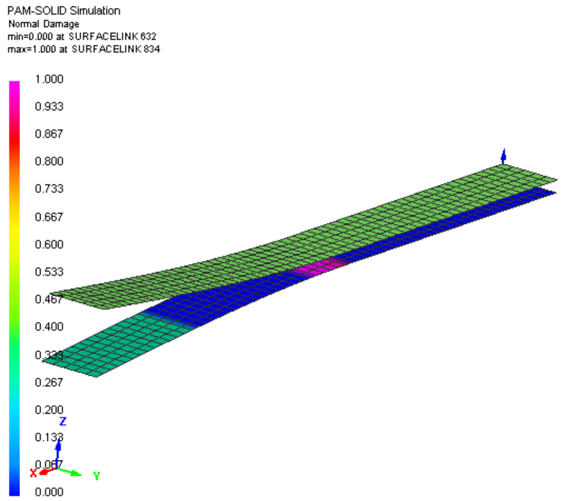
Tutorial 11

Delamination Modeling of a Composite Double Cantilever Beam (DCB) Specimen



Problem description

Outline	The Mode I crack propagation in a composites DCB specimen is modelled using the delamination contact interface.
Analysis type(s):	Explicit
Element type(s):	Multilayered composite shell and delamination interface
Materials law(s):	Composite and delamination
Model options:	Boundary conditions, contact, velocity loading
Key results:	Delamination growth
Prepared by:	Anthony Pickett, ESI GmbH/Institute for Aircraft Design, Stuttgart
Date:	July 2007
Version:	V5 (updated December 2012 for Visual-Crash PAM V8.0)

Background information

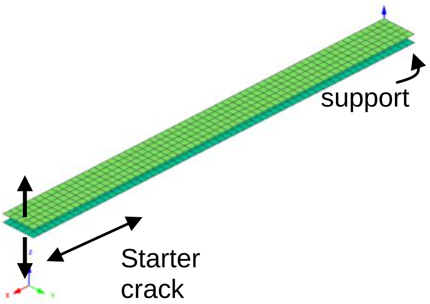
Pre-processor, Solver and Post-processor used:

- **Visual-Mesh:** For generation of the geometry and meshes.
- **Visual-Crash PAM:** To assign control, material data, loadings, constraints and time history (control) data.
- **Analysis (PAM-CRASH Explicit):** To perform an explicit Finite Element analysis.
- **Visual-Viewer:** Evaluating the results for contour plots, time histories, etc.

Prior knowledge for the exercise

It is assumed that Tutorials 1,2 and 3,4 have been worked through. In order to avoid unnecessary repetition some explanations on use of Visual for creation of entities will be kept rather brief, whereas some new options will be explained in more detail.

Problem description



Unit System: mm, kg, ms

Dimensions: Composite laminate 250mm long, 20mm wide, 8.4 mm thick and 50mm starter crack.

Loading: Imposed velocity (±2mm/msec) at the free ends of the starter crack. The other end is fully constrained.

Modeling: Two sets of composites shells represent the top and lower halves of the laminate. These are then joined (in the uncracked portion only) by a delamination tied interface.

Composite: Each laminate half is 4.2 mm thick and modelled using single multilayered shell elements. The material is Biaxial NCF/Epoxy resin with 16 plies quasi-isotropic layup [0/90/90/0]2s for <u>each side</u> of the starter crack.

Supplied datasets

The finite element mesh (in PAM-CRASH format) is supplied for this problem. Copy the mesh file to a model file which will be used to build the analysis model using **Visual-Crash PAM**.

Copy :

DCB_Delamination_Mesh.pc to **DCB_Delamination_Model.pc**

This will allow the work to be repeated if the model definition phase goes wrong.

Tutorial 11: Delamination modeling of a Double Cantilever Beam specimen

Basic theory

A full description of the delamination models are given in the ESI PAM-CRASH™ user's manual[1]. Briefly, the delamination interface is idealised using simple interface spring elements that transmit traction normal and shear forces between adjacent plies. Each interface is defined using 'Tied Link' interfaces (type 303) that will automatically generate connection spring elements from information on the adjoining plies. The plies may be represented using either composite solid (with a separation distance for the interface) or composite shell elements.

