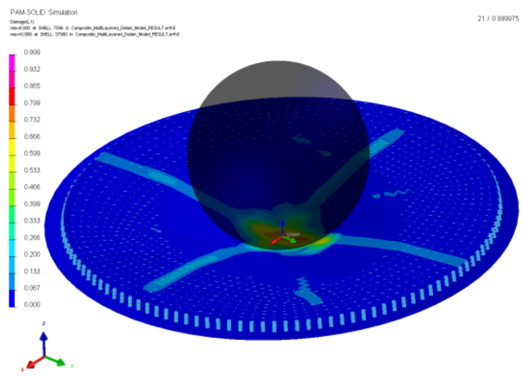
Tutorial 12

Composite Impact using Multi-layered Shell Elements and Delamination



Problem description

Outline	The transverse impact analysis of a simply supported composite disc is performed using layers of composite shell elements tied with delamination interface elements.
Analysis type(s):	Explicit
Element type(s):	Multi-layered composite shell, Delamination interface
Materials law(s):	Composite Global ply damage and failure law and delamination
Model options:	Boundary conditions, Contact, Initial velocities, Rigid body
Key results:	Stress distributions and damage with failure prediction, impact force time history

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, deformations, 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.

Also, it is important to have worked through Tutorial 10 for in understanding of this problem and ply modeling, and Tutorial 11 for an understanding of delamination modeling.

Problem description

Unit System: mm, kg, ms

Dimensions: Disc Φ 120mm, thickness 4.2mm

Rigid punch Φ 50mm

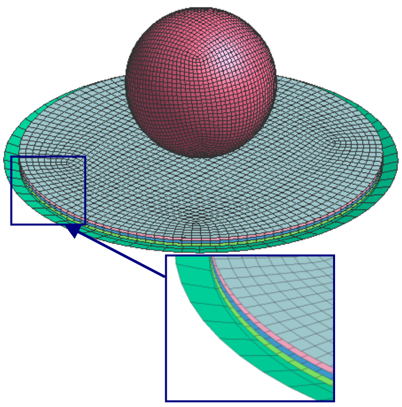
Support Φ 110mm inner, 130mm outer

Loading: Imposed velocity (10mm/msec max.)

Composite: Biaxial NCF and Epoxy resin,

Layup $[90/0/45/-45]_{2s}$

Modeling: Using 4 multi-layered shell element layers each having a 90/0/45/-45 layup



Delamination modeling and model preparation

The problem is setup and loaded in exactly the same way as the composite disc problem in Tutorial 10, which used multi-layered shell elements, except that inter-ply delamination is now included. This is represented by replacing the single layer of shell elements with 4 layers of multi-layered shell elements which are tied using the delamination model (Tutorial 11). This level of modeling improves accuracy since delamination (and changes in bending stiffness) can now be represented.

The mesh could easily be generated from the mesh in Tutorial 10 by repetitively 'copying' the layer of shell elements to obtain a stack of shell element layers. However, in order to simplify this tutorial the completed mesh is provided and only assignment of materials and entities will be briefly described. The prepared mesh is: **Composite_Multilayered_Delam_Mesh.pc**

Copy the mesh file to a model file which will be used to build the analysis model using **VCP**,

copy **Composite_Multilayered_Delam_Mesh.pc**

to **Composite_Multilayered_Delam_Model.pc**

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

Using VCP to make the analysis model

Start Visual-Crash PAM (VCP) and read in the new model file:

Select **File > Open** and open

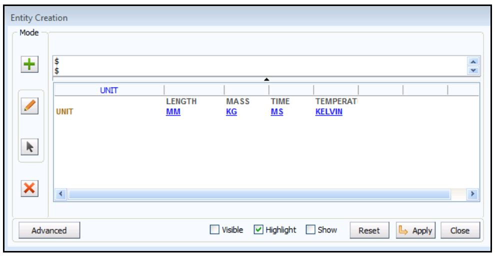
Composite_Multi-layered_Delam_Model.pc

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.**

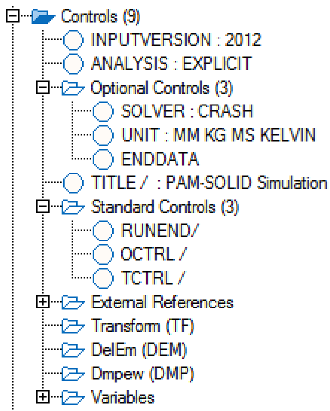
Finish with **Apply** and **Close.**



Defining some basic model data

For the PAM- controls the following parameters are set:

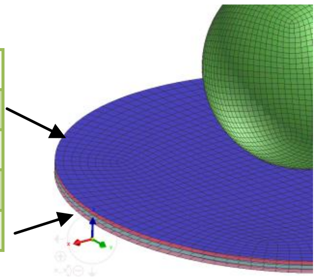
INPUT-VERSION	The PAM-CRASH version being used (e.g. **2012**)
RUNEND	Termination time for the analysis **(=1.0 msec)**
TITLE	The title used that will appear on all output plots
SOLVER	Use **CRASH** for a PAM-CRASH analysis
TCTRL	The nodal timestep (NODTSP) option is activated for the delamination interface 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 = 0.05**) for (RUNEND=1.0)/0.05 = 20 pictures • Parameter ERFOUTPUT - for a .erfh5 results file specify type 3 without compression (ICOMPRES=0)
ANALYSIS	Use **EXPLICIT** for a dynamic analysis



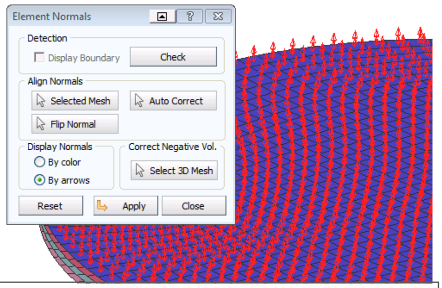
Composite Materials, Parts and Plies definitions

The shell layup for tutorial 10 was 16 plies with fibre orientation [90/0/-45/ 45/45/-45/0/90]s For this tutorial the same layup using four sub-laminates layers of shell elements are used. The equivalent layups are:

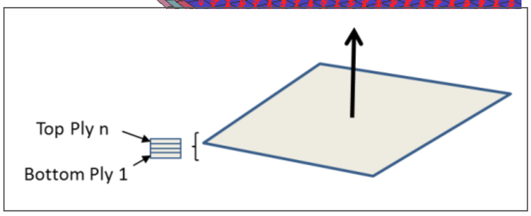
Position	Layer	Material group	Layup
Top (+z)	Shell layer 4	103	45/-45/0/90
	Shell layer 3	102	90/0/-45/45
	Shell layer 2	101	45/-45/0/90
Bottom (-z)	Shell layer 1	100	90/0/-45/45



An important consideration is the sequence of ply layers in a multi-layered shell element. By definition the layers 1, 2, 3… run from the bottom surface of the element to the top surface. The 'bottom' and the 'top' are with respect to the direction of the element normal direction, which can be checked and modified if necessary with the **Checks > Element Normals** option.



For the Disc this has been used and all normal are consistent and point in the positive z direction. The first ply layer is therefore the underside of each element with respect to the positive z-direction.



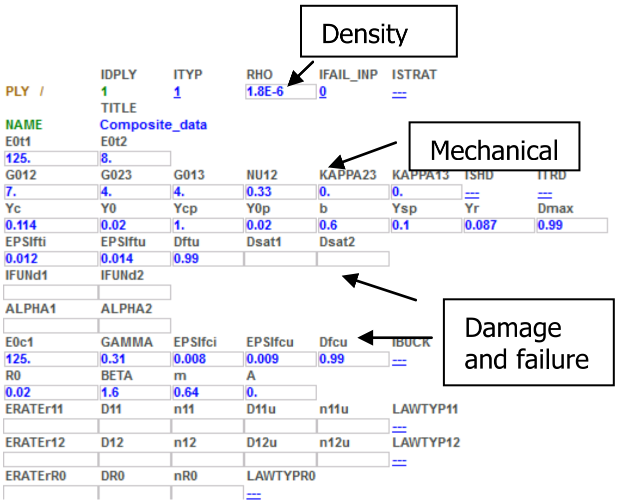
For each sub-laminate (layer of shells) the following 3 entities must be defined and linked:

1. **Ply data** – Mechanical/damage data (this is the same for all plies in all sub-laminates)
2. **Material data** – Layup/output (for each sub-laminate)
3. **Part data** – Thickness and vector for reference fibre direction (for each sub-laminate)

1. Ply data

In the object explorer click on **Ply** (or use **Crash>Materials>Composites>Ply**) to open a new 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. Composite material data

