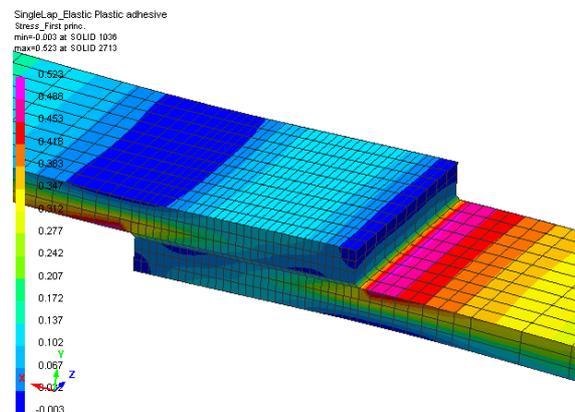


Tutorials 5 and 6

Adhesive Bonded Joint under Elastic and Elasto-Plastic with Failure Loading



Problem description

Outline	Analysis of a Single and Double Lap joint using elastic, elasto-plastic and material failure criteria.
Analysis type(s):	Implicit and Explicit
Element type(s):	3D solid
Materials law(s):	Elastic and elasto-plastic with failure
Model options:	Linear elastic and elasto-plastic analysis, materials failure, geometric nonlinear (implicit) analysis, boundary conditions, nodal loads
Key results:	Stress distributions and failure prediction
Prepared by:	Anthony Pickett, ESI GmbH/Institute for Aircraft Design, Stuttgart
Date:	December 2007
Version:	V4 (Updated May 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 Exercise 1 (Elastic loading of a cantilever beam) has been worked through. In order to avoid unnecessary repetition some explanations on the use of Visual for meshing and creation of the analysis model will be kept rather brief; whereas new options will be explained in more detail.

Problem data and description

Units: kN, mm, kg, ms

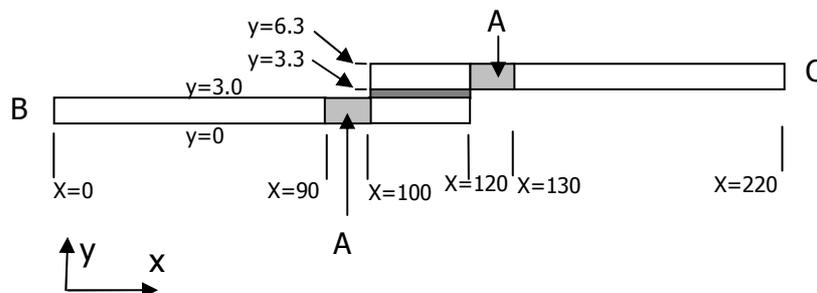
Description: The "single lap joint" consists of two Aluminium adherends (each 3mm thick) bonded with an adhesive layer (0.3mm thick). The joint overlap length is 20mm and the joint width is 20mm. Each adherend has a free length of 100mm with a 10mm mesh transition zones (denoted A in the figure below) adjacent to the bond line region. The exercise will analyse two joint configurations:

1. A single lap joint which is free in the lateral (y) directions; except at the end restraints.
2. A double lap joint which can be analysed by imposing symmetry constraints in the lateral (y) direction.

Loading: Imposed loading is via velocity control at one end (end C), with fixed constraint at the opposite end (end B).

Material: Two different material laws will be considered; namely an elastic and an elasto-plastic with failure law for each joint type:

1. Elastic analysis: The elastic analysis will use Aluminium adherend properties of $E=70 \text{ kN/mm}^2$ (GPa), $\nu=0.3$ and density $2.8 \cdot 10^{-6} \text{ kg/mm}^3$ and adhesive properties of $E=3 \text{ kN/mm}^2$ (GPa), $\nu=0.3$ and density $1.6 \cdot 10^{-6} \text{ kg/mm}^3$.
2. Plastic analysis with failure: An elasto-plastic law with yield stress of 20 MPa will be used with a maximum Von Mises failure strain criteria.