Essentially, the interface spring elements represent mechanical stiffness, strength and fracture energy absorption of the interface using the idealized stress-displacement response shown below, for example, for normal (Mode I) loading. A simple linear elastic law is assumed up to the failure stress σ_{Imax}, thereafter linear damaging is activated such that at final separation δ_{Imax}, the critical energy release rate G_{Ic} of the tied interface is absorbed. PAM-CRASH will compute δ_{Imax} from input information G_{Ic} and σ_{Imax} so that the correct energies are absorbed at full rupture. Identical arguments are used to define shear (Mode II) response. A simple maximum normal and maximum shear stress criteria is available which must be reached before the crack propagation criteria is active.

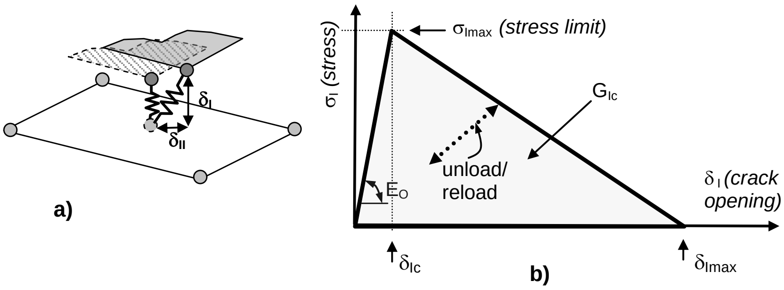


Fig: The inter-ply delamination model:
a) An attached node ('slave' node) constrained to an element 'master' surface
b) Diagram of the stress-crack opening curve for Mode I loading

The values of G_{Ic} and G_{IIc} are obtained from the standard Mode I Double Cantilever Beam (DCB[2]) and Mode II End Notched Flexure (ENF[3]) tests respectively. Also, coupling for mixed mode critical fracture toughness can be determined using the Mixed Mode Bending (MMB[4]) test.

Mixed mode interaction is often defined via the relationship,

$$\left(\frac{G_I}{G_{Ic}}\right)^{\eta} + \left(\frac{G_{II}}{G_{IIc}}\right)^{\eta} = 1,$$

where η=1 for linear and η>1 for nonlinear interaction. In PAM-CRASH a nonlinear interaction is possible to define via a curve function.

---

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

[2] ISO international standard DIS15024, Fibre-reinforced plastic composites – Determination of Mode I interlaminar fracture toughness, GIC, for unidirectionally reinforced materials.

[3] ASTM draft standard D30.06: 'Protocol for interlaminar fracture testing, End-Notched Flexure (ENF)', revised April 24, 1993.

[4] Reeder J R and Crews J H, Mixed-mode bending method for delamination testing, AIAA Journal, Vol. 28, No. 7, pp 1270-1276, 1989.

Using VCP to make the analysis model

Start Visual-Crash PAM (VCP) and read in the mesh:

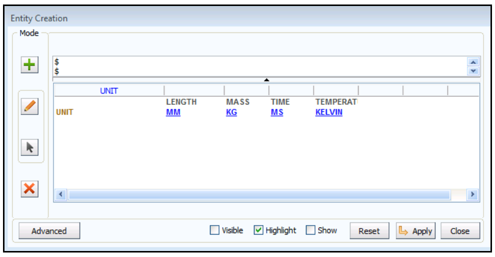
Select **File > Open** and open the file **DCB_Delamination_Model.pc**

Specify the model units system

Set the model units system by selecting **Crash > Optional Controls > Units** to open the adjacent panel,

Set the units to **mm, kg, ms** and **Kelvin.**

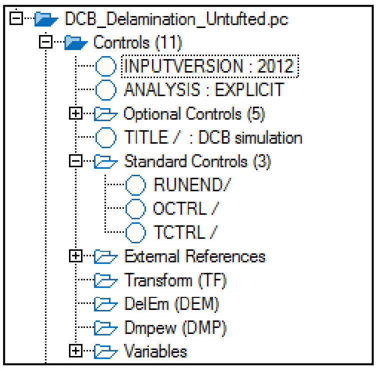
Finish with **Apply** and **Close.**



Defining some basic model data

For the PAM- controls the following parameters are set:

INPUT-VERSION	The PAM-CRASH version being used (e.g. **2012**)
RUNEND	Termination time for the analysis **(=10.0 msec)**
TITLE	The title used that will appear on all output plots
SOLVER	Use **CRASH** for a PAM-CRASH analysis
TCTRL	The nodal timestep (NODTSP) option is activated for the delamination interface with a minimum timestep (DTMASS = 0.0001) specified. Beware this adds mass and should be used carefully
OCTRL	• THPOUTPUT – output interval for graphical (x,y) time history information (e.g. use **POINTS = 1000** for one thousand points) • DSYOUTPUT - output interval for deformed states (e.g. use **INTERVAL = 0.25**) for (RUNEND=10.0)/0.25 = 40 pictures • Parameter ERFOUTPUT - for a .erfh5 file specify type 3 without compression (ICOMPRES=0)
ANALYSIS	Use **EXPLICIT** for a dynamic analysis



Defining the composite laminate

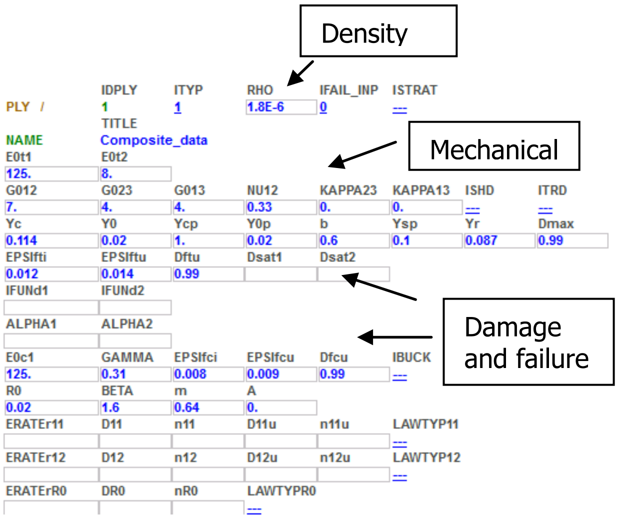
For a laminate the following 3 entities must be defined and linked:

1. **The Ply data** – Mechanical/damage data
2. **The Material data** – Layup and output
3. **The Part data** – Thickness and vector for reference fibre direction

1. Ply data

In the object explorer click on **Ply** (or use **Crash>Materials>Composites>Ply**) to open a new composite ply panel,

- Select a ply ID number (e.g. IDPLY = 1, or use the given default)
- Select ITYP=1 for Global Ply UD composite
- Specify the mechanical and damage parameters shown adjacent and give a suitable title (perhaps include the composite material type in this title)
- Finish with **Apply** and **Close**.



2. Materials data

Define a materials that can be used for each half of the DCB:

- Open a new material in the explorer panel, or via **Crash > Materials > Structural**
- Select type **131-Multilayered_Orth…** as the material type (= multi-layered orthotropic shell)
- Set the material parameters as below (see also next page),
  - Give a suitable title for the composite (e.g. type and layup)
  - Materials density = 1.8e-006 kg/mm^3
  - Set NOPER= 16 and ILAY=0, this opens 16 ply cards to be defined,
    1. Set all thicknesses = 0.2625mm
    2. Define the orientations = $[0/90/90/0/0/90/90/0]_s$ for the 16 plies
    3. Link all plies to the required ply cards (parameter IDPLY)
  - The other numbers are hourglass and other element default parameters (leave blank)
  - Below the layup data is PLYNUM and AUXVAR data which allows specify shell element information (strains, damage, etc.,…) to be output for post processing. As required specify,
    - PLYNUMx = ply number for the required output,
    - AUXVARx = corresponding auxiliary variable (see next page for definitions)

Finish with **Apply** and **Close.**

3. Part data

For each half of the DCB specify:

- An appropriate Part ID number.
- Link the part to the composite material (parameter IDMAT).
- Set the laminate thickness H=4.2 mm.
- Specify a reference vector for the fibres which is used together with angles on the material cards layup. Use IORT=0 for global frame and a vector 1,0,0.