- Open a new material in the explorer panel, or via **Crash > Materials > Structural**
- Select type **131-Multi-layered_Orth…** as the material type (= multi-layered orthotropic shell)
- Give each layer a unique material ID
- 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
  - Number of plies (set NOPER= 4 with ILAY=0). This opens 4 ply cards to be defined,
    1. Set all thicknesses = 0.2625mm
    2. Orientations are defined according to the above table
    3. Link all plies to the required ply cards (parameter IDPLY)
  - Defaults parameters are defined (or left blank) and any required outputs specified

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

3. Composite part data

Define a new Part for the laminate and specify,

- An appropriate ID number.
- Link the part layer to the corresponding composite material layer (parameter IDMAT).
- Set the laminate thickness H=1.05.
- 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 (e.g. for composites layer 1 = Material group 100):

MATER /									
	IDMAT	MATYP	RHO	ISINT	ISHG	ISTRAT	IFROZ		
MATER /	100	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	Duplicate_composite ply								
KSI	Fo	NOPER	ILAY	HGM	HGW	HGQ	As		
0.1	0.	4	0	0.01	0.01	0.01	0.833333		
IDPLY	THKPL	ANGPL							
1	0.2625	90.							
1	0.2625	0.							
1	0.2625	-45.							
1	0.2625	45.							
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

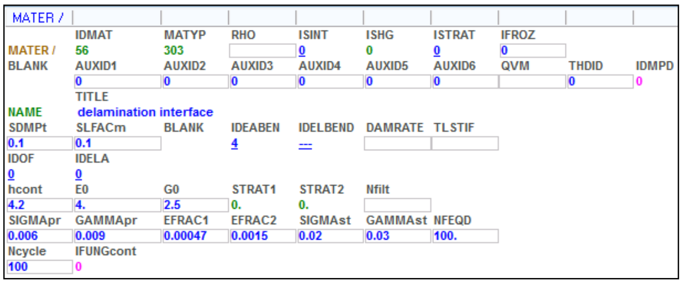
Ply and output information

The Tied delamination interfaces

For each pair of attached plies a delamination interface has to be defined similar to Tutorial 11. In each case the entities to be linked are:

1. **Material data** for delamination (only one material definition is needed for all interfaces)
2. **Part data** for the thickness/contact information on the interface (one definition per interface)
3. **Tied data** defining which parts (plies) are connected (one definition per interface)

Material cards



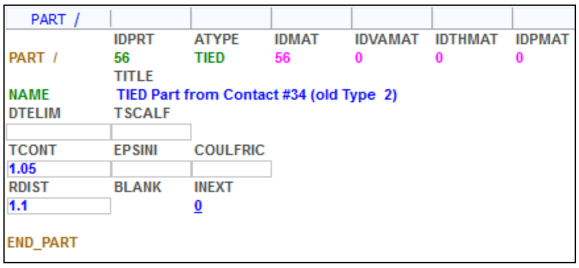
- Open the **Crash/Material Editor**
- Select the **303 Slink ... interface** model and specify a unique ID number, or use the selected default
- Specify the data shown adjacent.
- Fracture energies: Set Mode I and Mode II to $470 J/m^2$ and $1500 J/m^2$.
- Failure stresses: Typical starting values for 'uncracked' composites are $20 N/mm^2$ and $30 N/mm^2$ 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),

Mode I - Propagation stress = 0.3 * $20 N/mm^2$ = <u>$0.006 kN/mm^2$</u>, Starting stress = <u>$0.02 kN/mm^2$</u>

Mode II - Propagation stress = 0.3 * $30 N/mm^2$ = <u>$0.009 kN/mm^2$</u>, Starting stress = <u>$0.03 kN/mm^2$</u>

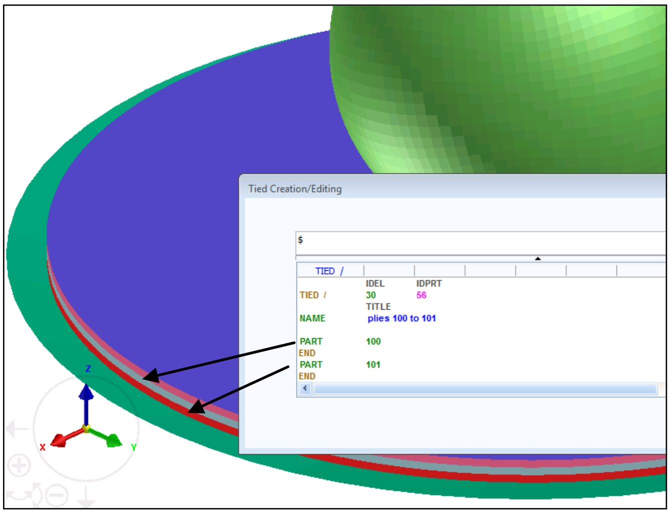
- Use **IDEABEN** =4 and the damping/filtering values shown. **IDOF**=0 ties all degrees of freedom
- Finish with **Apply** and **Close**