Tutorial 5,6: Single and Double lap adhesive joints

Supplied datasets

No datasets or meshes are needed to tackle the problem; the mesh will be generated as a part of the exercise.

It is recommended that you use the following names for the PAM-CRASH input and results files.

For the mesh model:

SingleLap_Mesh.pc

For the explicit analysis models:

Elastic problem: **SingleLap_Elastic_Explicit.pc** and **DoubleLap_Elastic_Explicit.pc**

Plastic problem: **SingleLap_Plastic_Explicit.pc** and **DoubleLap_Plastic_Explicit.pc**

and for the implicit analysis models:

Elastic problem: **SingleLap_Elastic_Implicit.pc** and **DoubleLap_Elastic_Implicit.pc**

Plastic problem: **SingleLap_Plastic_Implicit.pc** and **DoubleLap_Plastic_Implicit.pc**

In each case completed PAM-CRASH datasets are available in case you get into trouble.

Basic theory relevant to problem

For the double and single lap joints analytical solutions, assuming elastic material behaviour, are available in the literature and can be used to help validate the Finite Element analysis results.

For the elastic double lap joint see the theory of Volkerson¹ and for the elastic single lap joint see the theory of Goland and Reisner².

¹ Volkerson, O. "Die nietkraftverteilung in zubeanspruchten nietverbindung mit konstanten loshonquerschnitten", Luftfahrtforschung, Vol 15, P41, 1938.

² Goland, M. and Reissner, E. „Stresses in cemented joints“, J. Appl Mechanics, Vol 11, P17, 1944.

Contents

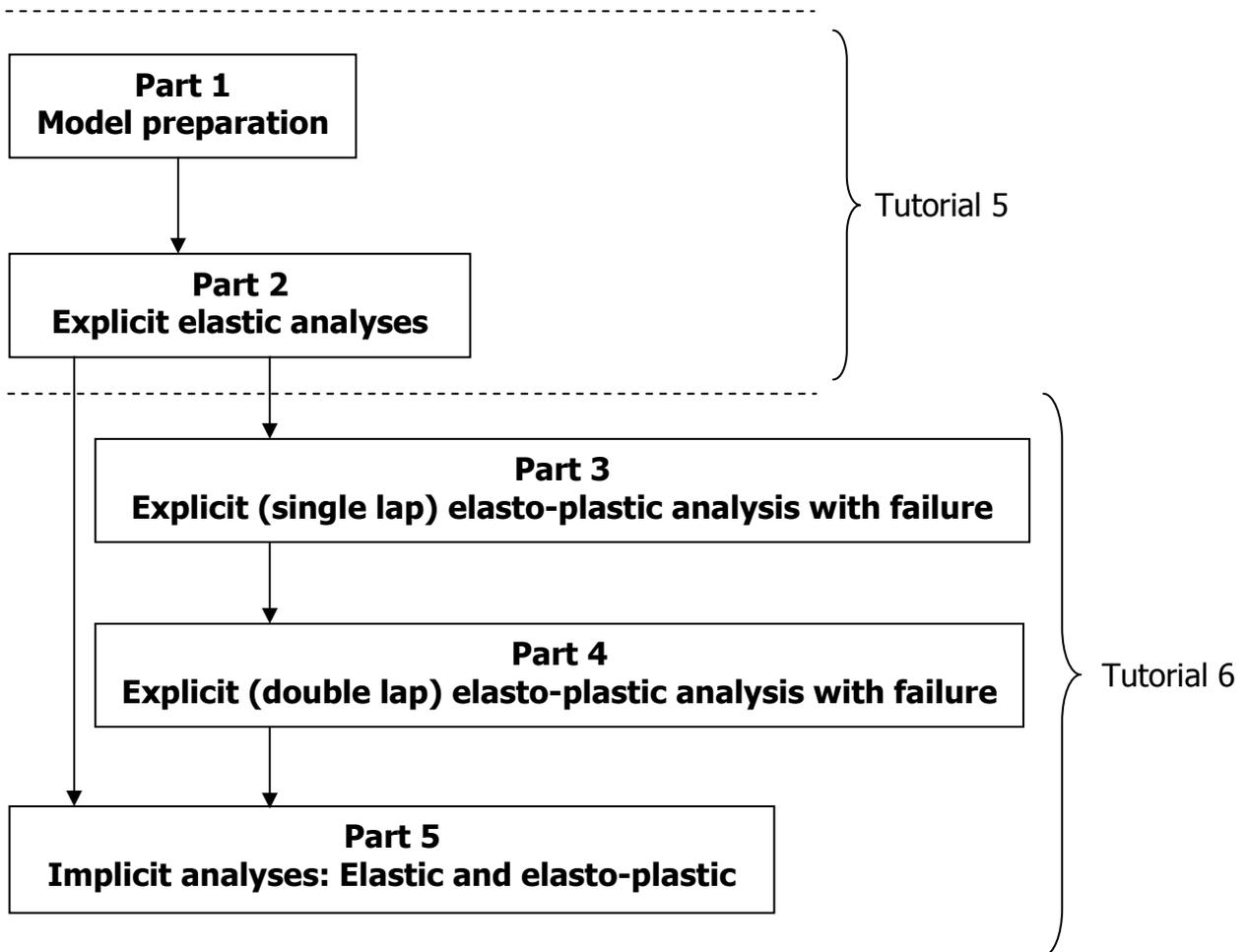
Tutorials 5 and 6 1
Adhesive Bonded Joint under Elastic and Elasto-Plastic with Failure Loading 1
Problem description..... 1