Finish with **Apply** and **Close.**

PAM-CRASH materials cards:

MATER /									
	IDMAT	MATYP	RHO	ISINT	ISHG	ISTRAT	IFROZ		
MATER /	1	131	1.8E-6	0	4	0	0		
BLANK	AUXID1	AUXID2	AUXID3	AUXID4	AUXID5	AUXID6	QVM	THDID	IDMPD
	0	0	0	0	0	0	1.	0	0
	TITLE								
NAME	composite ply								
KSI	Fo	NOPER	ILAY	HGM	HGW	HGQ	As		
0.1	0.	16	0	0.01	0.01	0.01	0.833333		
IDPLY	THKPL	ANGPL							
1	0.2625	0.							
1	0.2625	90.							
1	0.2625	90.							
1	0.2625	0.							
1	0.2625	0.							
1	0.2625	90.							
1	0.2625	90.							
1	0.2625	0.							
1	0.2625	0.							
1	0.2625	90.							
1	0.2625	90.							
1	0.2625	0.							
1	0.2625	0.							
1	0.2625	90.							
1	0.2625	90.							
1	0.2625	0.							
BLANK	NMIN	BLANK	GRUC_KW	GRUC_VAL	IFAIL	BLANK	ERATIO	BLANK	
	0				0				

Stacking data for the laminate layup

IDPLY1	IDAUX1	IDPLY2	IDAUX2	IDPLY3	IDAUX3	IDPLY4	IDAUX4	IDPLY5	IDAUX5	IDPLY6	IDAUX6	IDPLY7	IDAUX7	IDPLY8	IDAUX8
1	1	1	2	1	3	1	4	1	5	1	6	1	7	1	8
IDPLY9	IDAUX9	IDPLY10	IDAUX10	IDPLY11	IDAUX11	IDPLY12	IDAUX12	IDPLY13	IDAUX13	IDPLY14	IDAUX14	IDPLY15	IDAUX15	IDPLY16	IDAUX16
1	9	1	10	1	11	1	12	1	13	1	14	1	15	0	0
IDPLY17	IDAUX17	IDPLY18	IDAUX18	IDPLY19	IDAUX19	IDPLY20	IDAUX20	IDPLY21	IDAUX21	IDPLY22	IDAUX22	IDPLY23	IDAUX23	IDPLY24	IDAUX24
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
IDPLY25	IDAUX25	IDPLY26	IDAUX26	IDPLY27	IDAUX27	IDPLY28	IDAUX28	IDPLY29	IDAUX29	IDPLY30	IDAUX30	IDPLY31	IDAUX31	IDPLY32	IDAUX32
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
IDPLY33	IDAUX33	IDPLY34	IDAUX34	IDPLY35	IDAUX35	IDPLY36	IDAUX36	IDPLY37	IDAUX37	IDPLY38	IDAUX38	IDPLY39	IDAUX39	IDPLY40	IDAUX40
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
IDPLY41	IDAUX41	IDPLY42	IDAUX42	IDPLY43	IDAUX43	IDPLY44	IDAUX44	IDPLY45	IDAUX45	IDPLY46	IDAUX46	IDPLY47	IDAUX47	IDPLY48	IDAUX48
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0

PAM-CRASH documentation on output variables:

Output information

Notes for Material Types 130, 131, 132:

- Material types 130, 131 and 132 correspond to multi-layered shell materials. The plies can be assigned the following material types in the Ply Data Section (parameter ITYP)
  - ITYP=0: unidirectional composite bi-phase ply model, (material type 130, 131)
  - ITYP=1: unidirectional composite global ply model, (material type 131)
  - ITYP=2: isotropic elastic-plastic damaging ply model, (material type 131)
  - ITYP=6: fabric composite bi-phase ply model.(material type 132, 131)
  - ITYP=7: fabric composite global ply model, (material type 131)
- **Auxiliary variables saved for plots.** For material types 130, 131 and 132 the following auxiliary variables can be saved on the plot files, by specifying on Cards 8 to 13 the reference number given in the following table.

