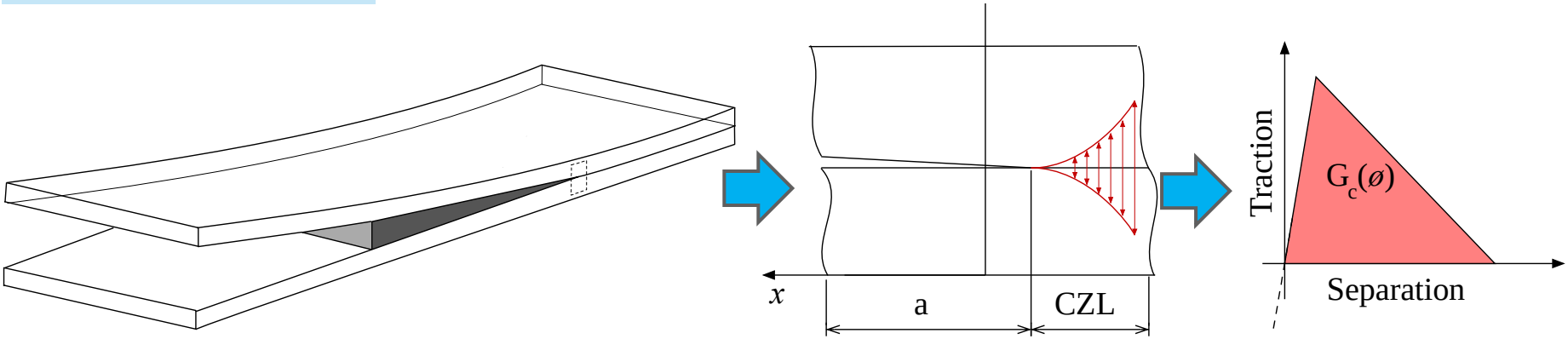




Skin-stiffener debonding under 4-point bend loading

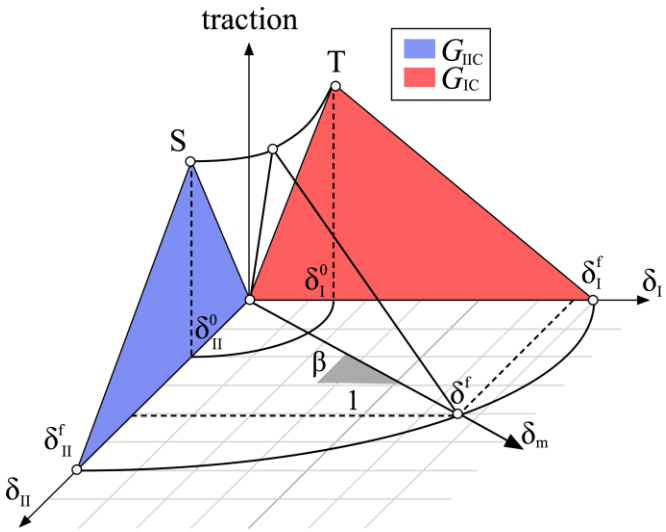
[Clegg et al., Evaluation of the effects of tufting on the performance of composite T-joints, ECCM17, 2016]