[Tutorial 5 \(Model preparation and first elastic analysis\)](#)

Part 1: Using VCP to construct the model..... 5
Part 2: Explicit analysis of the Single Lap joint (elastic)..... 18

[Tutorial 6 \(non-linear and failure analyses for single and double lap joints\)](#)

Part 3: Explicit analysis of the Single Lap joint (elasto-plastic with failure) 21
Part 4: Explicit analysis of the Double Lap joint (elasto-plastic with failure)..... 25
Part 5: Implicit analysis of the Single Lap joint (static and non-linear) 26



Part 1: Using VCP to construct the model

Preparing the mesh (Visual Mesh)

Meshing options for the adhesive bonded joints

A common problem of modelling adhesive bonded joints is accurate representation of the high stress gradients in the relatively thin glueline; especially, if accurate stress and strain predictions are needed for failure analysis. In theory many elements could be used in the adhesive layer region; however, this would quickly lead to very large meshes (number of elements) for the full joint which would make the analysis of a large structure computationally very expensive. Careful planning of the mesh and use of mesh grading methods are needed to obtain a suitably detailed mesh in the adhesive and adherends.

In this exercise 3D solid elements are used so that a good representation of the 3D stress distribution in the joint is possible. It should be noted that in-plane loaded 'wide' joints (normal to the plane of the joint) could be analysed as a plane strain (2D) problem; or, in the case of axial loaded tubular joints as an axi-symmetric problem. 2D analysis greatly simplifies the Finite Element model and CPU costs, but is limited to specific in-plane loading only.

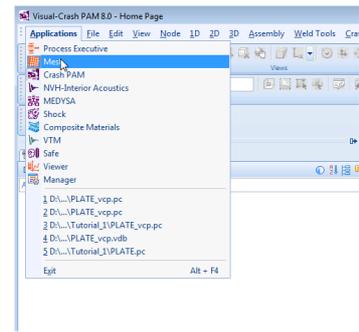
For this exercise both a single and double lap joint will be analysed. It will be shown that the same mesh can be used for each, with the main difference being the boundary conditions which convert the single lap joint to one symmetric half of a double lap joint. In order to generate the 3D solid model the following meshing steps are followed:

1. First generate key points (x,y,z coordinates) to define the geometry and mesh transition zones on the section of the bonded joint.
 2. Connect these points to obtain 2D surface patches which are then meshed with 2D elements.
 3. The 2D surface mesh is then translated in the joint width direction and used to generate a full 3D solid model of the joint.
 4. Finally, the redundant surface 2D elements are eliminated.
-

Tutorial 5,6: Single and Double lap adhesive joints

Start the Visual Crash Program (VCP) and under the Applications tab select the **Mesh** option.

