Model calibration

Composite Bi-Phase models 30 and 131 for elastic, damage and failure

- PAM-CRASH material model 30 is for solid and 131 for multi-layered shell elements.

- Within these models different ply types can be used for different fibre reinforcements and damage laws. This document covers the ESI composite Bi-Phase damage law for:

  1. Ply type 0 for uni-directional composites (solid and shell elements).

  2. Ply type 8 for woven type composites (shell elements only).

Problem description

Outline	Application of the Bi-Phase composite damage model to a single element under various loadings
Analysis type(s):	Explicit
Element type(s):	Applicable to shell (model 131) and solid elements (model 30)
Materials law(s):	PAM-CRASH material models 30 and 131 (ply types 0 and 8). Fibre and matrix are calibrated for:
	1) Orthotropic elasticity
	2) Orthotropic fibre and matrix damage with failure
Model options:	Boundary conditions, Imposed velocity loads, Composite materials
Key results:	Outputs for composites (fibre stresses and strains, matrix damage, fibre orientations…)
Prepared by: Date:	Anthony Pickett, ESI GmbH/Institute for Aircraft Design, Stuttgart September 2012

1

Background information

Pre-processor, Solver and Post-processor used:

- **Visual-Crash PAM:** To assign control, material, loading, constraint and control data.
- **Analysis (PAM-CRASH Explicit):** To perform the explicit Finite Element analysis.
- **Visual-Viewer:** To evaluate results for time history plots, stresses and strains, etc.

Prior knowledge for the exercise

It is assumed the user has a working knowledge of **Visual** and **PAM-CRASH**.
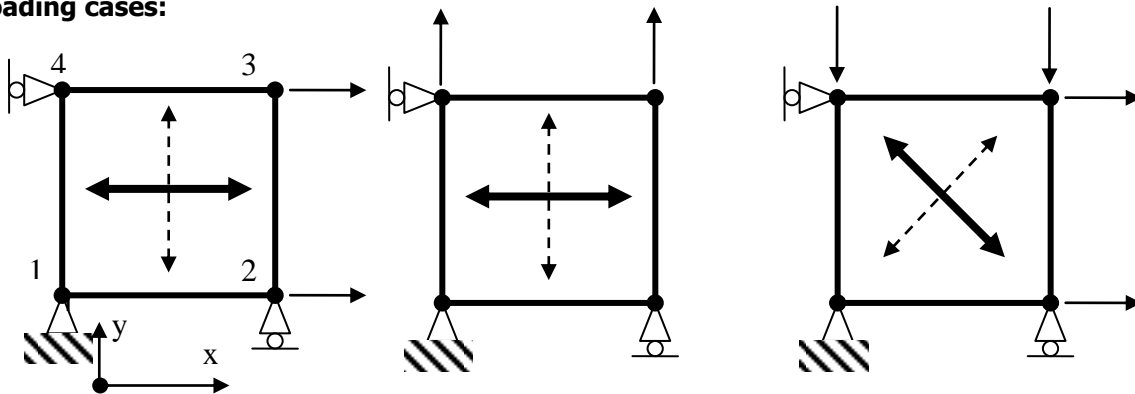
Problem data and description

Units: kN, mm, kg, ms

Description: Single shell element 10mm * 10mm and 1mm thick

Loading: Imposed constant velocity at 1mm/msec (duration of study 1msec) ⇨ total displacement 1mm giving an engineering strain 0.1 (= 10%)

Loading cases:

Case 1: Axial fibre loading   **Case 2: Transverse loading**   **Case 3: Shear loading**

Material: The composite analysed is a UD ply (E_{fibres} = 230 GPa, E_{matrix} = 5 GPa) with an assumed fibre volume ratio (V_f) = 0.5 (=50%).

Assume for simplicity the shear modulus ($G_{12}=G_{23}=G_{13}$) = 7 GPa, and Poisson's ratios are $v_{12}=v_{23}=v_{13}=0.3$. The density is $1.8*10^{-6}$ kg/mm^3.

Supplied datasets

All completed datasets are supplied. The main aim is to demonstrate the strategy for model calibration which is done here using the shell element (material model 131, ply type 0).