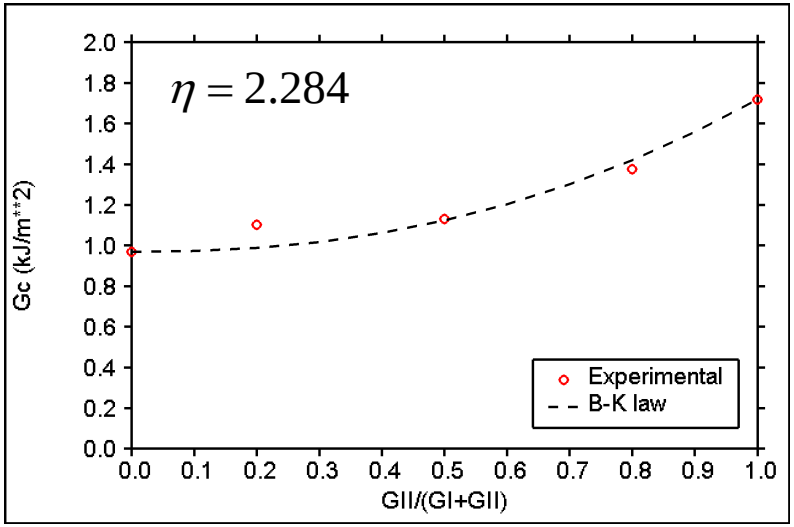
General Remarks



- 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

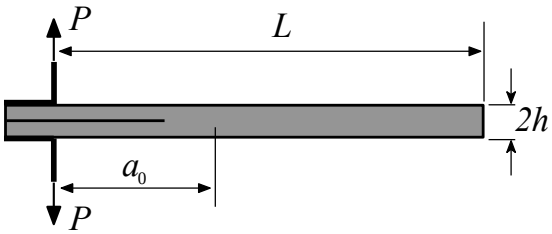


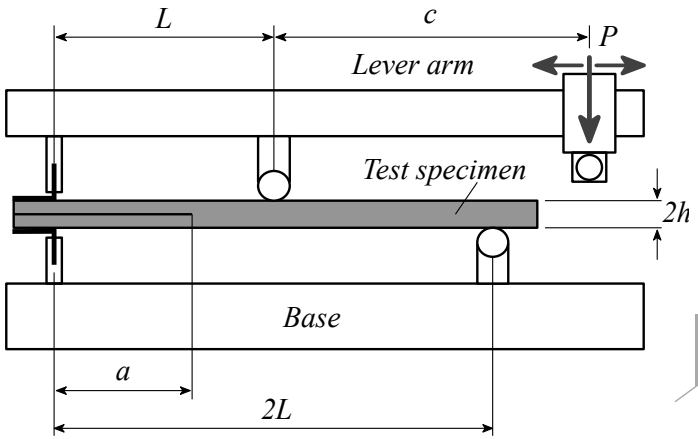


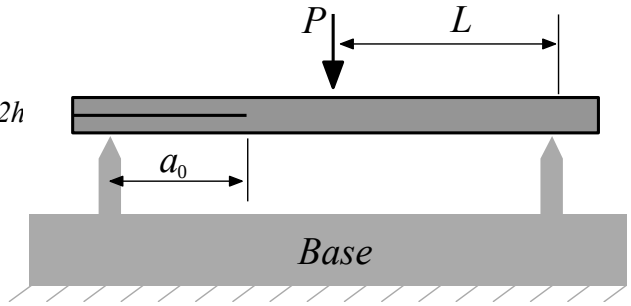
Double Cantilever Beam (DCB)
Mode I

Mixed-Mode Bend (MMB)
Mode I/II

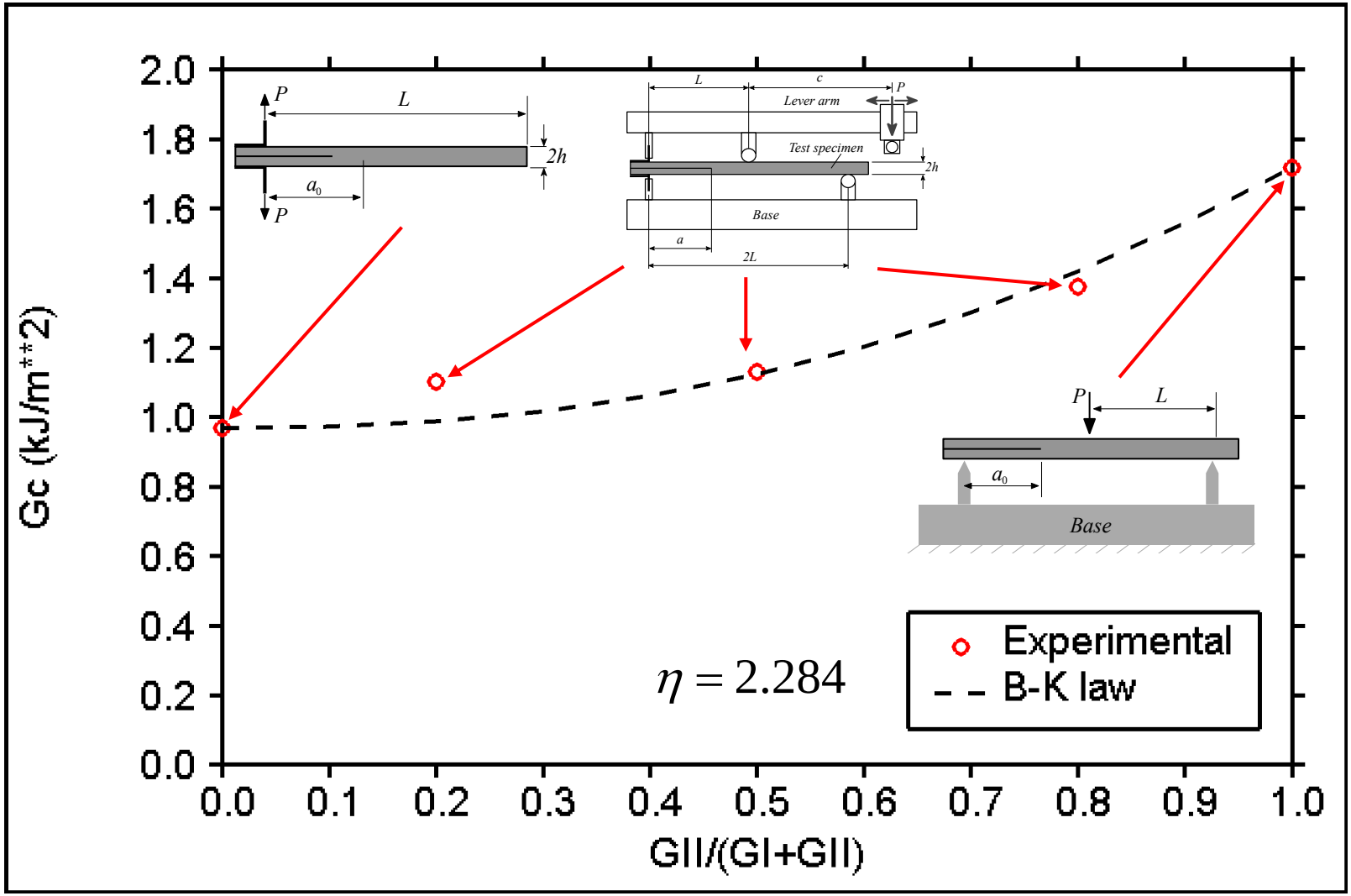
End-Notched Flexure (ENF)
Mode II



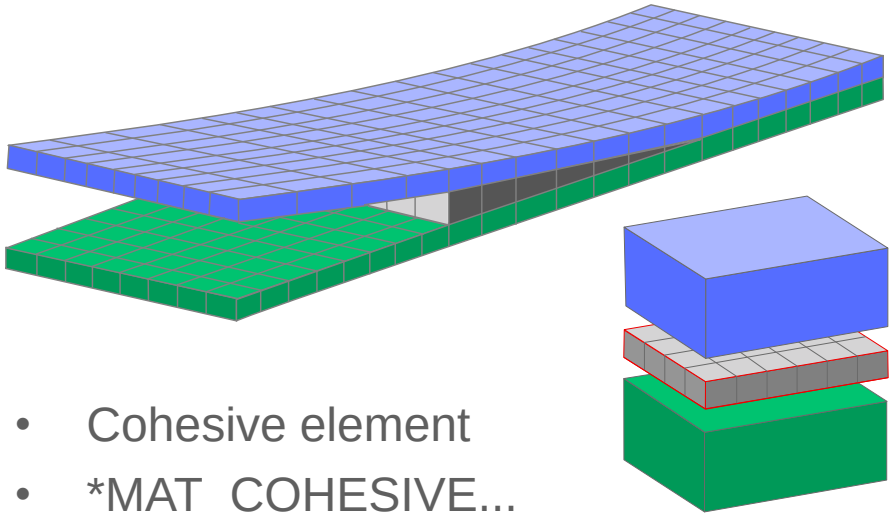




Fracture toughness vs. Mode-mixity

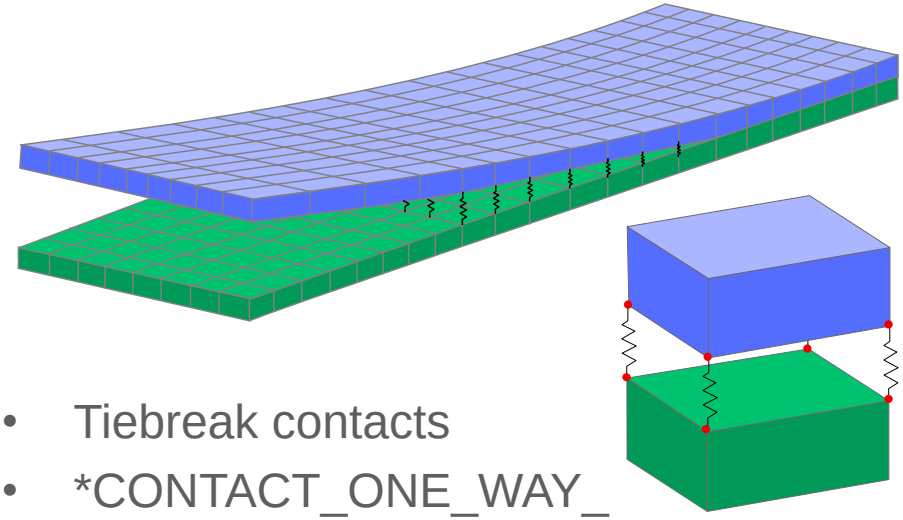


LS-DYNA Option 1:
Cohesive Zone Elements



- Cohesive element
- *MAT_COHESIVE...
- Finite or zero thickness elements
- Composite shells, tshells or solid elements allowed

LS-DYNA Option 2:
Tiebreak Contacts

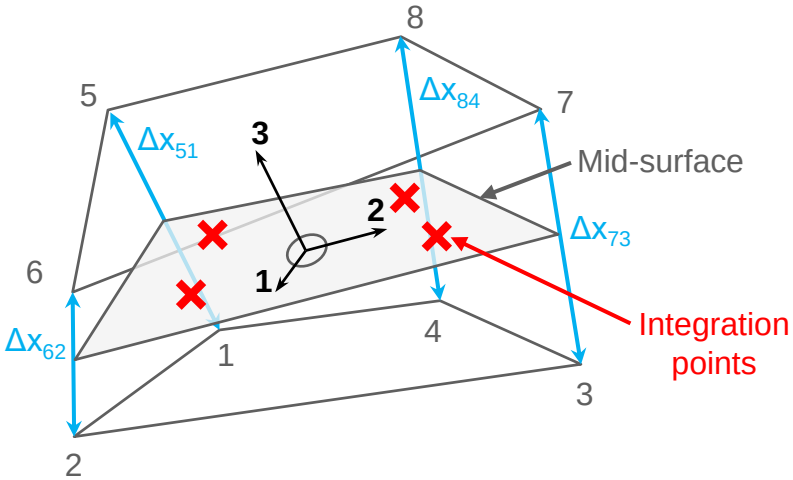


- Tiebreak contacts
- *CONTACT_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
- Composite shells, tshells or solid elements allowed

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

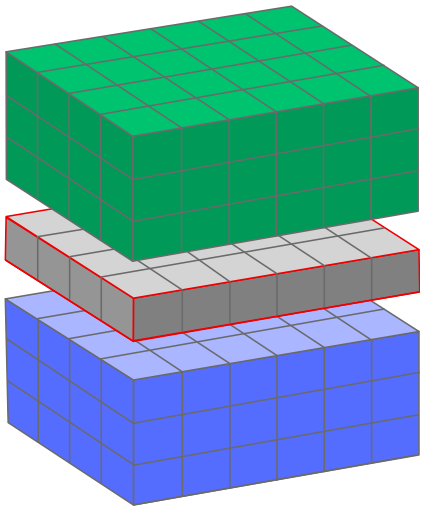
Cohesive Zone Elements

- **Cohesive elements** typically formulated in terms of a **traction vs. relative displacement** relationship instead of the traditional stress vs. strain relation.
- Tractions 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.



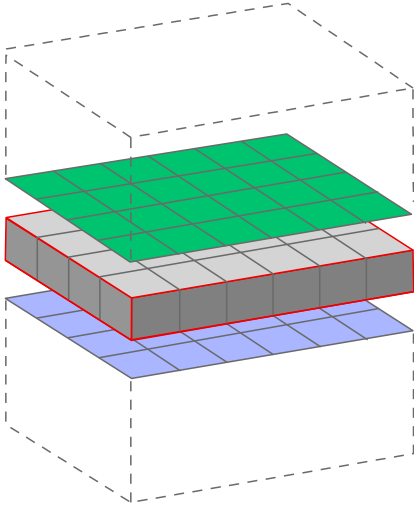
8-node Type 19 Elements

- 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



Cohesive Zone Elements: Formulation

Relative nodal displacements

Local relative displacement at integration points

Local interface tractions

Internal nodal forces

Displacements at an integration point are:

$$\Delta \mathrm{x} = \sum_{i=1}^{4} N_i(s,t)\Delta \mathrm{x}_{i+4,i}$$

$$\Delta \mathrm{X} = \sum_{i=1}^{4} N_i(s,t)\Delta \mathrm{X}_{i+4,i}$$

$$\Delta\delta = R^T(t)\Delta \mathrm{x} - R^T(0)\Delta \mathrm{X}$$

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

$$\begin{bmatrix} t_1 \\ t_2 \\ t_3 \end{bmatrix} = \begin{bmatrix} E_T & 0 & 0 \\ 0 & E_T & 0 \\ 0 & 0 & E_N \end{bmatrix} \begin{bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{bmatrix}$$

Units:

$$\left[\frac{N}{mm^2}\right] = \left[\frac{N}{mm^3}\right] \cdot \left[\frac{mm}{}\right]$$

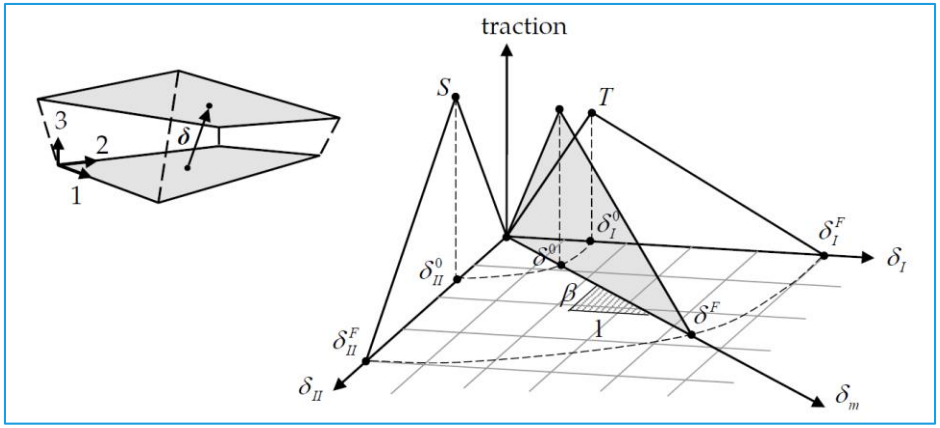
Integrate tractions over the mid-surface and rotate back to global system:

$$F_i = \mathbf{R}(t)\sum_{g=1}^{4} \mathbf{T}_g N_i(s_g,t_g)\det(\mathrm{J}_g), \text{ for } 1 \le i \le 4$$

