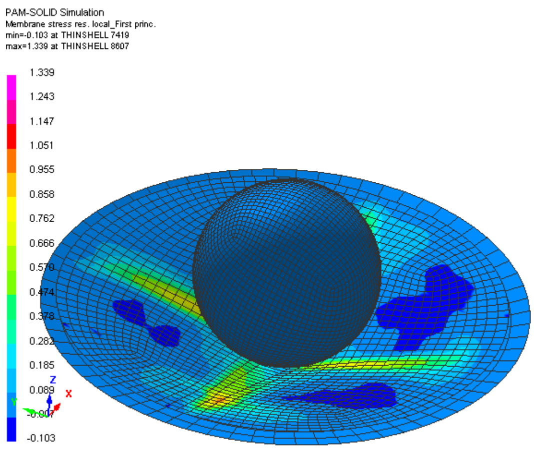
Tutorial 10

Impact of a Composite Disc using Multilayered Shell Elements



Problem description

Outline	The transverse impact analysis of a simply supported composite disc is performed using a single layer of multi-layered shell elements.
Analysis type(s):	Explicit
Element type(s):	Multilayered composite shell
Materials law(s):	Composite using the Global ply damage and failure law
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 November 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.

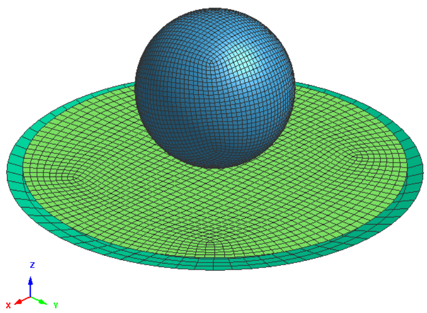
Problem Data

Unit System: mm, kg, ms

Dimensions: Disc Φ 120mm, thickness 4.2mm
Rigid punch Φ 50mm
Support Φ 110mm inner, Φ 130mm outer

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

Composite: Biaxial NCF with Epoxy resin,
Lay-up [90/0/45/-45/-45/45/0/90]s



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 :

Composite_SingleMultiLayerShell_Mesh.pc

to **Composite_SingleMultiLayerShell_Model.pc**

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

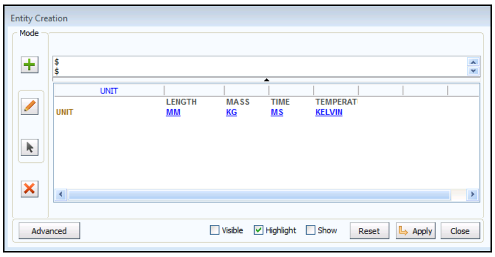
Using VCP to make the analysis model

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

Select **File > Open** and open the file

Composite_SingleMultiLayerShell_Model.pc

Specify the model units system

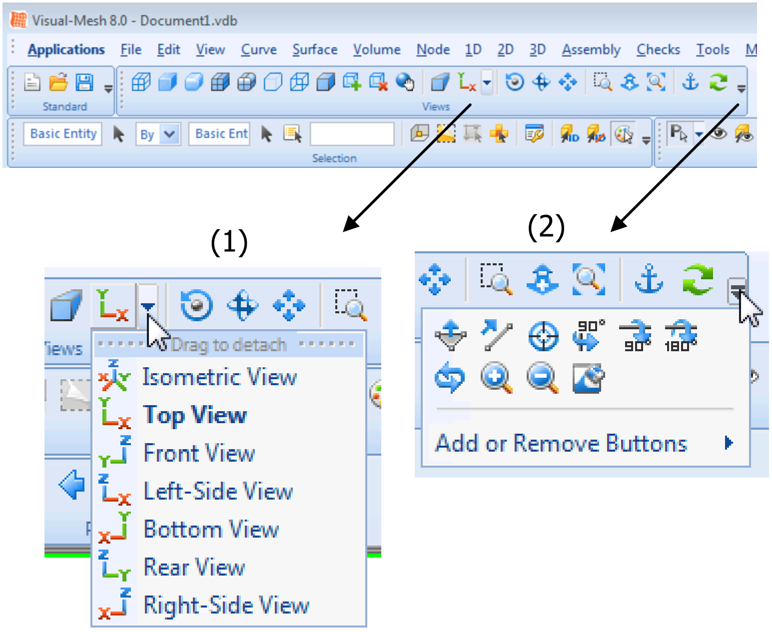


Set the model units system by selecting **Crash > Optional Controls > Units** to open the adjacent panel,

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

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

Positioning and viewing options:



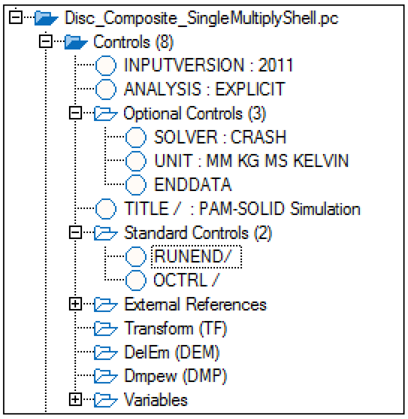
Important options are available in the main (top) panel to position, center, zoom (in and out) and gererally vary viewing of the model.