Calibration for orthotropic elastic properties:

Fibre direction in tension	**Ply0_TensionX_Elastic.pc**
Fibre direction in compression	**Ply0_CompressionX_Elastic.pc**
Transverse direction in tension	**Ply0_TensionY_Elastic.pc**
Shear loading	**Ply0_ShearXY_Elastic.pc**

Calibration for damage and failure:

Fibre direction in tension (tensile fibre failure)	**Ply0_TensionX_FibreFailure.pc**
Shear loading (matrix shear failure only)	**Ply0_ShearXY_MatrixShearFailure.pc**
Fibre dir. tension (tensile fibre and shear failure)	**Ply0_TensionX_FibreFailure+MatrixShearFailure.pc**
Transverse matrix failure in tension	**Ply0_TensionY_Failure.pc**

Composite models 30 and 131: Ply types 0 and 8 calibration

Basic theory

A full description of the model is given in the ESI PAM-CRASH[TM] user's manual[1]. Briefly, this model separates fibre and matrix phases (Bi-Phase). A classical law of mixtures (Voigt model) is used to combine fibre and matrix for mechanical stiffness in the fibre direction. The transverse and shear data input represents homogenised stiffness.
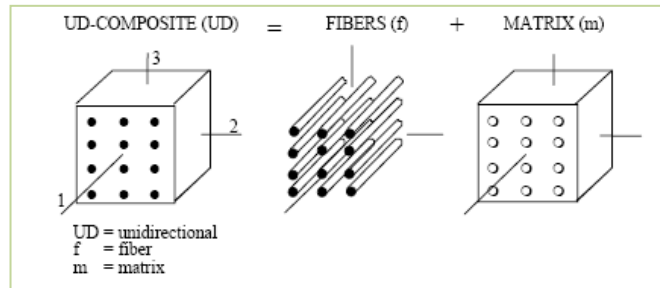

Fig: The Bi-Phase model to separate fibre and matrix phases

One particular advantage of the Bi-Phase approach is that fibre and matrix phases can be damaged (or failed) separately. For fibres a simple strain based damage and failure criteria for tension and compression is used; fibre damage is assumed to be permanent. For matrix the applied strain is converted into first and second strain invariants for hydrostatic and deviatoric (shear) loading respectively. Each of these can then be used to damage the matrix phase according to input values provided by the user. Damage in this model is treated as reduction in the original material stiffness.

$$E(\varepsilon) = (1 - d(\varepsilon)) Eo$$
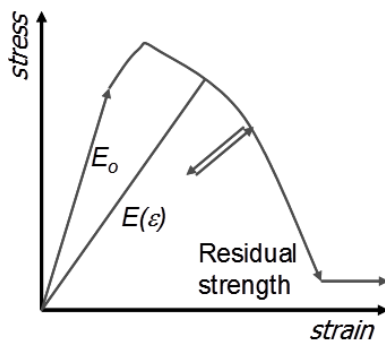
Damage function: this is based on the state of fibre strain, or J1/J2 in the matrix.

Fig: Material damage via stiffness reduction. Three strain levels control damage evolution for initial (start), intermediate (ultimate load) and final (residual or rupture) strain

Failure for any constituent is assumed once damage reaches the limiting value 1.0. An important point is that the remaining constituent (fibre or matrix) can continue to carry load until it also reaches a fully damage state. Alternatively, classical failure criteria such as maximum strain and Tsai-Wu are available which can be used to eliminate the element.

---

[1] PAM-CRASH™ Users manuals, Engineering Systems International, 20 Rue Saarinen, Silic 270, 94578 Rungis-Cedex, France. www.esi-group.com

Composite models 30 and 131: Ply types 0 and 8 calibration

Micro-mechanics in the Bi-Phase model

The Bi-Phase model uses micromechanics for fibre direction (11) only. It is assumed   properties all in other directions (22, 33) and shears (12, 13, 23) are homogenized composite values for both stiffness and failure. This mixed approach was an original idea to allow independent failure for the fibre and matrix parts in the fibre direction.

As an example the following table gives typical fibre and matrix properties and shows how the Bi-Phase model uses micro-mechanics to determine composite stiffness,

	E_{11} (GPa)	E_{22} (GPa)
Carbon fibre	230	15*
Glass fibre	74	74
Matrix	4	4

 * Carbon fibres are highly anisotropic, whereas Glass fibres (and Matrix) are isotropic.

Fibre direction

The Bi-Phase model can be used in various forms, but the intended approach is to use individual properties for stiffness and failure of fibre and matrix in the fibre direction; which also requires knowledge and input of the fibre volume ratio (V_f). The mechanical stiffness is then computed by PAM-CRASH using the standard Law Of Mixtures (LOM),

$$E_{11} = E_{1f}V_f + E_mV_m$$

E.g. using data from the above table PAM-CRASH would compute E_{11} for Carbon or Glass fibre composite to be 117GPa, or 39GPa, respectively (assuming for example V_f=0.5).