Table (a): Auxiliary variables per ply for material types 130, 131 and 132

Reference number	ply type 0 (130,131) ply type 6 (131)	ply type 1 (131)	ply type 2 (131)	ply type 7 (131)	ply type 6 (132)
1	ε_{11}	ε_{11}	ε_{11}	ε_{11}	ε_{11}
2	ε_{22}	ε_{22}	ε_{22}	ε_{22}	ε_{22}
3	ε_{12}	ε_{12}	ε_{12}	ε_{12}	ε_{12}
4	ε_{23}	ε_{23}	ε_{23}	ε_{23}	ε_{23}
5	ε_{13}	ε_{13}	ε_{13}	ε_{13}	ε_{13}
6	σ_{11}	σ_{11}	σ_{11}	σ_{11}	σ_{11}
7	σ_{22}	σ_{22}	σ_{22}	σ_{22}	σ_{22}
8	σ_{12}	σ_{12}	σ_{12}	σ_{12}	σ_{12}
9	σ_{23}	σ_{23}	σ_{23}	σ_{23}	σ_{23}
10	σ_{13}	σ_{13}	σ_{13}	σ_{13}	σ_{13}
11	d^t (total)	d' = shear damage	d = damage	d_{12} (shear)	d^t (total)
12	d^t_S (shear)	d = transverse damage	$\varepsilon^{plastic}$	d_{11} (fiber 1)	d^t_S (shear)
13	d^t_V (volume)	$\varepsilon^{plastic}_{22}$	$\dot{\varepsilon}$	d_{22} (fiber 2)	d^t_V (volume)
14	σ^f	$2\varepsilon^{plastic}_{12}$		$2\varepsilon^{plastic}_{12}$	σ^{f1} (fiber 1)
15	d^f	$\dot{\varepsilon}$		$\dot{\varepsilon}$	d^{f1} (fiber 1)
16					σ^{f2} (fiber 2)
17					d^{f2} (fiber 2)
18					θ
19					ε^f_{11}
20					ε^f_{22}

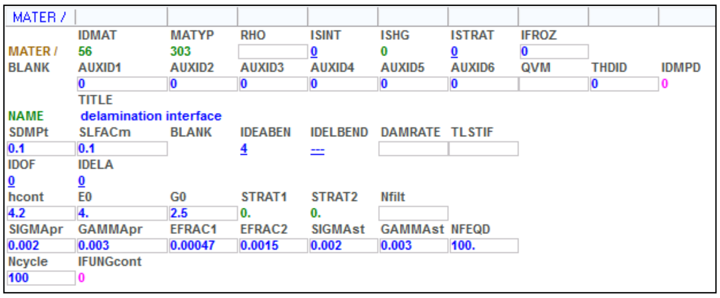
SHELL MATERIALS

The Tied delamination interface

Entities to be linked:

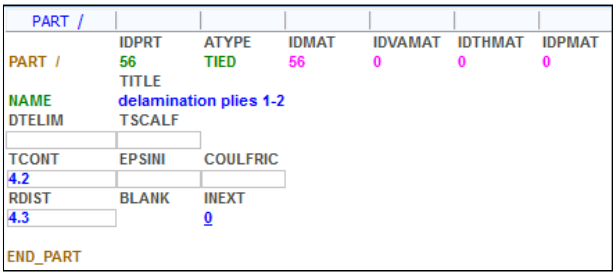
1. **Material data** for delamination
2. **Part data** for the thickness/contact
3. **Tied data** defining which parts are connected

Material cards



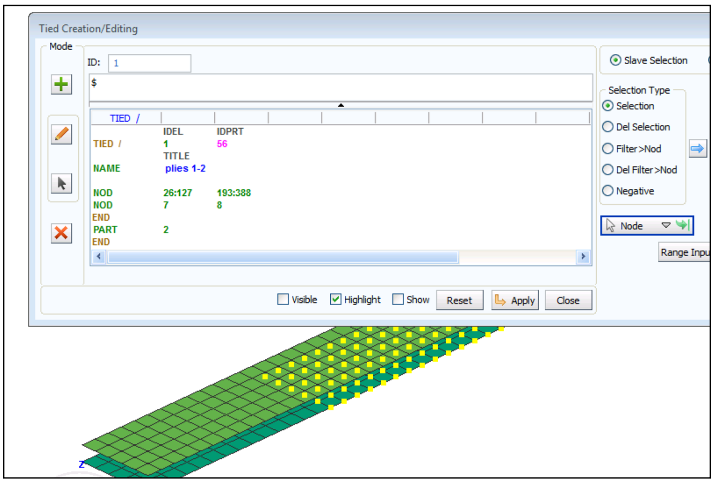
- Open the **Crash/Material Editor**
- Select the **303 Slink … interface** model and specify a unique ID number, or use the selected default
- Specify the data shown adjacent.
- Fracture energies: Set Mode I and Mode II to 470J/m^2 and 1500J/m^2.
- Failure stresses: Typical starting values for ‘uncracked` composites are 20N/mm^2 and 30N/mm^2 for Modes I and II respectively. The propagation stresses are usually assumed to be 10-30% of these values (NB an exact value is unimportant since it is the energies that control delamination). Since the DCB tests is ‘pre-cracked’ we can assume,
  Mode I - Propagation stress = Starting stress ⇒ 0.1 * 20N/mm^2 = <u>0.002 kN/mm^2</u>
  Mode II - Propagation stress = Starting stress ⇒ 0.1 * 30N/mm^2 = <u>0.003 kN/mm^2</u>
- Use **IDEABEN** =4 and the damping/filtering values shown. **IDOF**=0 ties all degrees of freedom
- Finish with **Apply** and **Close**