Cohesive Material Models: Overview

LS-DYNA Material Model	Traction-Separation Law/Shape	Initiation & Propagation Criteria	Comments
***MAT_138:** COHESIVE_MIXED_MODE	Bilinear (linear softening)	Quadratic mixed-mode. Supports power law (XMU > 0) and B-K law (XMU < 0)	Simplification of *MAT_COHESIVE_ GENERAL.
***MAT_184:** COHESIVE_ELASTIC	Linear	Stress based criteria governed by normal and/or tangential strengths	Simple cohesive elastic model
***MAT_185:** COHESIVE_ELASTIC_TH	Tri-linear	Quadratic mixed-mode governed by dimensionless separation parameter λ (i.e. traction drops to zero when λ =1)	Cohesive model by Tvergaard and Hutchinson. Same loading and unloading path; completely reversible
***MAT_186:** COHESIVE_GENERAL	Arbitrary normalised traction-separation law given by a load curve (TSLC)	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	
***MAT_240:** COHESIVE_MIXED_MODE ELASTO_PLASTIC_RATE	Tri-linear	Quadratic yield and damage criterion in mixed-mode loading. Damage evolution is governed by a power-law	Rate-dependent, elastic-ideal plastic

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:

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

$$\delta_I = \delta_3$$

Mode I

$$\delta_{II} = \sqrt{\delta_1^2 + \delta_2^2}$$

Mode II

- The **mixed-mode damage initiation displacement** is defined as:

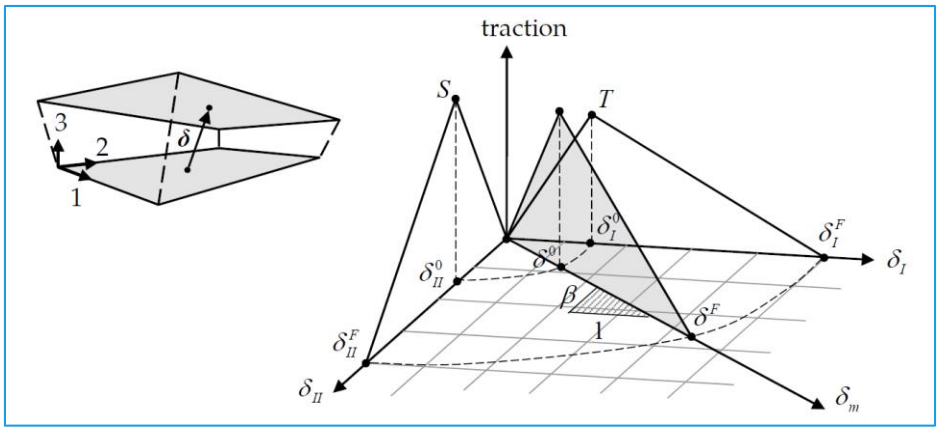
$$\delta^0 = \delta_I^0 \delta_{II}^0 \sqrt{\frac{1+\beta^2}{\left(\delta_{II}^0\right)^2 + \left(\beta\delta_I^0\right)^2}}$$

$$\delta_I^0 = T / EN$$

$$\delta_{II}^0 = S / ET$$

$$\beta = \delta_{II} / \delta_I$$

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 δ^F (total failure):

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

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

- **Formulation 2**: Benzeggagh-Kenane (**XMU** < 0):

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

$$GC = GIC + \left(GIIC - GIC\right)\left(\frac{GII}{GI + GII}\right)^{\eta}$$

MAT_COHESIVE_MIXED_MODE: MAT_138

	1	2	3	4	5	6	7	8
Card 1	MID	RO	ROFLG	INTFAIL	EN	ET	GIC	GIIC
Card 2	XMU	T	S	UND	UTD			

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

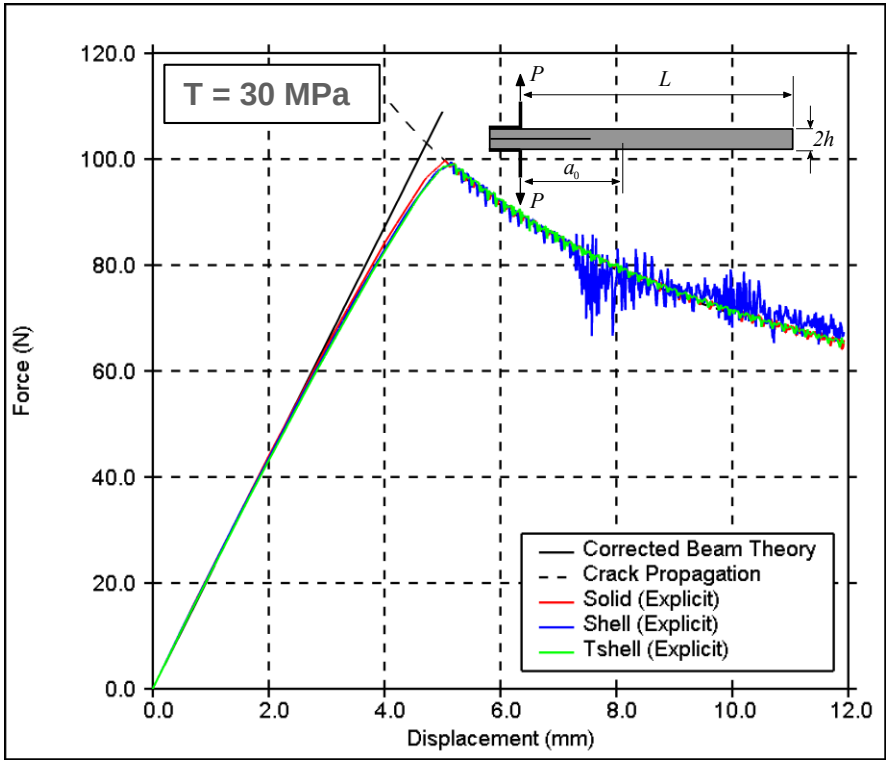
T: Peak traction in normal direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

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

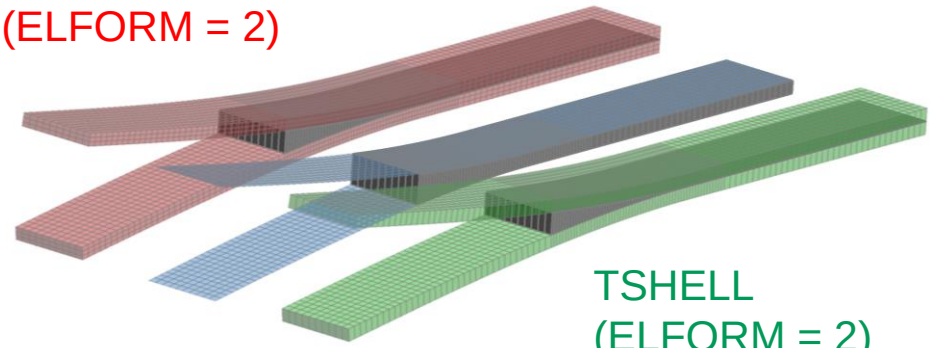


Explicit Analysis: Quasi-Static

- Comparison of element formulations

SHELL
(ELFORM = 16)

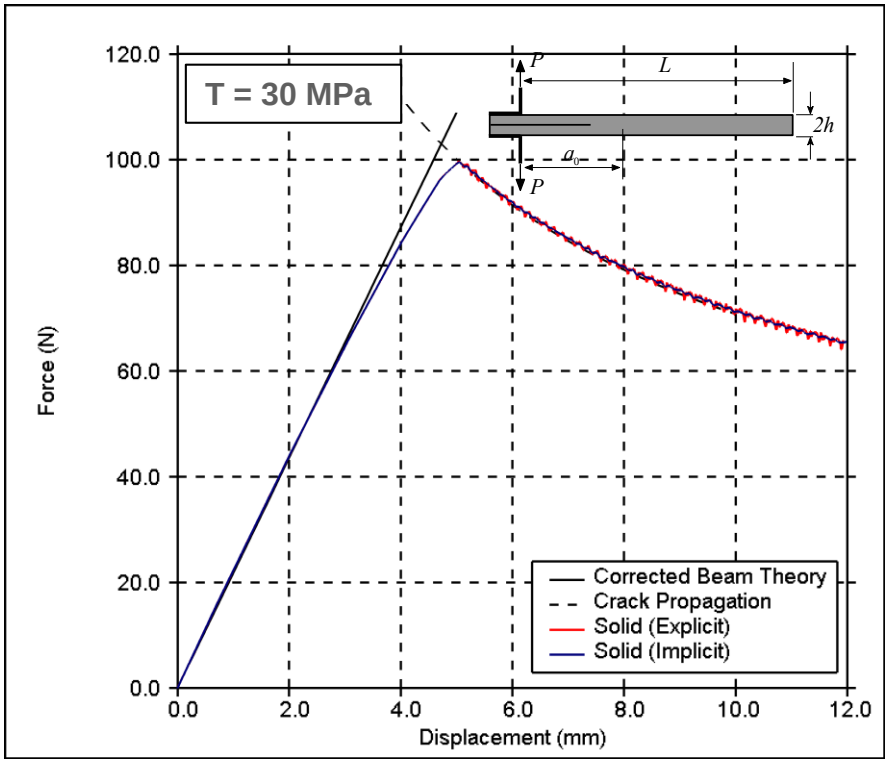
SOLID
(ELFORM = 2)



TSHELL
(ELFORM = 2)

	1	2	3	4	5	6	7	8
Card 1	MID	RO	ROFLG	INTFAIL	EN	ET	GIC	GIIC
	138	**1.0E-5**	**0.0**	**0.0**	**1E+5**	**1E+5**	**0.969**	**1.719**
Card 2	XMU	T	S	UND	UTD			
	-2.284	**30.0**	**60.0**	**-**	**-**			