Transverse and shear directions

For the transverse directions (E_{22}, E_{33}) and shears (G12, G23, G13) only the homogenized values are input for stiffness and failure. These are found from coupon tests, or could be computed using constituent properties and appropriate law of mixtures formulae. For example E22 uses the standard inverse LOM,

$$\frac{1}{E_{22}} = \frac{V_f}{E_{2f}} + \frac{V_m}{E_m}$$

E.g. using data from the above table E_{22} (= E_{33}) for Carbon and Glass composite equals 6.32GPa and 7.59GPa respectively (assuming V_f=0.5).

Note:

1. The inverse law of mixtures gives poor estimates for homogenized stiffness and better models are available such as 'Chamis' and 'Halpin-Tsai'. An Excel based program can be used for some calculations,

    (http://www.ifb.uni-stuttgart.de/FEM/TutorialsAndMaterialCalibration.htm#software

2. Similar law of mixtures formulae are available for homogenized shear and Poisson's ratio values.

Composite models 30 and 131: Ply types 0 and 8 calibration

The following gives typical PAM-CRASH ply elastic data for a uni-direction Carbon fibre-Epoxy composite; some points to note,

- A fibre modulus (Eft) 230GPa is assumed in the tension direction, with slightly reduced value 180GPa in compression.

- In this example a fibre volume ratio 0.5 is assumed.

- The matrix modulus Et11 (tension and compression) is 4GPa. PAM-CRASH will use this data to compute composite stiffness (E11) in the fibre direction.

All other elastic properties (transverse moduli, all shear stiffness's and Poisson's ratios) are homogenized values from coupon tests, micromechanics formulae, or some other source.

Some other points to note:

- The fibre properties can be zeroed and stiffness defined only via Et11 and Ec11; but this approach would prevent possibilities to control independent fibre failure.

- The above input description is for Ply Typ 0 having a single fibre direction. A similar logic applies to Ply type 8 for woven fabrics (see Part 5) having two orthogonal ply directions. That is, two fibres are defined with individual fibre and matrix properties, with all other properties being homogenised values.

Part 1: Basic input data

Typical model loadings and supports

(e.g. for dataset Ply0_TensionX_Elastic.pc which is loaded in the positive x-direction)

Open the dataset in **Visual-Crash PAM** (VCP) and study the supports and the loading:

1. In the **Loads > Displacement Boundary Condition** definitions you will find:
   - Node 1 is fully constrained.
   - Node 2 is fully fixed, except free in the x-direction.
   - Node 4 is fully fixed, except free in the y-direction.

2. In the **Loads > 3D BC** nodes 2 and 3 are loaded at a constant velocity in the positive x-direction (for tension fibre loading).
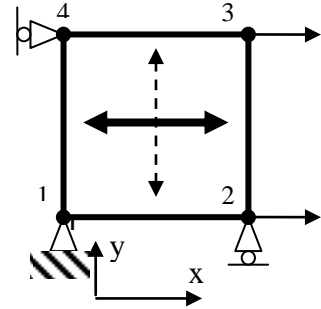
Remark: This simple setup ensures the element is properly supported and has pure tension/ compression loading in the x-direction with free contraction/expansion in the y-direction.

Ply, Material and Part data cards

Note in the dataset:

1. **The parts** data contain information on geometric data such as the reference fiber orientation. A link to associated material data (variable **IMAT**) is also made.

2. **The materials** cards (**IMAT**) contain information on density, the number of plies in the laminate and stacking sequence (for shells only). Within this stacking sequence information on each ply includes the reference ply set (parameter **IDPLY**), the ply thickness and a fibre orientation angle.

3. **The ply** cards (**IDPLY**) contain all ply mechanical data for stiffness, damage and failure.

Considering the sequence in which the data refers to each other (1→2→3) it is best to define things in reverse order, i.e. ply data first, then materials and finally parts; all three are needed.

Ply cards (e.g. Ply elastic properties for dataset Ply0_TensionX_Elastic.pc).

In the ply cards you will find mechanical and optional damage/failure data assigned to the matrix and fibres. ITYP is the ply type (0 or 8).

o Note the fibre and matrix data for (dir 11) are constituent values; PAM-CRASH uses to Law of Mixtures (LOM) to compute composite values.

o For all other stiffness, shear and Poisson's ratio values the composite homogenised values are input.