Part cards



- Open the **Crash > Parts > Part creation** to generate a parts entity for this interface
- Specify in the ATYPE box **TIED**
- Use a thickness for the contact 4.2mm and a search thickness slightly larger (e.g. 4.3mm)
- Assign a title and link this to the corresponding material
- Finish with **Apply** and **Close**

Tied cards



- Open the **Crash > Links > Tied** to generate the interface
- Specify a unique ID number (or use the default), a suitable title and link this to the Parts cards via parameter IDPRT
- Slave nodes - select the nodes (all except ca. 50mm at the free end by dragging a box over the nodes
- Master elements – select the top composite ply (part)
- Finish with **Apply** and **Close**

Finishing the model

Velocity loading for the DCB loading tabs

For all nodes along the upper loading edge a positive (upward +z) velocity is applied via a velocity time history. For those on the lower edge the same velocity time history is applied in the downward (-z) direction.

For the upper set of nodes:

1. Use **Crash > Loads > 3D BC** and then select type **VELBC** for velocity loading.
2. Define a curve function for **IFUN3** (= dir. z) having a velocity that increases from 0mm/msec to 2mm/msec at 1msec. Then hold the velocity constant to 20msec.
3. Assign this curve and select the nodes.
4. Use a SCALE3 = 1 for +z loading.
5. Finish with **Apply** and **Close.**

For the lower set of nodes repeat the same operations. Use the same velocity function, but this time set SCALE3 = -1 to reverse the velocity direction.

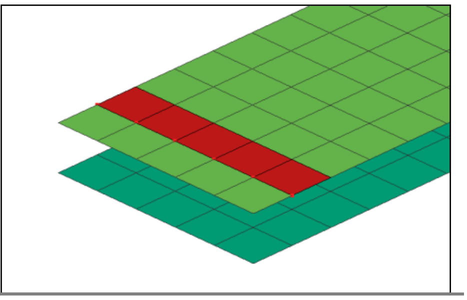
Boundary conditions for the free end

Use **Crash > Loads > Displacement BC** to fix 2 or 3 rows of nodes at the free (unloaded) end with displacement boundary conditions 111111.

Output information

1. Nodal displacement time histories: Use the option **Crash > OutPut > Nodal TH** to identify and store one node from each of the loading points.
2. Section force time histories: Use the option **Crash > OutPut > Section Force** with Type **SECTION** to define and define section cuts close to the upper and lower loading points. For each section a line of nodes are selected as SLAVES, and a line of attached elements are selected as MASTERS.

   This information allows the resistance force to crack opening to be monitored and possibly compared to test measurements.



Save (update) the dataset (DCB_Delamination_Model.pc) using the Export option.

Running the model and investigating results

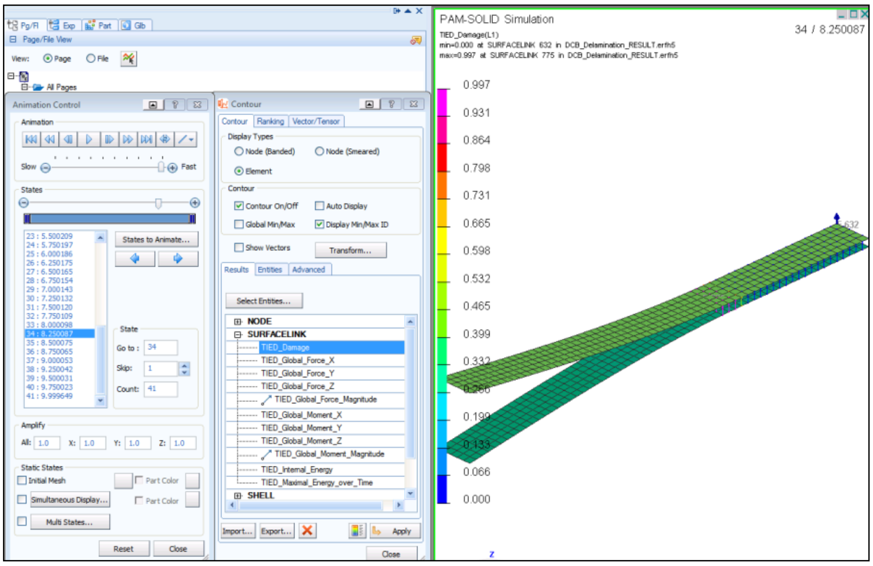
The PAM-CRASH dataset (DCB_Delamination.pc) is run; then open the results file,