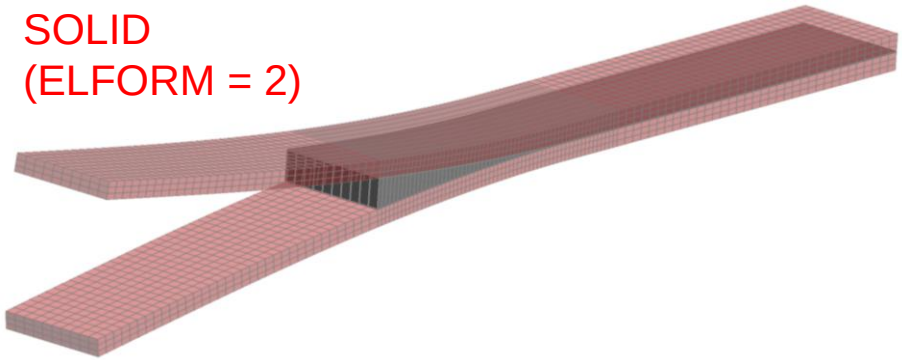
Double Cantilever Beam: Mode I



Implicit Non-linear Analysis

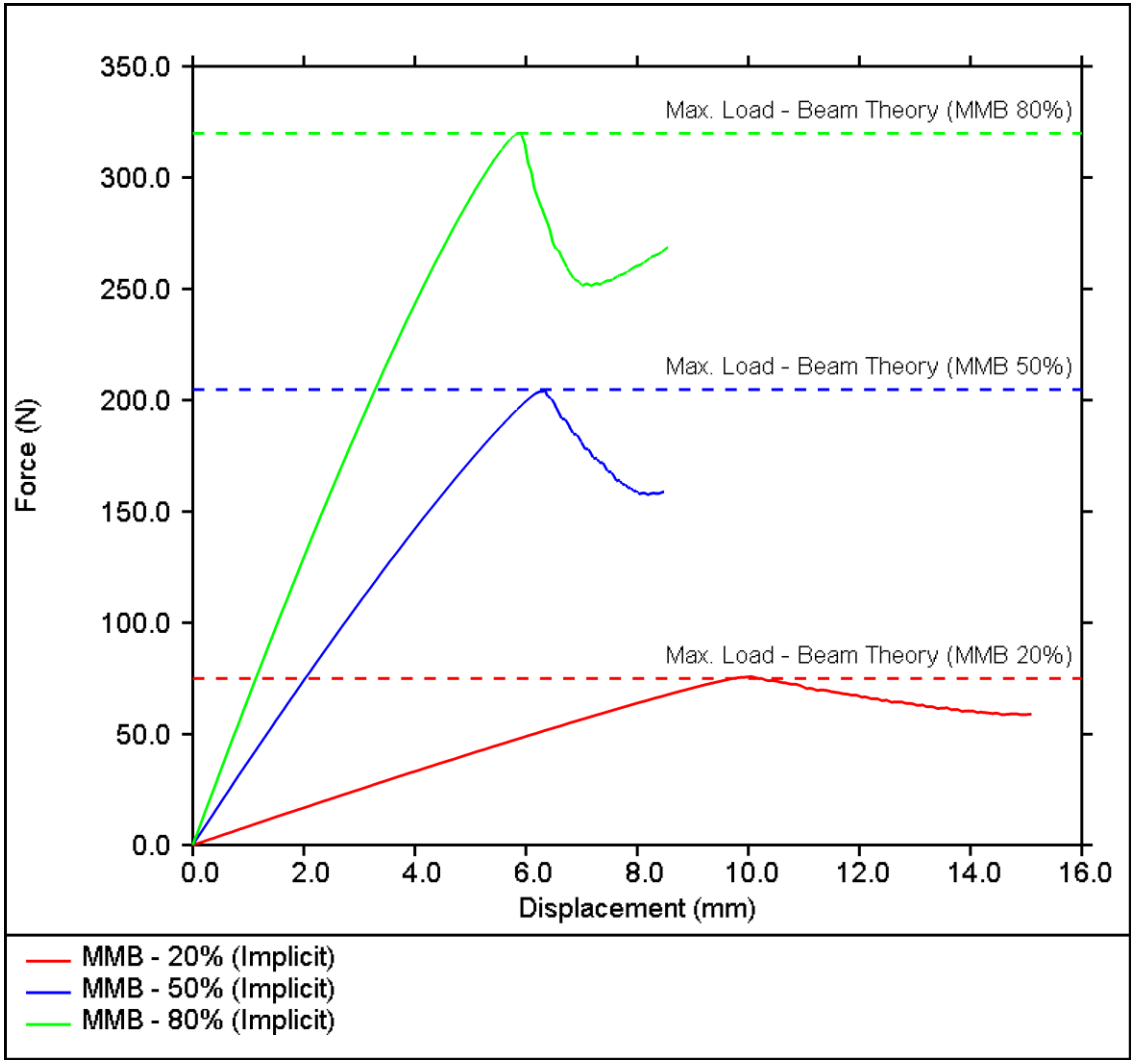
- Explicit vs implicit non-linear static analysis

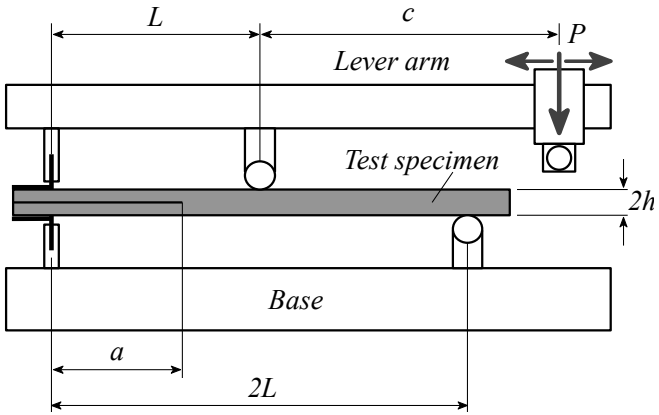
SOLID
(ELFORM = 2)

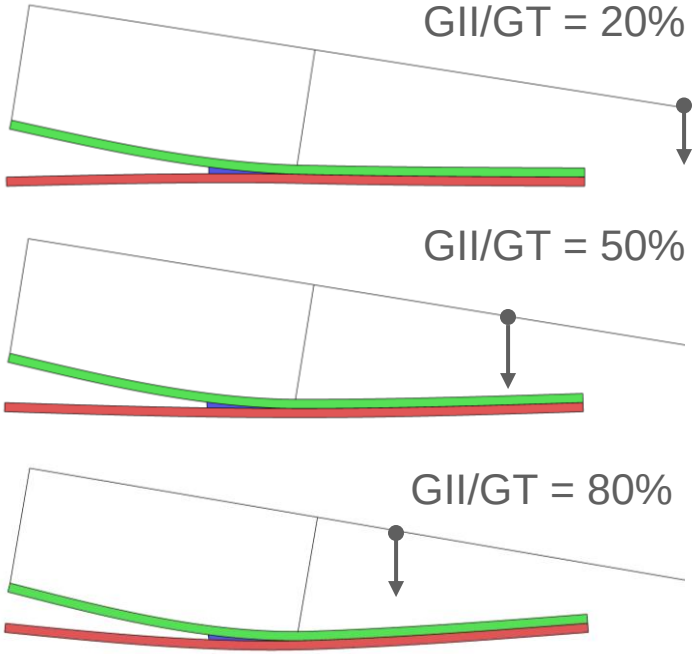


	1	2	3	4	5	6	7	8
Card 1	MID	RO	ROFLG	INTFAIL	EN	ET	GIC	GIIC
	138	**1.0E-5**	**0.0**	**0.0**	**1E+5**	**1E+5**	**0.969**	**1.719**
Card 2	XMU	T	S	UND	UTD			
	-2.284	**30.0**	**60.0**	**-**	**-**			