PLY /					
	IDPLY	ITYP	RHO	IFAIL_INP	ISTRAT
PLY /	1	0	1.8E-6	0	==
	TITLE				
NAME	UD ply data				
Et11	Et22	Et33			
4.	6.3	6.3			
Gt12	Gt23	Gt13	NUt12	NUt23	NUt13	ISH	ITR
2.5	4.2	4.2	0.25	0.25	0.25	==	==
EPSlmtsi	EPSlmts1	EPSlmtsu	Dmts1	Dmtsu	IFUNmts
0.	0.	0.	0.	0.	0
EPSlmtvi	EPSlmtv1	EPSlmtvu	Dmtv1	Dmtvu	IFUNmtv
0.	0.	0.	0.	0.	0
Eft	ALPHAft	EPSlfti	EPSlft1	EPSlftu	Dft1	Dftu	IFUNft
230.	0.5	0.	0.	0.	0.	0.	0
KAPPA23	KAPPA13	ALPHA1	ALPHA2
Ec11	Ec22	Ec33
4.	6.3	6.3
Gc12	Gc23	Gc13	NUc12	NUc23	NUc13
2.5	4.2	4.2	0.25	0.25	0.25
EPSlmcsi	EPSlmcs1	EPSlmcsu	Dmcs1	Dmcsu	IFUNmcs
0.	0.	0.	0.	0.	0
EPSlmcvi	EPSlmcv1	EPSlmcvu	Dmcv1	Dmcvu	IFUNmcv
0.	0.	0.	0.	0.	0
Efc	ALPHAfc	ALPHAfi	ALPHAf1	ALPHAfu	Dfc1	Dfcu	IFUNfc
180.	0.5	0.	0.	0.	0.	0.	0

E.G. for fibre tension the LOM gives $E_{11t} = E_{ft} * V_f + E_{t11} * (1 - V_f) = 117$GPa

where E_{ft} is fibre modulus (230GPa)

E_{t11} matrix modulus (4GPa)

V_f fibre volume ratio (ALPHAft =0.5)

Material cards (e.g. Material properties for dataset Ply0_TensionX_Elastic.pc).

1. The number of plies in the laminate are defined (in this case NOPER=1) and for each ply an associated ply number (IDPLY), the ply thickness (THKPL) and fibre angle (ANGPL) are specified. Note the fibre angle is with respect to a reference vector defined in the Parts cards (see below).

Material Editor							
ID: 1	Element Type: All(Struct)	Type: 131-MULTILAYERED_ORTHOTROPIC_BI_PHAS					
$---5---10---5---20---5---30---5---40---5---50---5---60---5---70---5---80$							
MATER /							
	IDMAT	MATYP	RHO	ISINT	ISHG	ISTRAT	IFR
MATER /	1	131	1.8E-6	0	0	0	
BLANK	AUXID1	AUXID2	AUXID3	AUXID4	AUXID5	AUXID6	QV
	0	0	0	0	0	0	1.
	TITLE						
NAME	UD ply						
KSI	Fo	NOPER	ILAY	HGM	HGW	HGQ	As
0.1	0.	1	0				
IDPLY	THKPL	ANGPL					
1	1.	0.					
BLANK	NMIN	BLANK	GRUC_KW	GRUC_VAL	IFAIL	BLANK	ERA
	0		___	___	0		
IDPLY1	IDAUX1	IDPLY2	IDAUX2	IDPLY3	IDAUX3	IDPLY4	IDA
1	1	1	2	1	3	1	6
IDPLY9	IDAUX9	IDPLY10	IDAUX10	IDPLY11	IDAUX11	IDPLY12	IDA

2. Special output results to the **.THP** and **.DSY** files (or **.ENF** file if used) are possible such as fiber, transverse and shear stresses/strains. For this the ply number in the laminate lay-up and an auxiliary reference number for the variable to be output are given. Examples of auxiliary reference numbers are Aux's 1-3 for fibre strain11, strain22 and shear strain12, and Aux's 6-8 for corresponding fiber stresses. Auxiliary reference numbers are also available for results such as fibre and matrix damage.

Part cards (e.g. Part properties for dataset Ply0_TensionX_Elastic.pc).

1. A connection to the associated material is made via the **IDMAT** number (in this case = 1).

2. A vector for the reference fibre direction is defined. Here the global vector option with a reference vector 1,0,0 (i.e. the global x-direction) is used. For shell elements the transverse direction is normal to the fiber direction in the element plane.

PART /					
	IDPRT	ATYPE	IDMAT	IDVAMAT	IDTHMAT	IDPMAT
PART /	1	SHELL	1	0	0	0
	TITLE					
NAME	PART_1					
DTELIM	TSCALF					
TCONT	EPSINI	COULFRIC				
H	NINT	OFFSET	NINTh			
1.	3					
IORT	BLANK	XDIR	YDIR	ZDIR		
0		1.	0.	0.		
END_PART						

3. For solid elements two reference vectors are required.

Part 2: Calibration of orthotropic elastic properties

Tension

Using dataset   Ply0_TensionX_Elastic.pc   and the results files.

Create a cross plot time history of shell element Aux4 versus Aux1 for element stress11 versus element strain11 in the fibre frame. The slope of this curve is the composite stiffness (=117 GPa) in the fibre direction. This approximately corresponds to the composite stiffness in the fibre direction using the law of mixtures.