This will activate the meshing module (**Visual Mesh**) of the VCP program which is dedicated to generating finite element meshes.



Select the option **New file** then specify (if required) the model unit system to be used (**mm, kg, millisec, kelvin**).



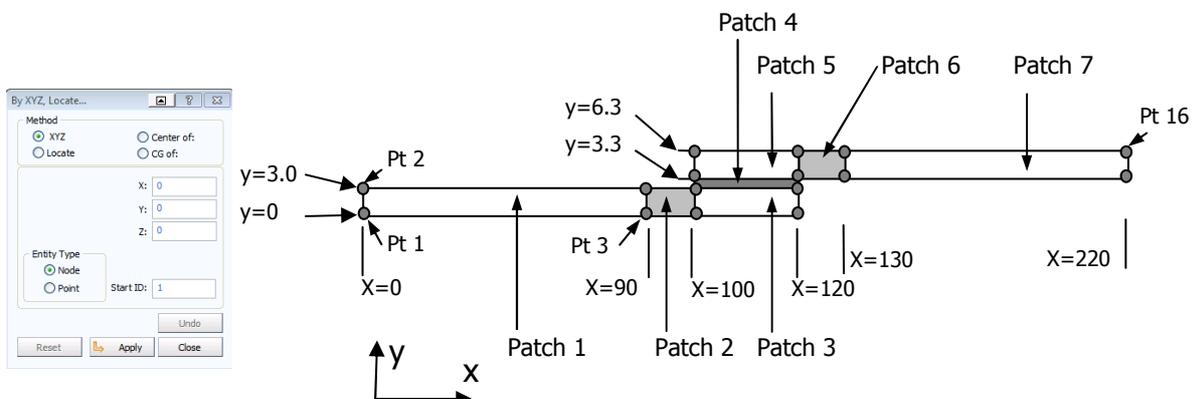
Preparing the joint 2D surface mesh

The joint could be consider as having seven rectangular surface patches (three per adherend and one for the glue line) which are described by 16 points (*), as shown in the figure below. Each of these surfaces will be meshed with 2D elements, but first the 16 points must be defined. It is arbitrarily chosen that this model lies in the x-y plane with the lower left corner at the origin (x=0, y=0); all other points having the coordinates shown (use z=0 for all points).

Use the **Node > By XYZ, Locate...** panel with the required coordinates and click the **Create Node** tab to create the 16 points:

Pt 1	Pt 2	Pt 3	Pt 4etc	Pt 16
0, 0, 0	0, 3.0, 0	90.0, 0, 0	90, 3.0, 0		220.0, 6.3, 0

Use the default ID numbers. The ID numbers and the order that the nodes are defined is not important.



Tutorial 5,6: Single and Double lap adhesive joints

Positioning and zooming the model:

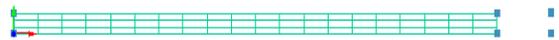
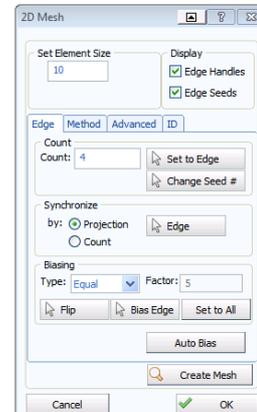
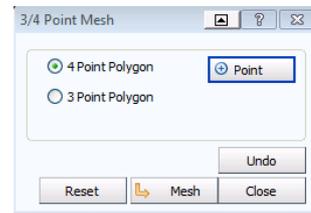
The positioning and viewing tabs explained in tutorials 1 through 4 are used to center and manipulate the position/viewing angle of the nodes specified. Try to obtain the view shown below.