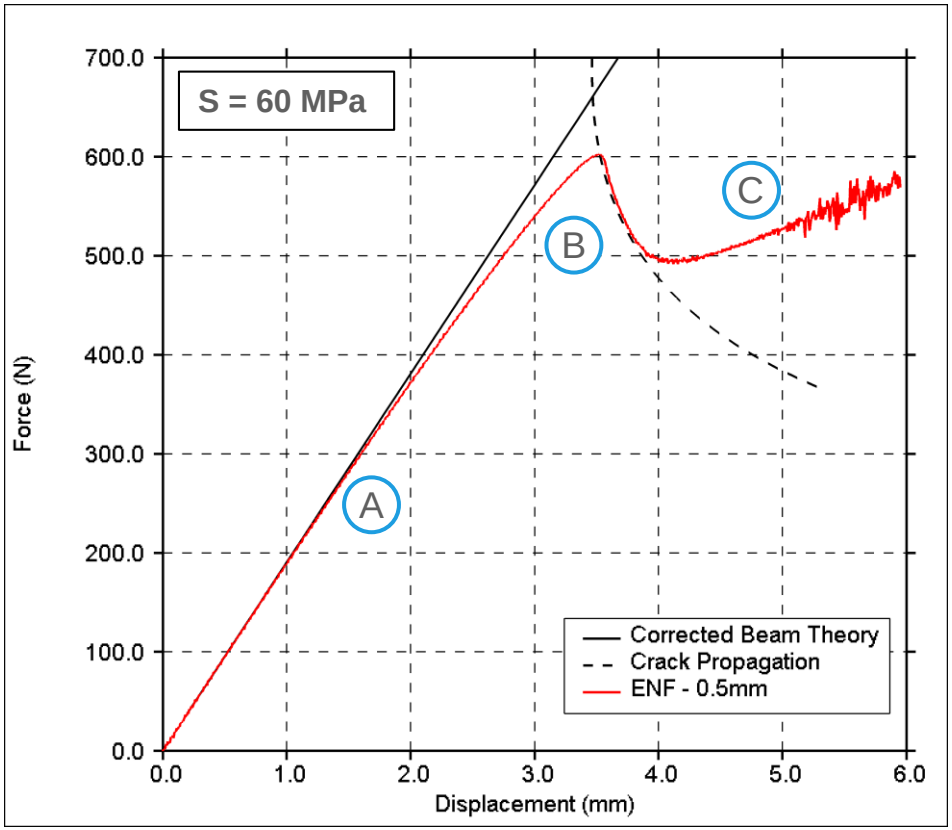
Mixed-Mode Bend: Mode I/II

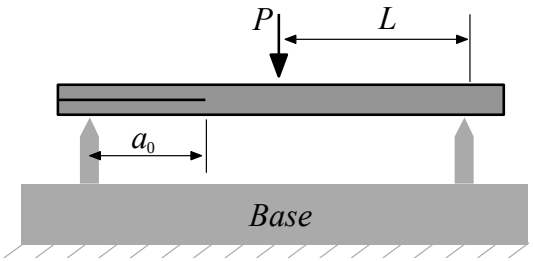


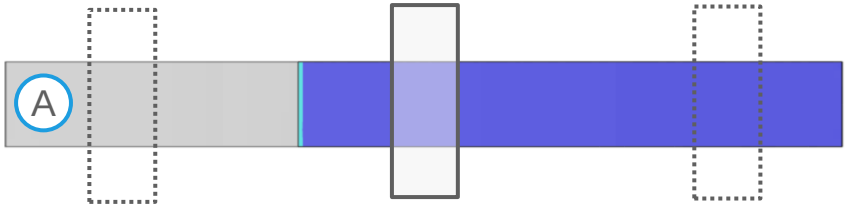




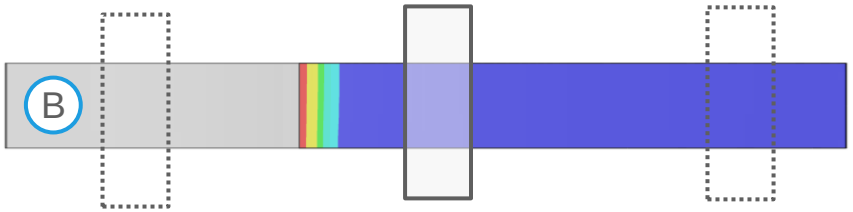
3-Point End Notched Flexure: Mode II



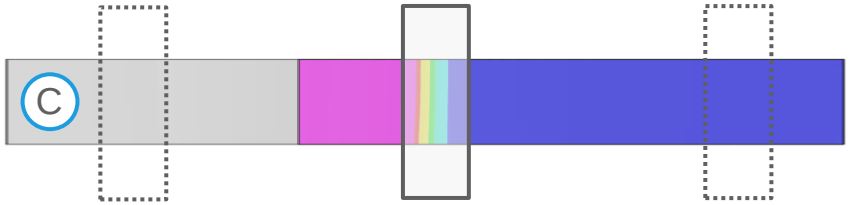




First row of crack front element enter the damage regime

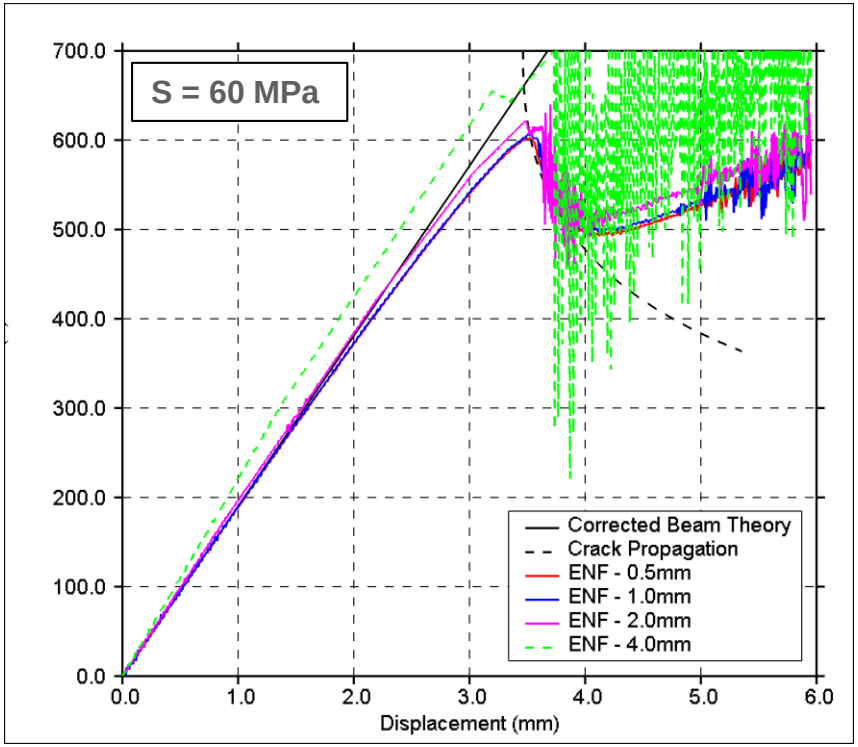


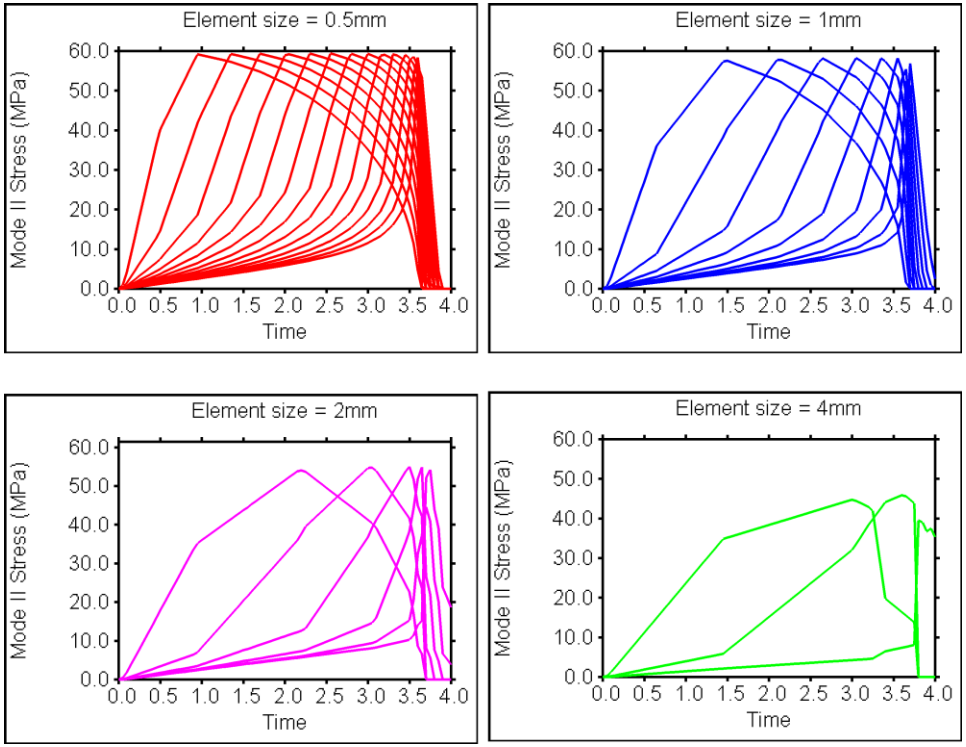
Damage flag prior to crack propagation (PS =1)



Crack propagation (PS =1)

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

Mesh Dependency – Strength

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

Young's modulus of the material

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

Critical energy release rate

Maximum strength

Cohesive zone length

Scaling factor

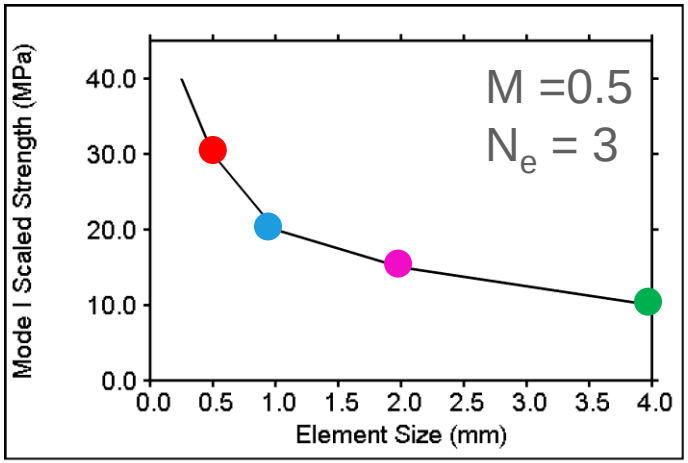
$$\sigma_{\max} = \sqrt{\frac{MEG_c}{N_e l_e}}$$

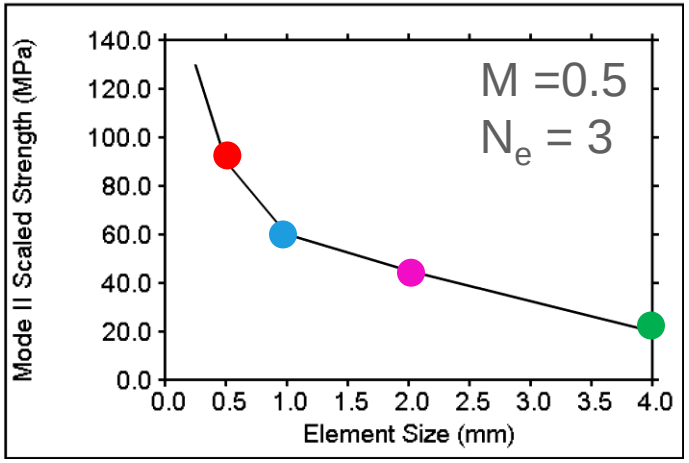
$$N_e = \frac{l_{cz}}{l_e}$$

Number of elements in FPZ

Element size

(Turon et al. 2007)

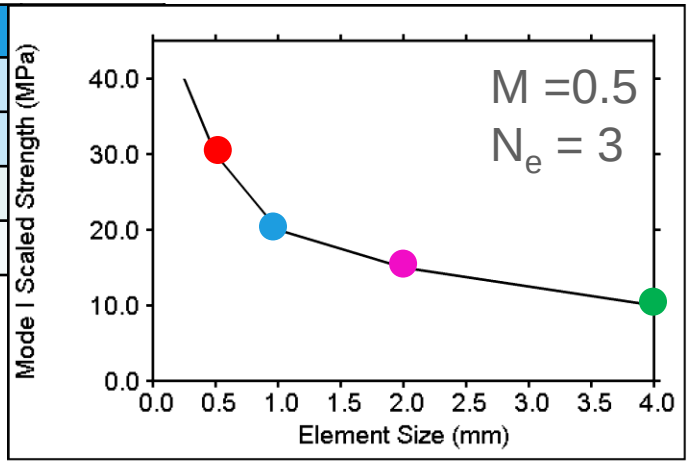




Mesh Dependency – Strength

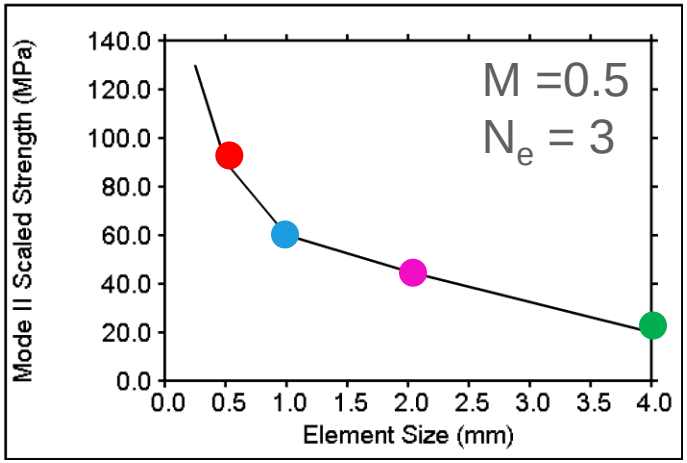
Current mesh design guidelines recommend at **least three elements** within the fully developed fracture process zone (FPZ) to preserve numerical accuracy.