Note: The element maximum strain (x-axis) is 0.095 which is the natural logarithmic strain (not conventional engineering strain). All input information and output results use natural logarithmic strains.

Engineering strain: The imposed velocity gives a nodal displacement of 1.0mm. For an initial element length of 10mm the engineering strain is,

$$\Delta L/Lo = 1.0/10.0 = \underline{0.1}$$

Logarithmic strain: Is defined as,
$$\ln(L/Lo) = \ln(11.0/10.0) = \underline{0.0953}$$
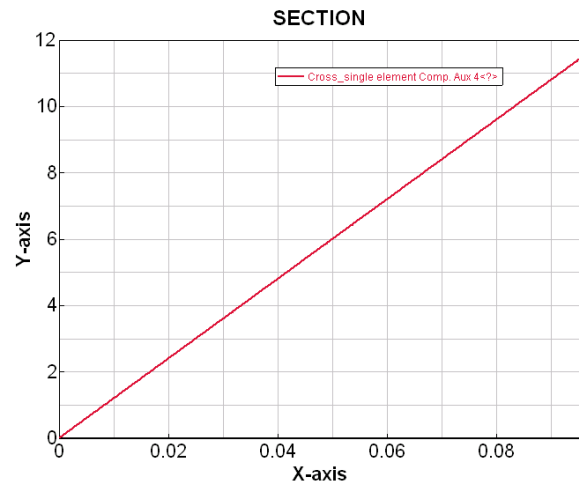
Compression

Using dataset   Ply0_CompressionX_Elastic.pc   and the results files.

The dataset is identical to the tension case above, except that compression loading is applied in the fibre direction (imposed using a scale factor of -1 in the appropriate **3D BC** cards).

To demonstrate the possibility for different compression stiffness the compression fibre modulus is set to 180 GPa (tension = 230 GPa); all other mechanical data is unchanged.

The curve shown uses the above tension curve and the **Chase Curve(s)** option with file Ply0_CompressionX_Elastic.THP to add the new compression curve result.

The compression stiffness approximately corresponds to the value computed from the compression law of mixtures.

8
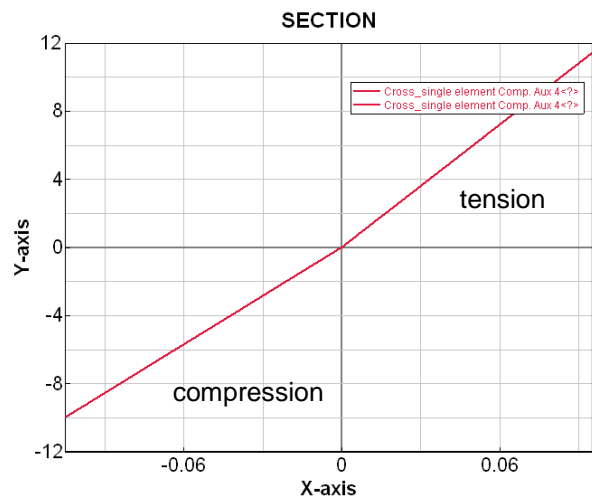
Transverse tension

Using dataset   Ply0_TensionY_Elastic.pc   and the results files.

In this dataset velocities are applied to nodes 3 and 4 in the positive y-directions giving an engineering strain of 0.1 (=0.095 logarithmic). The nodal boundary conditions are unchanged.

Create a cross plot time history of shell element Aux5 versus Aux2 for element stress22 versus element strain22 in the fibre frame. The transverse stiffness is the slope of this curve ($\approx$0.66/0.095 = 6.9GPa).

Note: The slope is not quite equal to this input modulus (6.3GPa) due to interaction of stress11 and stress22 via Poisson's effect.
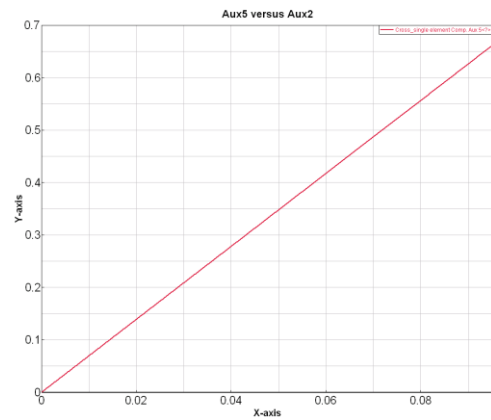
Shear

Using dataset   Ply0_ShearXY_Elastic.pc   and the results files.

For pure shear loading velocities are now applied to nodes 3 and 4 in the negative y-direction and to nodes 2 and 3 in the positive x-direction using the same function curve, also the fibre directions are rotated 45°. All other boundary conditions are unchanged.