DCB_Delamination_Model_RESULT.erfh5

in a new **Visual Viewer** session.

Deformed state results

1. Click **Results > Animation Control** to visualise the model and use the control panel to examine deformations (either at a certain time, or as a continuous animation).
2. Click **Results > Contour** and under Entity types activate SURFACELINK and type TIED_Damage to visualise damage contours at the delamination interface. Note that other visualisation options are available. The evolution of contour damage can be seen at specific states (via the **Results > Animation Control**), or animated.

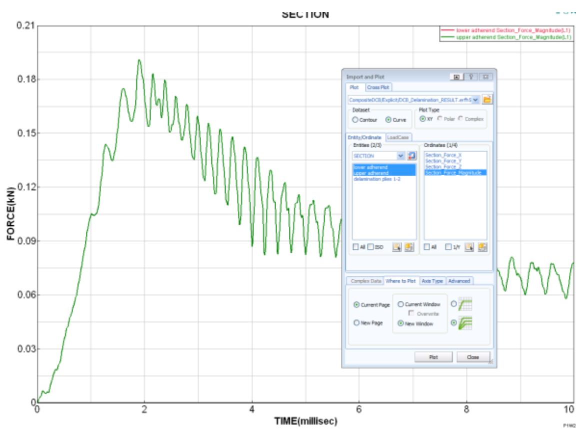


Time history results

Start time history contour plotting using **File > Import and Plot**

In the panel that appears activate,

- SECTION in the Entities
- The required sections to be plotted
- Section_Force_Magnitude
- Click PLOT

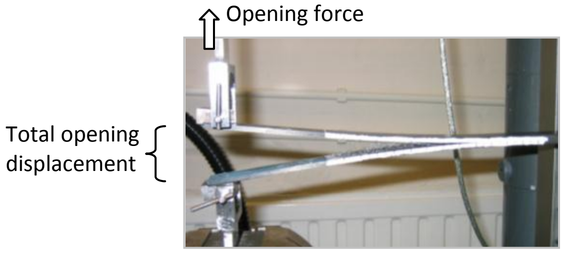


The green (oscillating) section force time history curve should appear giving a maximum force of about 0.18kN. This information can be passed through a filter if required.

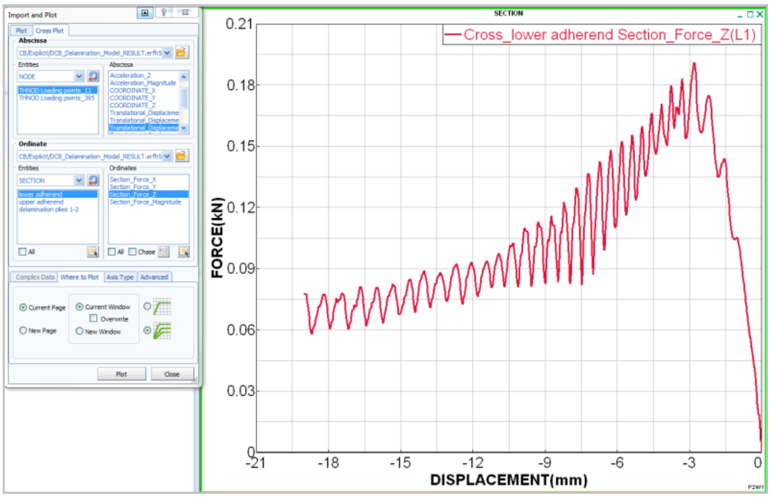
The behaviour can also be smoothed by modifying the materials input parameters using lower propagation stress values or more filtering. Alternatively, use a finer mesh or slower velocity loading.