	1	2	3	4	5	6	7
Card 1	MID	RO	ROFLG	INTFAIL	EN	ET	GIC
	138	**1.0E-5**	**0.0**	**0.0**	**1E+5**	**1E+5**	**0.969**
Card 2	XMU	T	S	UND	UTD		
	-2.284	**-30.0**	**-60.0**	**-**	**-**		



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

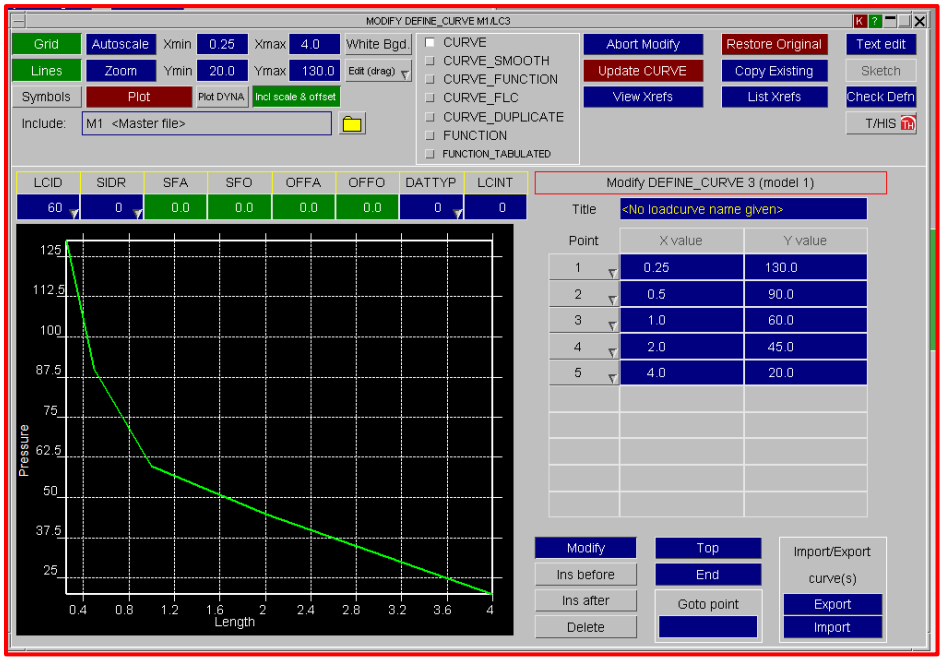
S: Peak traction in tangential direction
LT.0.0: Load curve ID, which defines peak traction as a function of element size

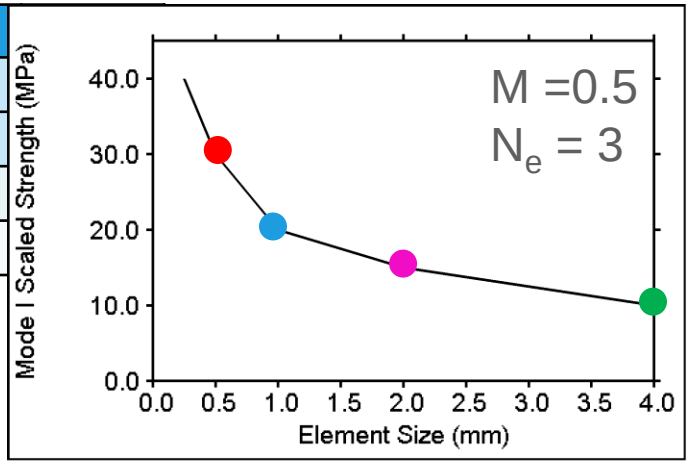


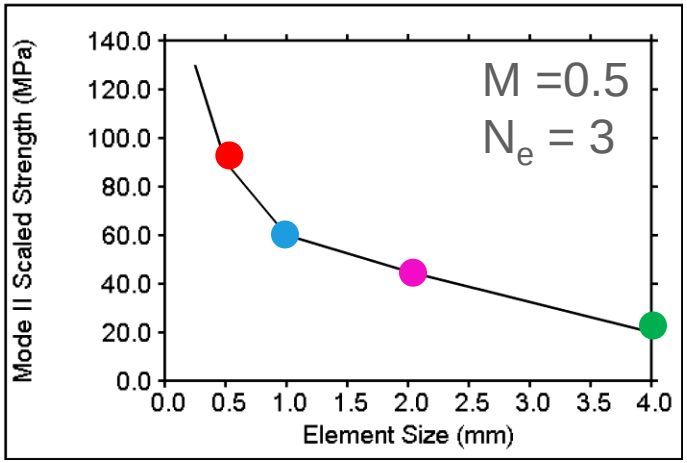
Mesh Dependency – Strength

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

	1	2	3	4	5	6	7
Card 1	MID	RO	ROFLG	INTFAIL	EN	ET	GIC
	138	**1.0E-5**	**0.0**	**0.0**	**1E+5**	**1E+5**	**0.969**
Card 2	XMU	T	S	UND	UTD		
	-2.284	**-30.0**	**-60.0**	**-**	**-**		