Create a cross plot time history of shell element Aux6 versus Aux3 for element shear stress12 versus element tensor shear strain12 in the fibre frame.

Note: PAM-CRASH input and output use 'tensor' shear strain (ε_{12}) which is related to 'engineering' shear strain by,

$$\gamma_{12} = 2\varepsilon_{12}.$$

The shear modulus is the slope of the plot of shear stress12 versus 'engineering' shear strain12 giving,

G$_{12}$ = 0.5 / (2*0.1) = 2.5 GPa

which is the same as the ply homogenised input value (Gt12 = 2.5GPa).

9
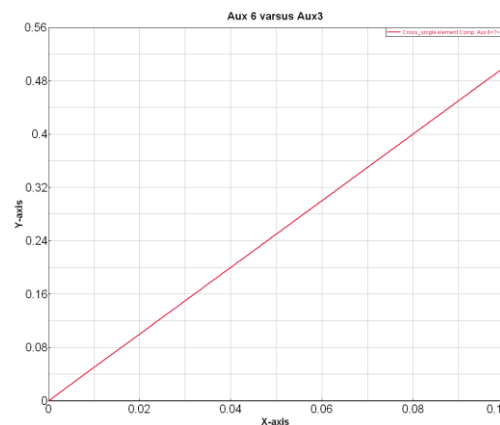
Part 3: Calibration of orthotropic damage and failure properties
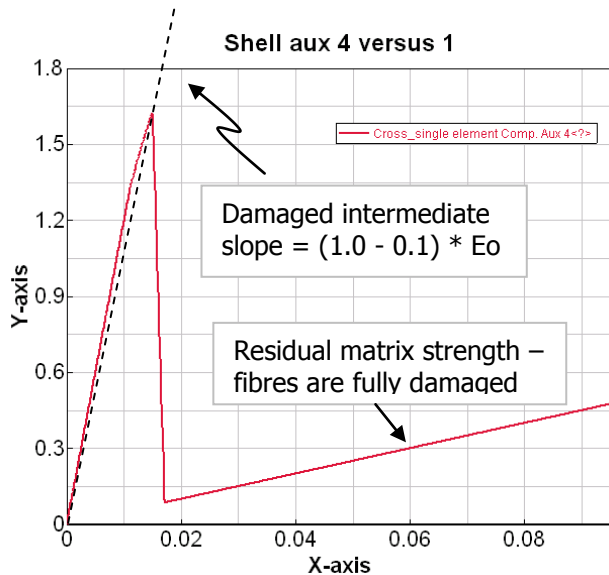
Tension (and compression) fibre failure

Using dataset   Ply0_TensionX_FibreFailure.pc and the results files.

First fibre failure is defined. This is calibrated against a coupon test that measures tension fibre failure (i.e. a uni-axial 0° tension coupon test). For compression properties an identical calibration is followed fitted to a uni-axial 0° compression coupon test.

The following data is obtained from test measurements. Strains for limit linear behaviour, ultimate load and final rupture are 0.011, 0.015 and 0.017 respectively. The intermediate and final damages are 0.1 and 1.0.

Create a cross plot time history of shell element Aux4 versus Aux1 for element stress11 versus

element strain11 in the fibre frame. At inter-mediate strain (0.015) the modulus is damaged by 10% (0.1). Thereafter the fibres rapidly damage to ultimate failure at 0.017 strain. The progressive rupture is useful in a simulation to steady damage evolution and helps damp the solution. The residual elastic stress shown is the matrix contribution which is not defined to damage in this data set.

Shear loading for matrix shear failure

Using dataset   Ply0_ShearXY_MatrixShearFailure.pc   and the results files.

Matrix shear damage is now defined. This would be calibrated against a coupon test that loads the composite for intra-ply matrix shear failure (e.g. axial loading of a ±45° coupon). In this case matrix damage is controlled by shear strains and shear damage data.

For this example matrix 'tensor' shear damage strains (tension and compression) are set to 0.03, 0.04 and 0.05 with intermediate and final damage 0.1 and 1.0 respectively.