1. Click the axis tab and with the 'left' mouse key to open options to position the model in the x,y,z or perspective (isometric) frame.
2. Click the viewing tab with the 'left' mouse key to open options to zoom in/out and generally position the model.

Tutorial 10: Composite impact using multi-layered shell elements

Defining some basic model data

For the PAM- controls set the following parameters

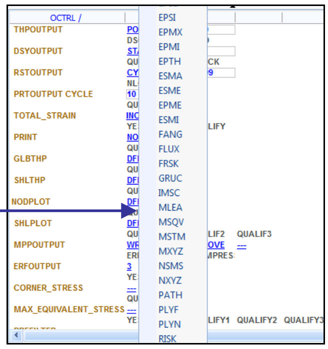


INPUT-VERSION	Set the PAM-CRASH version being used (e.g. **2012**)
RUNEND	Termination time for the analysis **(=2.0 msec)**
SOLVER	Use **CRASH** for a PAM-CRASH analysis
OCTRL	• THPOUTPUT – output interval for graphical (x,y) time history information (e.g. use **POINTS = 1000** for one thousand points) • DSYOUTPUT - output interval for deformed states (e.g. use **STATE = 50** for fifty pictures) • Parameter ERFOUTPUT - for a .erfh5 results file specify type 3 without compression (ICOMPRES=0)
ANALYSIS	Use **EXPLICIT** for a dynamic analysis

Further composites specific output:

In the control option **OCTRL** (already defined above for output intervals) are additional options for other specific outputs to the results files (SOLPLOT, NODPLOT, etc..; respectively for solid elements, nodes, etc.,…). In this case,

- For SHLPLOT (shell elements outputs), the following options for specific output are available.
- Check DFLT is activated (done by default) which gives basic forces, moments, etc., for shell elements. It also includes DMG for output of composites damage.
- Available outputs are defined in VCP and the manuals.



Defining Parts and Materials for the Support and Punch

For each Material (Punch and Support)

- Open the **Material Editor**
- Select Shell and Element type 101 as the material type (= elastic shell material)
- Set the material parameters as shown,
  - Material ID number and a suitable title
    1. Use e.g. ID 1 for support (title = Support)
    2. Use e.g. ID 2 for punch (title = Punch)
  - Material density = 7.8e-006 kg/mm^3
  - Modulus = 70.0 kN/mm^2 and Poisson's ratio = 0.3
  - The other numbers are hourglass and element default parameters (assign or leave blank)
- Click on **Apply** and **Close**

Then for each Part (Punch and Support)

Define a new Part for the Punch and then for the Support. In each case give,

- An appropriate ID number
- Link the part to the corresponding Punch or Support material (parameter IDMAT)
- Assign a thickness of 1mm

Defining the composite laminate

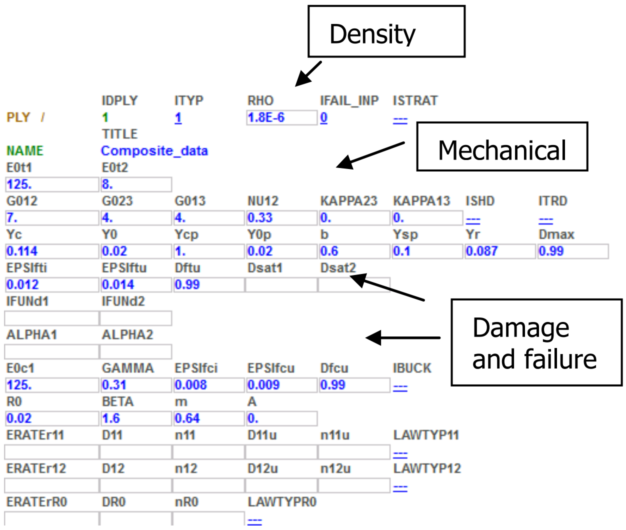
For a laminate the following 3 entities must be defined and linked:

1. **The Ply data** – Mechanical/damage data
2. **The Material data** – Lay-up/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 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. Defining the composite materials

- 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
  - Number of plies (set NOPER= 16 with ILAY=0). This opens 16 ply cards to be defined,
    1. Set all thicknesses = 0.2625mm
    2. Orientations = $[90/0/-45/45/45/-45/0/90]_s$ (NB the 's' means symmetry giving [90/0/-45/45/45/-45/0/90/90/0/-45………/0/90]
    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. The composite Part