Meshing Patch 1 (lower adherend)

For each of the seven surface patches the following steps are done:

1. Generating the surface. Conveniently position the model (e.g. using the **XY tab** and **Center tab**). Then activate the **2D > 3/4 Point Mesh** tabs with the **4 point polygon** option. Click on the four corner nodes in sequence (clockwise or anti-clockwise) for the patch representing the longer length of the lower adherend; on clicking the last node the surface is generated as shown.
2. The **Mesh** tab is then clicked and definition of the mesh can begin.
3. The adjacent panel will appear together with a proposed mesh, which we shall now modify. The mesh density can be changed either by specifying an "element size", or clicking on the numbered boxes with the left mouse key. Click twice on each of the boxes for the thickness direction of the adherend to change from 1 to 3 (left mouse key). Use shift and the left mouse key to lower numbers. Similarly, click on the length direction boxes to change these from 9 to 20. Then click **Create Mesh** to visualise the mesh.
4. Finally, the regular mesh is accepted with **OK**.



Tutorial 5,6: Single and Double lap adhesive joints

Meshing the transition zone (Patch 2)

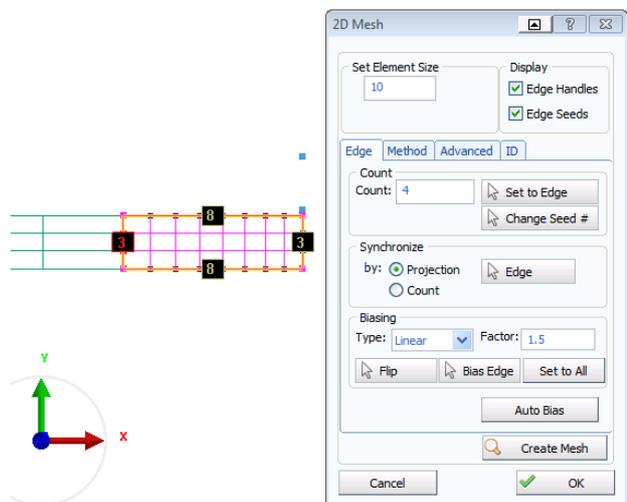
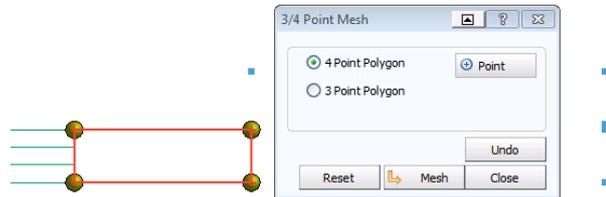
Between each adherend and the adhesive zone a graded mesh is made as follows:

1. Generating the surface using the **2D > 3/4 Point Mesh** tabs with the **4 point polygon** option and click on the four corner points. Click on **Mesh** to start the meshing.
2. Change the number of elements to 3 and 8 for the height and length directions respectively.

In the thickness direction the default regular mesh is used. For the length direction a graded mesh is now defined; first for the bottom line and then for the top. In option **Biasing** use **Type linear**; use a **Factor = 1.5** (the first element is 1,5 times larger than the last element in the line). Click **Bias Edge** and then on the line to be graded, if the mesh grading is in the reverse direction use the **Flip** tab to reverse the Bias; we want the finer mesh to be at the adhesive end where the stress concentrations are obviously highest.

Repeat the operation for the top line.