Tutorial 11: Delamination modeling of a Double Cantilever Beam specimen

To create a plot of opening force (at the section) versus total opening displacement proceed as follows:

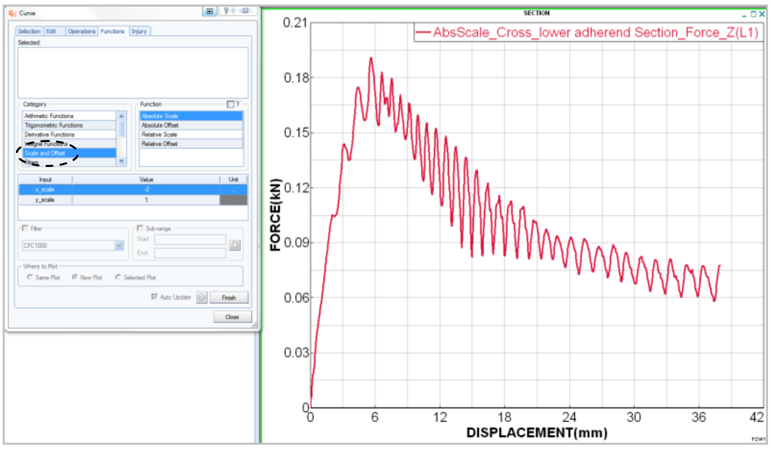


1. Do a cross plot of nodal displacement versus opening force (both in direction z for the lower DCB arm). The reference directions may be unfortunate giving some information as negative. If necessary this can be modified as shown below.

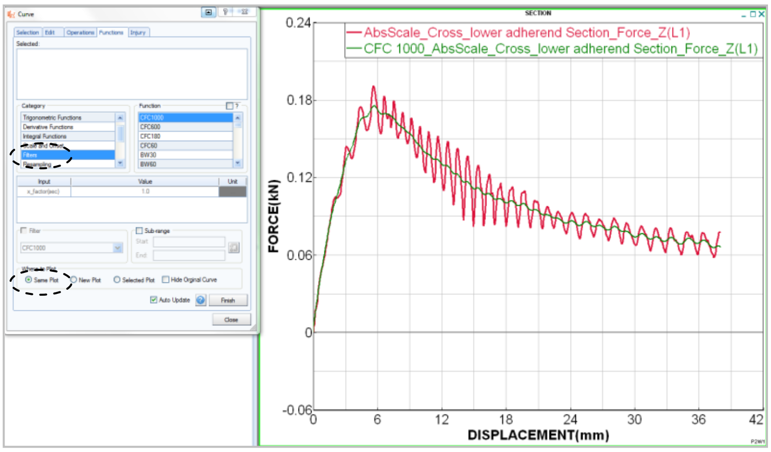


2. With the left mouse key click on the curve and it will appear black. Then click the right mouse key to open an options panel and activate Tools > **Functions…**

   The **Scale and Offset** option is selected to scale displacements by (-2) to obtain total positive opening displacement <u>for both</u> upper and lower DCB arms.



3. The **Filters** option with, for example, a CFC1000 filter allows high frequency information to be filtered as shown by the new superimposed green curve. Activate the tab 'same plot' to superimpose the two curves.



Checking the results

The opened (+ damaged) crack length and specimen width can be estimated to compute the total cracked area. This area multiplied by the Mode I fracture toughness (G_{IC} = 0.00047 J/mm^2) gives the total delamination energy absorbed. This energy should approximately correspond to the PAM-CRASH computed value (= 1.54 J). This value is found from either the output listing (see the energies per material near the end of the .out file), or from a time history plot of internal energy for the delamination interface (Part 56). Both are reproduced below.

From the output listing: Internal energy for delamination interface (Part 56)

PART ID	INTERNAL ENERGY 1	INTERNAL ENERGY 2	INTERNAL ENER TOT	TRANSL. KIN. ENER	X-MOMENT	Y-MOMENT	Z-MOMENT
1	6.19E-01	1.19E-04	6.19E-01	2.44E-02	-2.13E-03	1.98E-04	-3.35E-02
2	6.19E-01	1.19E-04	6.19E-01	2.44E-02	-2.13E-03	1.98E-04	3.35E-02
56	1.54E+00	0.00E+00	1.54E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00

From time history plot: Internal energy plot of delamination interface (Part 56)