Define a new Part for the laminate and specify,

- An appropriate ID number.
- Link the part to the composite material (parameter IDMAT).
- Set the laminate thickness H=4.0.
- 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:

	IDMAT	MATYP	RHO	ISINT	ISHG	ISTRAT	IFROZ		
MATER /	13	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	90.							
1	0.2625	0.							
1	0.2625	-45.							
1	0.2625	45.							
1	0.2625	45.							
1	0.2625	-45.							
1	0.2625	0.							
1	0.2625	90.							
1	0.2625	90.							
1	0.2625	0.							
1	0.2625	-45.							
1	0.2625	45.							
1	0.2625	45.							
1	0.2625	-45.							
1	0.2625	0.							
1	0.2625	90.							
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_S^t (shear)	d = transverse damage	$\varepsilon^{plastic}$	d_{11} (fiber 1)	d_S^t (shear)
13	d_V^t (volume)	$\varepsilon_{22}^{plastic}$	$\dot{\varepsilon}$	d_{22} (fiber 2)	d_V^t (volume)
14	σ^f	$2\varepsilon_{12}^{plastic}$		$2\varepsilon_{12}^{plastic}$	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_{11}^f
20					ε_{22}^f

SHELL MATERIALS

Tutorial 10: Composite impact using multi-layered shell elements

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 fixed 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. Save (update) the dataset (Composite_SingleMultiLayerShell_Model.pc) using the Export option.

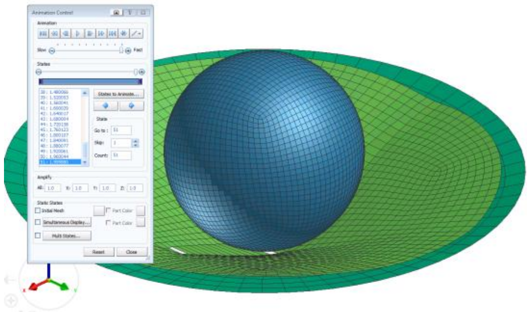
Running the model and investigating results

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

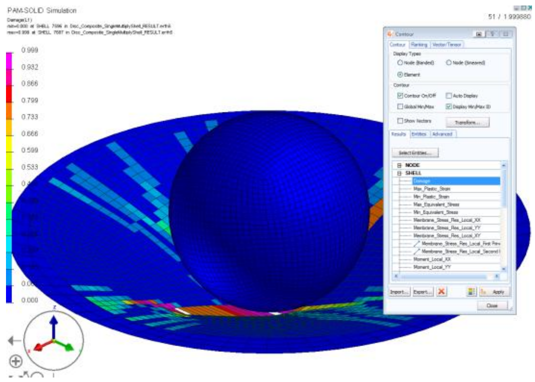
Composite_SingleMultiLayerShell_Model_RESULT.erfh5

in a new **Visual Viewer** session.

Deformed state results

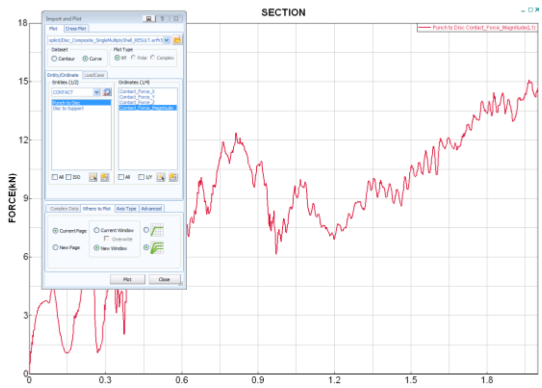


1. Click **Results > Animation Control** to visualise the model and use the adjacent panel to examine deformations (either at a certain time, or as a continuous animation).



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.

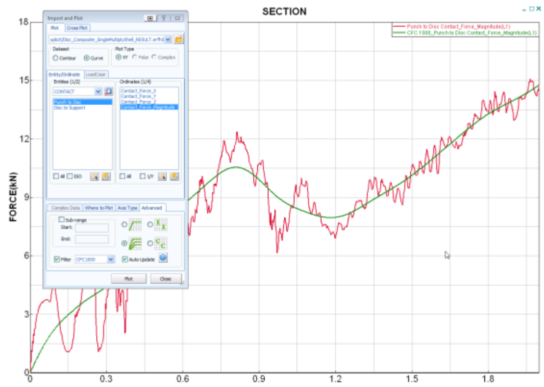
Time history results



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

3. In the panel that appears activate,
   - CONTACT in the Entities
   - The required contact: Punch to Disc
   - Contact Force Magnitude
   - Click **PLOT**

The red (oscillating) contact force time history curve should appear giving a maximum force of ≈15kN.



This information can be passed through a filter (activate the filter under the tab **Advanced**). E.g. use a filter type CFC1000 to get the smoothed curve shown (green). This takes out all high frequency oscillations above 1000Hz.