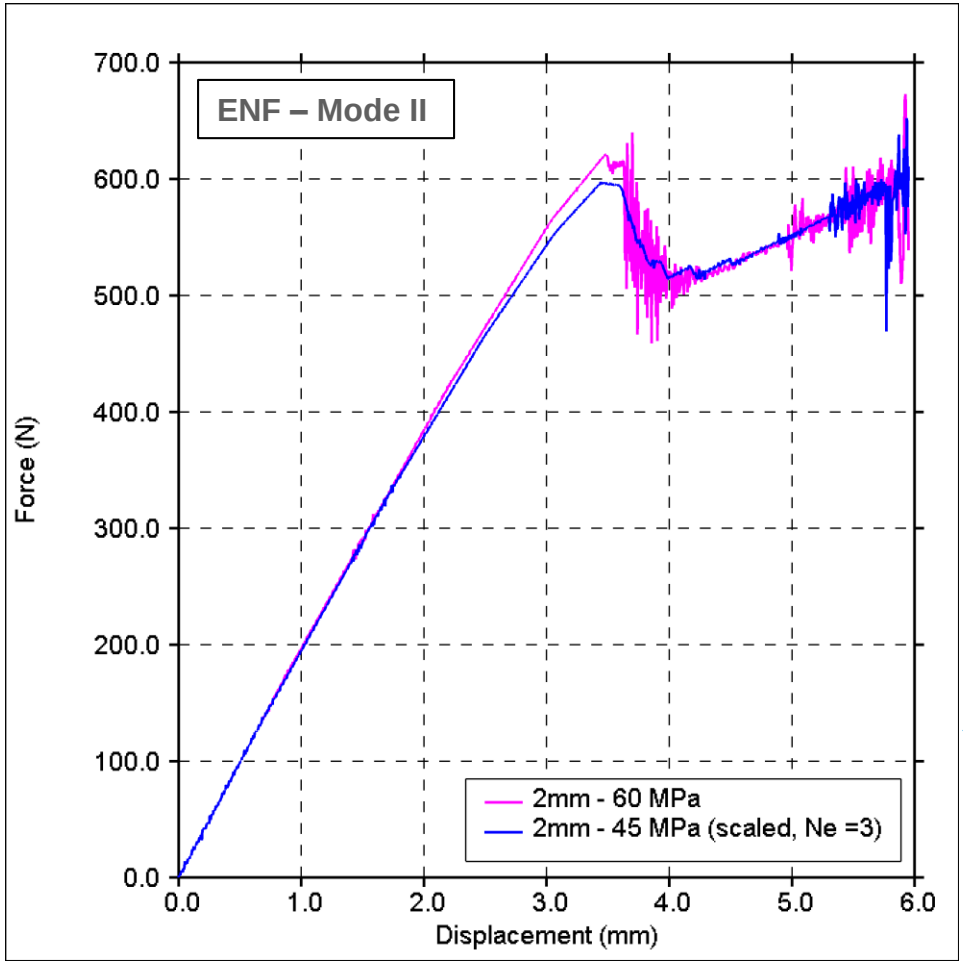


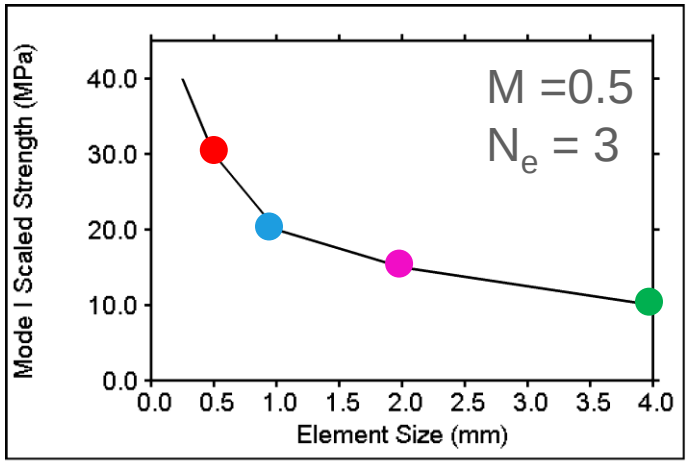


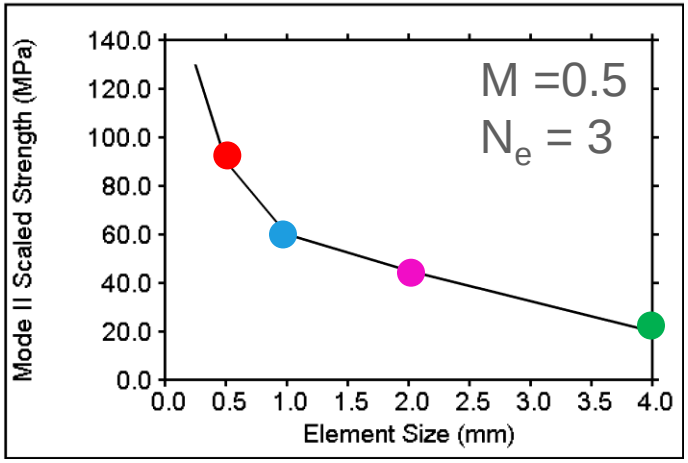


Mesh Dependency – Strength

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

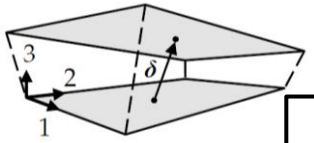






Tiebreak Contacts

- 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**



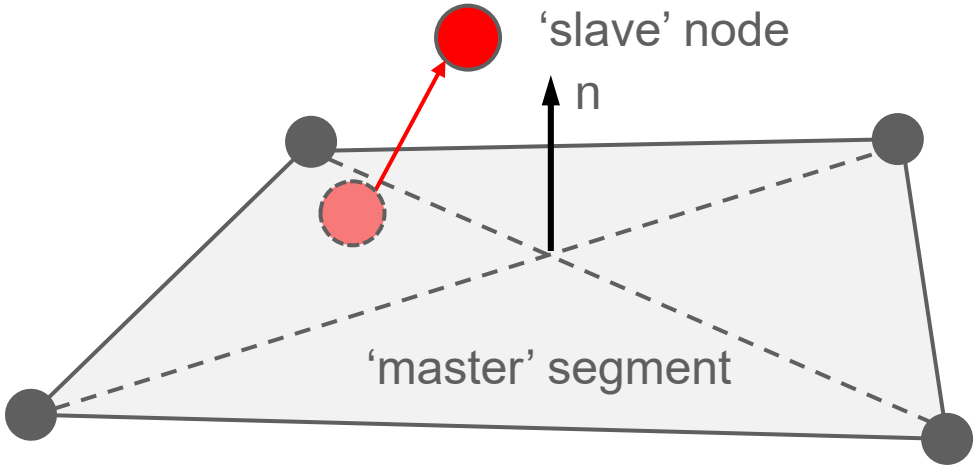
Three components of relative displacements between slave node and master segment

$$\boldsymbol{\delta} = \begin{bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{bmatrix}$$

Shear

Normal

Nodal force output = Contact stresses **x** area of the segment

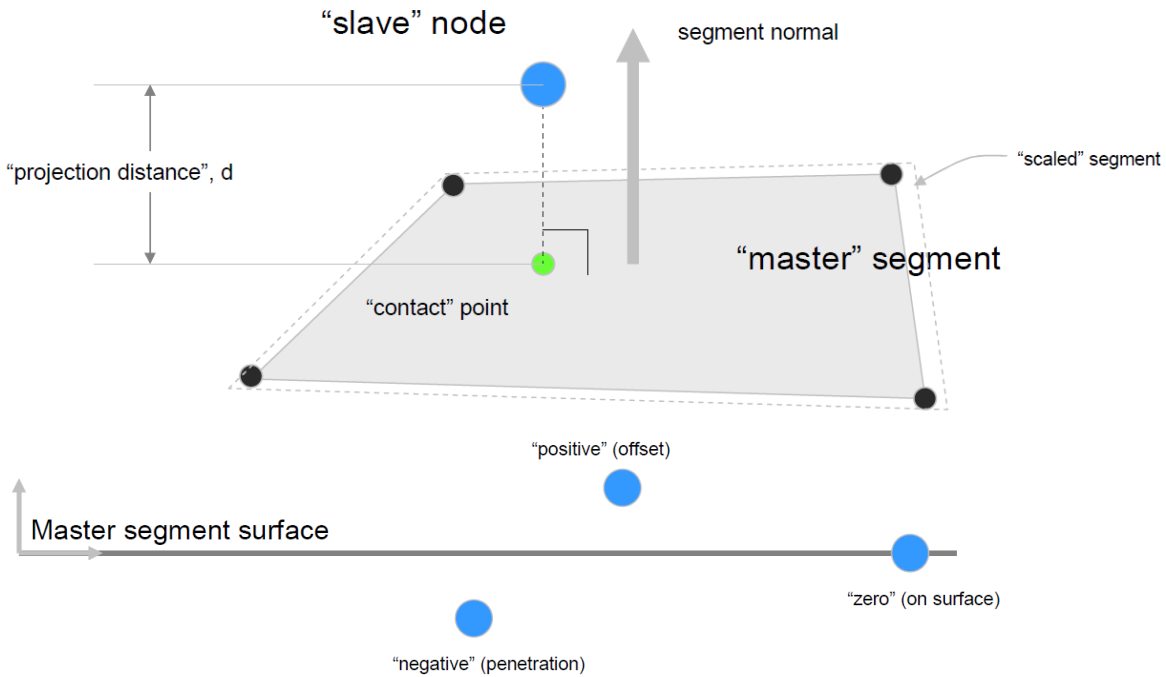


- *<u>Segment:</u>*
  3-noded or 4-noded connectivity references a shell element or one of the faces of a solid element

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.

CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_ TIEBREAK_*OPTION

	1	2	3	4	5	6	7	8
Card 1	SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
Card 2	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Card 3	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Card 4	**OPTION**							

_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

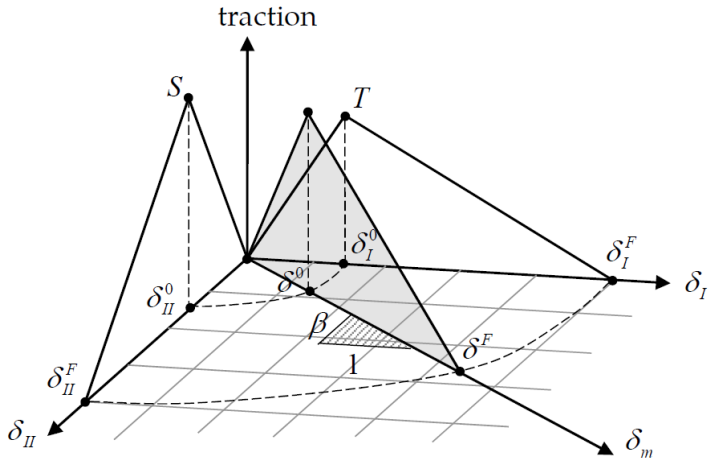
- 14 options available with up to 7 parameters
- Some options apply to offset shell elements (moments are calculated between shell elements)
- **OPTION 9/11** follows same damage initiation and evolution criteria as *MAT_COHESIVE_MIXED_MODE (*MAT_138)

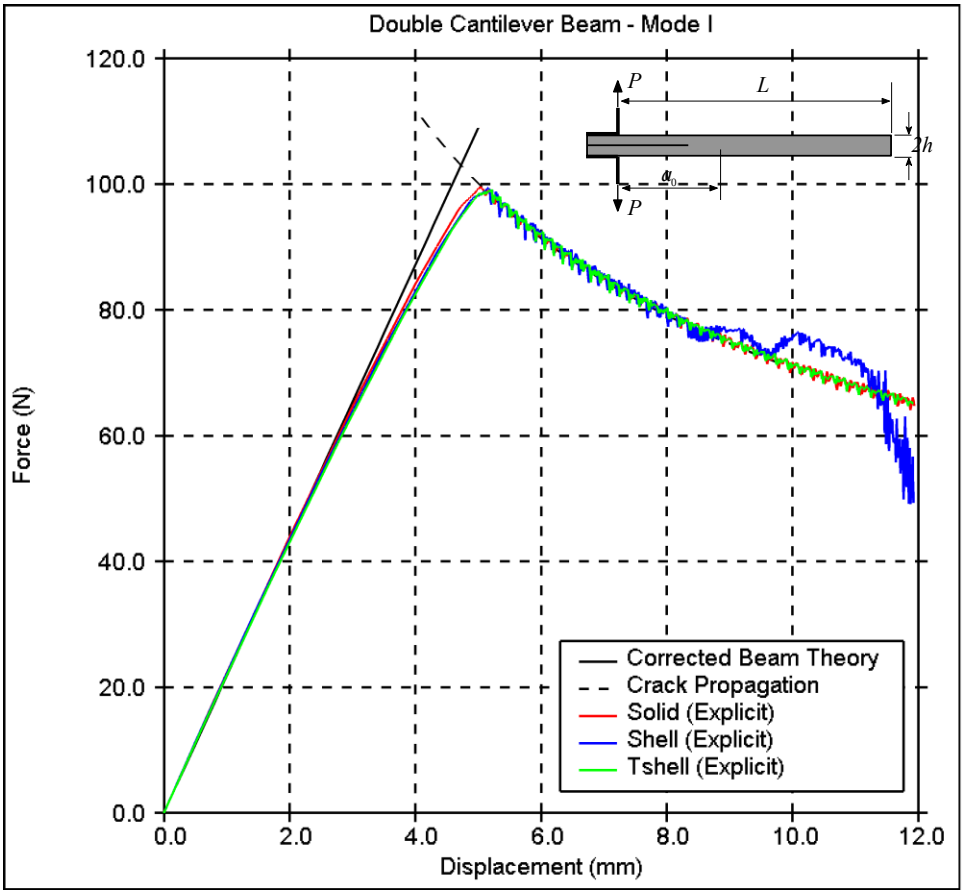
CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_ TIEBREAK_*OPTION