Create a cross plot time history of shell element Aux6 versus Aux3 for element stress12 versus element strain12 in the fibre frame. The undamaged, partial damage and final rupture zones are visible between shear strains 0.→0.03, 0.03→0.04 and 0.04→0.05 respectively; final matrix failure occurs at strain 0.05.
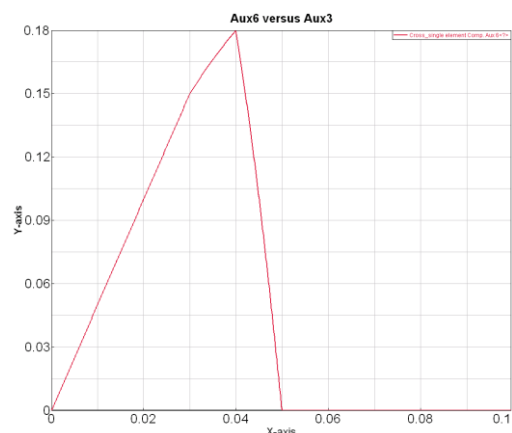
Note: The Bi-phase model also allows hydrostatic matrix damage. This is difficult to test and unlikely to happen in practice; it is therefore easiest to set all hydrostatic damage to zero and calibrate only shear damage.

# Composite models 30 and 131: Ply types 0 and 8 calibration

Fibre direction tension (tensile fibre and shear failure)

Using dataset   Ply0_TensionX_FibreFailure+MatrixShearFailure.pc   and the results files.

In this dataset the previous fibre and matrix failure data are combined. Loading is tension in the fibre direction.

Create a cross plot time history of shell element Aux4 versus Aux1 for element stress11 versus element strain11 in the fibre frame. Results show that fibre failure occurs first (due to the lower failure strain assigned), followed by matrix rupture.

Note: Experimental testing may not show this post-fibre failure response. However, the model is only a idealisation. Also, tests are conducted under load control, whereas the simulation uses velocity/displacement loading; these two loading conditions do create different post failure behaviours.
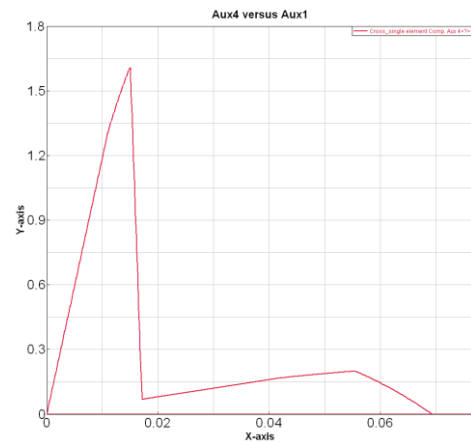
Transverse matrix failure in tension

Using dataset   Ply0_TensionY_Failure.pc and the results files.

Transverse matrix failure should be checked against results from a 90° tension test. In this study the combined matrix (shear) and fibre failure data are used with the element loaded in transverse tension.

Create a cross plot time history of shell element Aux5 versus Aux2 for element stress22 versus element strain22 in the fibre frame. Results show the matrix shear failure data is able to reasonably represent matrix failure under tension loading.
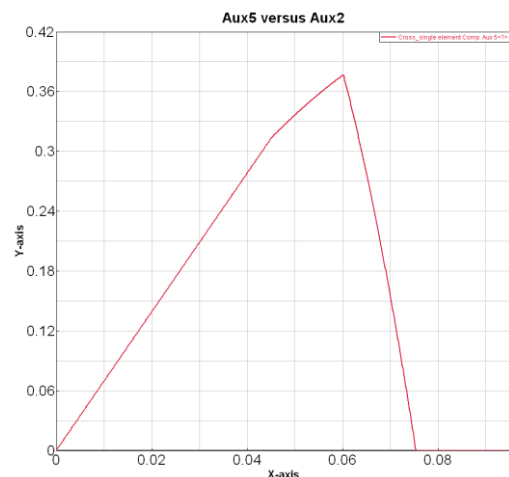
Note: A limitation of the Bi-Phase model is that matrix normal (tension or compression) failure and matrix shear failure cannot be decoupled; they are linked via the shear strain invariant failure criteria.

Remarks:
1. For the complete composite material model all data should be combined for mechanical stiffness, fibre failure and matrix failure; for both tension and compression.
2. It is straightforward to assign different mechanical and failure data for tension and compression. Composites are generally weaker in compression and this can be easily treated.
3. The fibre directions can be visualized. Load the .pc file into Visual-Crash and click on **Checks > Display fibre orientations**.
4. Auxiliary variables 11, 12 and 13 give other useful output for total, shear and volumetric damage; auxiliary 15 is fibre damage. These variables are available for contour and time history plots.