Part cards



- Open the **Crash > Parts > Part creation** to generate a parts entity for each interface
- Specify in the ATYPE box **TIED**
- Use a thickness for the contact 1.05mm and a slightly larger search thickness (=1.1mm)
- 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 Part cards via parameter IDPRT
- Slave nodes - select the part for one interface ply
- Master elements – select the part for the attached interface ply
- The order of Slave and Master is not important

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

Parts and material data for the punch and support

For the punch and support use a shell elastic material model (Type 101 – Elastic Shell). In each case specify,

- Thickness = 1mm
- Elastic modulus = 70GPa and Poisson's ratio = 0.3
- Density = 2.8E-06 kg/mm^3
- Link the part to the material

Finishing the model

1. Entities for the Punch

For convenience the stiff punch is defined as an 'approximate' rigid body. A simple and CPU fast method is to fix all nodes in the x-y plane and specify a constant velocity in the vertical direction. Note this loading is not the same as a punch with initial velocity that slows down during impact as kinetic energy is converted into plate deformation energy.

1. Use **Crash > Loads > Displacement BC** and fix all nodes in the punch to have displacement boundary conditions 110111.
2. Use **Crash > Loads > 3D BC** and then select type VELBC for velocity loading. Define a curve function for IFUN3 (= dir. z) having a constant velocity -10mm/msec over a duration longer than the analysis (e.g. 0→100 msec).

2. Entities for the Support

Fix the support with displacement boundary conditions (all nodes = 111111).

3. Contacts: For the Punch-to-Disc and Disc-to-Support

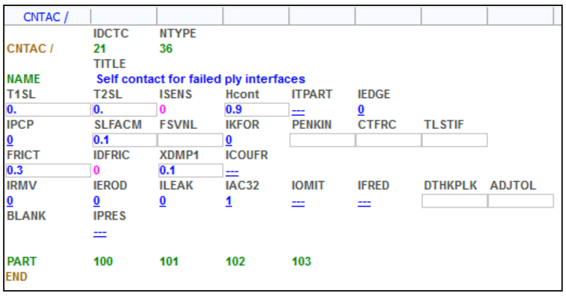
For the Disc-to-Support:

1. Open **Crash > Contacts** and select contact type 34 (one sided contact).
2. Define one side (e.g. the Disc part) as SLAVE and the other contact part (e.g. the Support part) as MASTER.
3. Set the contact distance hcont=0.95mm. The actual separation of parts is approximately 1mm and this smaller contact distance will ensure there are no initial penetrations at the start.
4. Set the contact friction FRICT = 0.2 and contact damping XDMP1 = 0.1.
5. All other default parameters can be used (leave blank).

Repeat the same operations for the Punch-to-Disc contact. For the punch either the complete punch can be selected, or to save some CPU time only obvious nodes on the lower surface of the Punch (that will make contact) can be selected.

4. Additional Contact for the Disc

Once the ties in the delamination start to fail the layers will penetrate each other. In order to prevent this a type 36 self contact is defined between all plies using the adjacent data. The thickness Hcont should be less than the actual ply separation distance (e.g. 0.9mm).



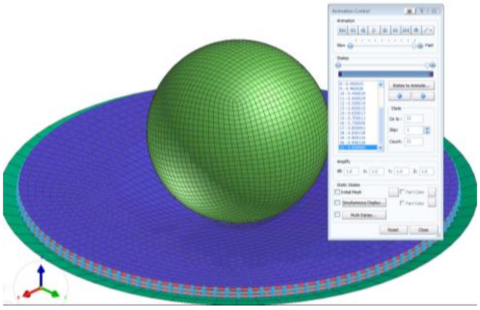
5. **Save (update) the dataset** (Composite_Multi-layered_Delam_Model.pc) using the Export option.