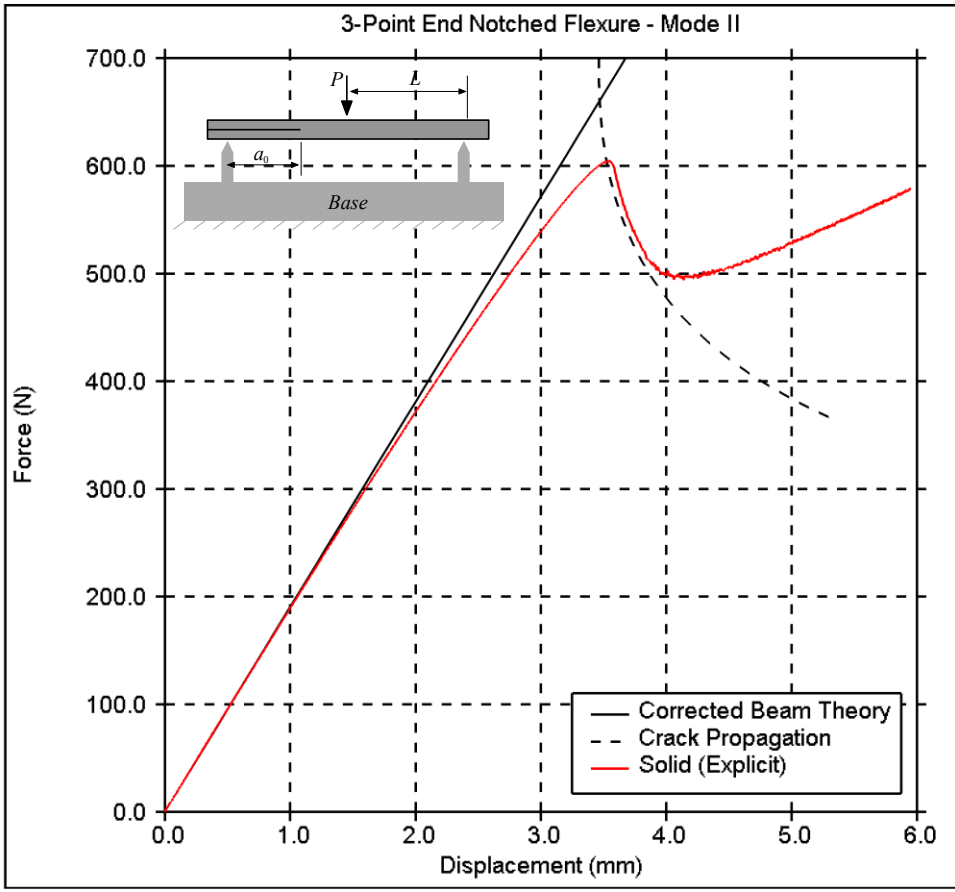
	1	2	3	4	5	6	7	8
Card 1	SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
Card 2	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Card 3	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Card 4	**OPTION**	NFLS	SFLS	PARAM	ERATEN	ERATES	CT2CN	CN

- **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 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
- *CONTACT_*option*_ID

Slave/Master side information is written

	1	2	3			6	7	8
Card 1	**CID**							
Card 2	SSID	MSID	SSTYP	SBOXID	MBOXID	**SPR**	**MPR**	

- *DATABASE_NCFORC

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

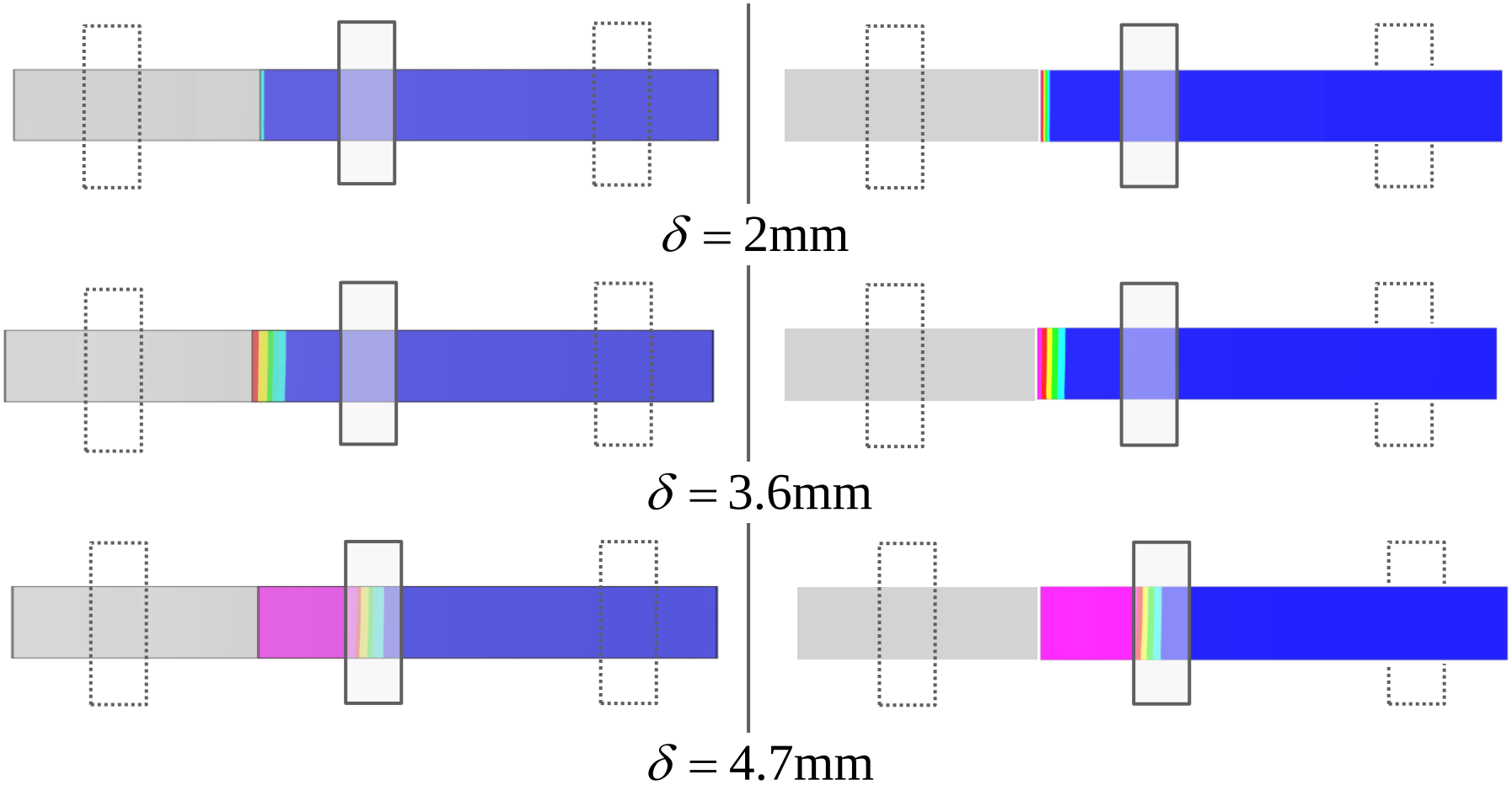
	1	2	3	4	
Card 1	**DT**				

- *DATABASE_BINARY_INTFOR

Binary output file s=filename
- Contact force
- Contact pressure
- Contact gap

	1	2	3	4	5
Card 1	**DT**				

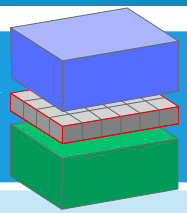
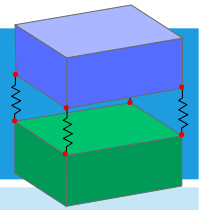
Fringe Plot: Cohesive Elements vs. Tiebreaks



Cohesive Element Model
(Plastic Strain Flag)

TIEBREAK Contact Model
*DATABASE_BINARY_INTFOR

Comparison of Methods*

Cohesive Elements 		Tiebreak Contacts 
Element numbering and orientation, Coincident nodes or tied-contact mesh	**Pre-processing**	Arbitrary ‘partners’, no elements.
Elasto-damage, mixed-mode, user-defined material models (umatXXc in dyn21b.f)	**Constitutive models**	Elasto-damage, mixed-mode
Higher resolution: more smooth results	**Output results**	Lower resolution: More coarse results
Additional contact or ‘keep elements’	**Post-failure response**	Automatically
Element stresses, displacements, damage and failure flags	**Post-processing**	Only nodal values, Intfor: Contact gap

[Liebold et al.,*DYNAmore, 2018]

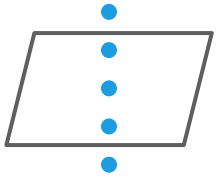
Other recommendations

- 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

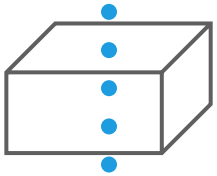
Outline – FE Modelling of Composites

Intra-laminar behaviour

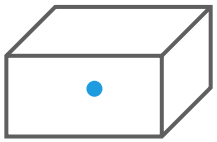
Element



Shells
(2D stress state)



Thick shells
(2D or 3D stress state)



Solids
(3D stress state)

Material

- *MAT_022: COMPOSITE DAMAGE
- *MAT_054/055: ENHANCED_ COMPOSITE DAMAGE
- *MAT_058/158: LAMINATED_COMPOSITE_ FABRIC/RATE_SENSITIVE
- *MAT_059: COMPOSITE_FAILURE_ MODEL
- *MAT_261: LAMINATED_FRACTURE_ DAIMLER_PINHO
- *MAT_262: LAMINATED_FRACTURE_ DAIMLER_CAMANHO
- *MAT_161/162: COMPOSITE_MSC (Commercial)

Structure

- One Shell Layer
- Layers of Solid Elements
- Layers of Shell Elements

Inter-laminar behaviour

- *MAT_138: COHESIVE_MIXED_MODE
- *MAT_184: COHESIVE_ELASTIC
- *MAT_185: COHESIVE_TH
- *MAT_186: COHESIVE_GENERAL
- *MAT_240: COHESIVE_MIXED_MODE_ ELASTO_PLASTIC_RATE
- *CONTACT_TIEBREAK: AUTOMATIC_ONE_WAY_ SURFACE_TO_SURFACE
- *MAT_169: MAT_ARUP_ADHESIVE

ARUP

www.arup.com/dyna

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