Part 4: Alternative options for damage and failure

<u>Damage curve functions</u>

For fibre and matrix damage the shape of the damage curve has so far been defined using initial, intermediate and final strain values together with intermediate and final damage values. The model assumes parabolic curves between points and can reasonably fit mildly non-linear behaviour. However, the fit becomes difficult for large strain highly non-linear behaviour.
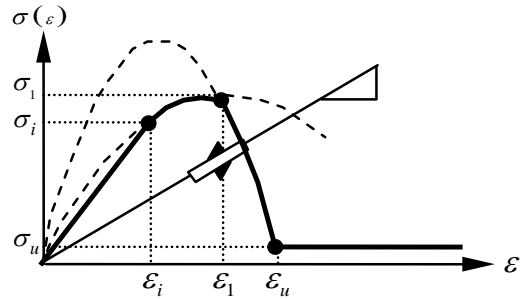

For this reason alternative damage evolutions via curve functions (IFUNxxx) are possible. The curves are damage versus strain,

i.e. $(1 - E_D/E_O)$

or $(1 - G_D/G_O)$ versus strain.

A typical damage curve function is shown (x is strain information and y is damage information). The curve starts at zero with an initial undamaged zone for linear behaviour. The final damage in this case reaches 1.0. In this case 5 points on the test stress-strain curve have been selected to construct the damage function curve.
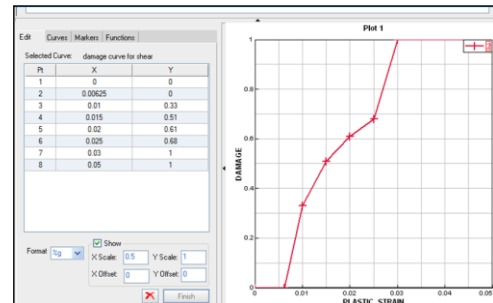
<u>Superimposed failure criteria</u>

If fibre (or matrix) damage reaches 1.0 then that constituent is fully damaged and offers no resistance to loading; the other constituent can carry loading until it is fully damaged.

In addition it is also possible to superimpose other failure criteria such as Tsai-Wu, Max. stress, etc. Set IFAIL_INP=1 and choose the failure criteria with flag FAILTYP. Depending on the selected failure criteria additional failure information is required.

If the state of stress or strain reaches the failure criteria the element will be eliminated (if FAILDAM = 1). In the materials cards the IFAIL option must also be set.

Part 5: Specific information for woven reinforcement (Mat 131 ply type 8)

The data shown is typical information for ply type 8. Some points to note are:

1. The structure and general information required is similar to ply type 0.

2. IMPORTANT: Fibre volume ratios are used in the LOM to get E11 (using VOLRf1) and E22 (using VOLRf2). Typical values are 0.5 for each.

3. Main fibre direction 11 is with respect to the reference axis defined in the Part cards and the angle in the Materials cards. The second fibre direction is by default at 90°, or as specified by ANG2.

PLY /							
	IDPLY	ITYP	RHO	IFAIL_INP	ISTRAT		
PLY /	1	8	1.8E-6	0	---		
	TITLE						
NAME	UD ply data						
Et11	Et22	Et33					
4.	6.3	6.3					
Gt12	Gt23	Gt13	NUt12	NUt23	NUt13		
2.5	4.2	4.2	0.25	0.25	0.25		
EPSImtsi	EPSImts1	EPSImtsu	Dmts1	Dmtsu	IFUNmts		
0.03	0.04	0.05	0.1	1.	0		
EPSImtvi	EPSImtv1	EPSImtvu	Dmtv1	Dmtvu	IFUNmtv		
0.	0.	0.	0.	0.	0		
Eft1	GAMMAt1	EPSIft1i	EPSIft11	EPSIft1u	Dft11	Dft1u	IFUNft1
230.		0.011	0.015	0.017	0.1	1.	0
Eft2	GAMMAt2	EPSIft2i	EPSIft21	EPSIft2u	Dft21	Dft2u	IFUNft2
230.		0.011	0.015	0.017	0.	0.	0
As	ANG2	VOLRf1	VOLRf2	IDAMCR	IWARP	ALPHA1	ALPHA2
		0.5	0.5	---	---		
Ec11	Ec22	Ec33					
4.	6.3	6.3					
Gc12	Gc23	Gc13	NUc12	NUc23	NUc13		
2.5	4.2	4.2	0.25	0.25	0.25		
EPSImcsi	EPSImcs1	EPSImcsu	Dmcs1	Dmcsu	IFUNmcs		
0.03	0.04	0.05	0.1	1.	0		
EPSImcvi	EPSImcv1	EPSImcvu	Dmcv1	Dmcvu	IFUNmcv		
0.	0.	0.	0.	0.	0		
Efc1	GAMMAc1	EPSIfc1i	EPSIfc11	EPSIfc1u	Dfc11	Dfc1u	IFUNfc1
180.		0.011	0.015	0.017	0.1	1.	0
Efc2	GAMMAc2	EPSIfc2i	EPSIfc21	EPSIfc2u	Dfc21	Dfc2u	IFUNfc2
180.		0.011	0.015	0.017	0.	0.	0