Running the model and investigating results

The PAM-CRASH dataset is run; then open the results file,

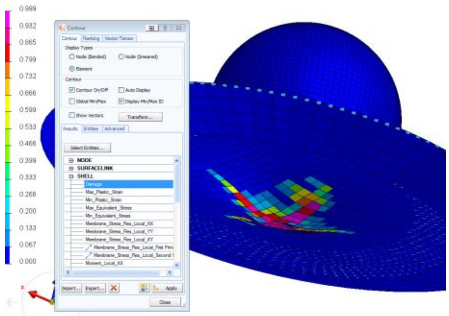
Composite_Multi-layered_Delam_Model _RESULT.erfh5

in a new **Visual Viewer** session.

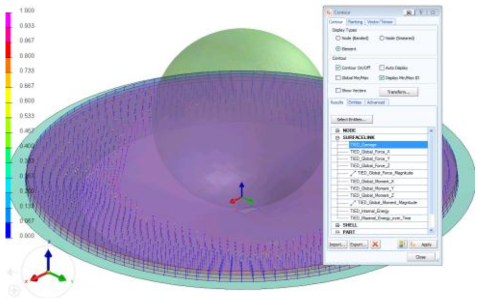


Deformed state results

1. Click **Results > Animation Control** to visualise the model and use the adjacent panel to examine deformations (either at a certain time, or as a continuous animation).



2. Click **Results > Contour** and under Entity types activate SHELL and type Damage to visualise total damage in the shells. 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.



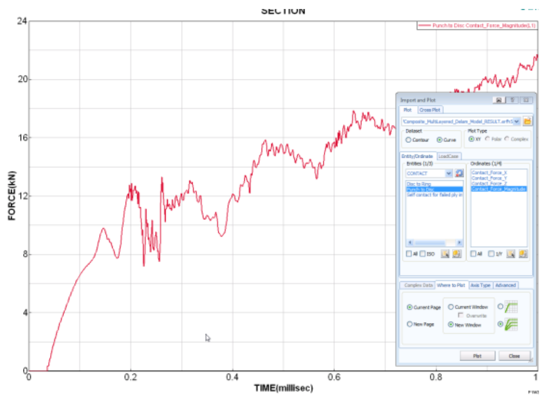
3. Click **Results > Contour** and under Entity types activate SURFACELINK and type TIED_Damage to visualise damage in the interfaces. The transparent viewing option is useful to help see the area of delamination (The ties are eliminated and not shown in areas that have fully delaminated).

Time history results

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

In the panel that appears activate,

- CONTACT in the Entities
- The required contact: Punch to Disc
- Contact Force Magnitude
- Click **PLOT**



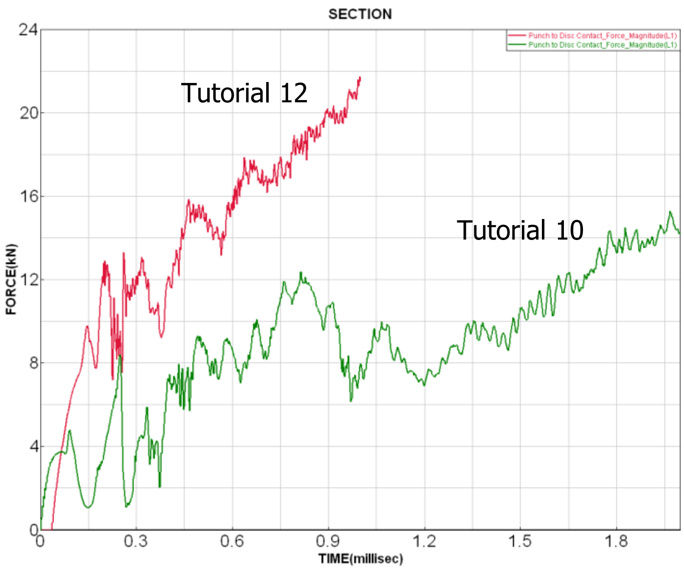
The red contact force time history curve is generated giving a maximum force of ≈21kN at 1msec. This curve could be filtered if required.

Comparing impact force time histories for Tutorials 10 and 12

The time history impact force from Tutorial 10 can be superimposed to the curve for Tutorial 12. For example, by using the **Chase Curves** option.

Delamination model: Delamination has occurred under the punch which has allowed load redistributions and thereby limited ply damage evolution.

Ply failure (only) model: Localised deformations under the punch have caused earlier and greater ply damage. The Impact force response is softer with lower ultimate failure loads reached.



The figure below compares ply damage for the two models which suggests this assumption is possible.