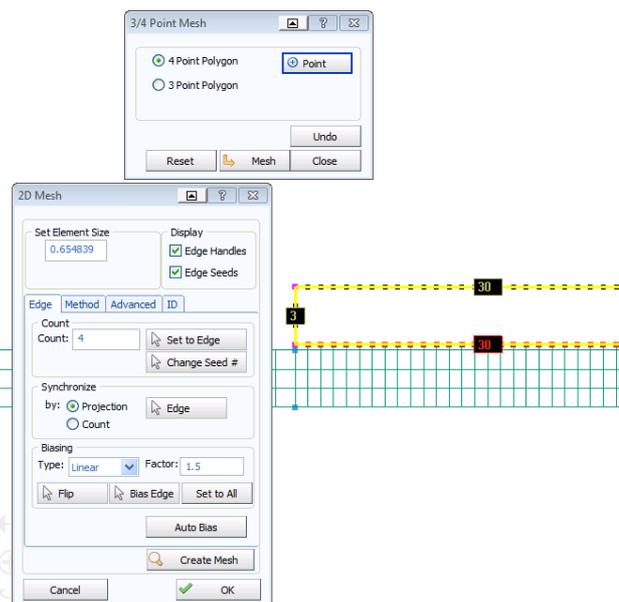
3. Once the edges have the correct element numbers and grading click **Create Mesh** to visualise the mesh and **OK** to accept.



Meshing patches 3, 4 and 5

For each of these patches a regular mesh is used. The adherend parts (patches 3 and 5) have a 3*30 element mesh and the adhesive layer has a 1*30 element mesh. The steps used are identical to those above for patch 1, except for the different element densities.

Remark: Note that the number of elements assigned to each of the side are consistent with adjacent patches. This is important for a proper 'continuous' mesh across the patch boundaries.

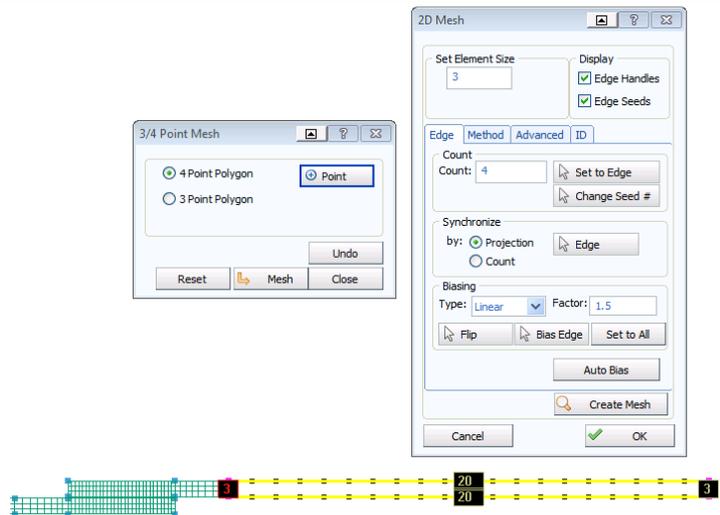


Tutorial 5,6: Single and Double lap adhesive joints

Meshing adherend patches 6 and 7

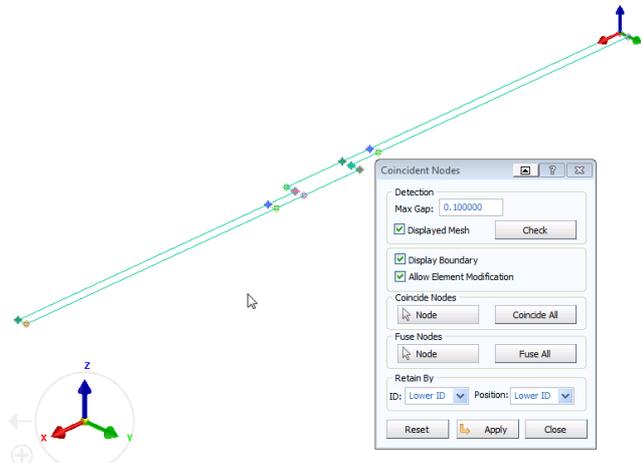
Patch 6 is meshed in identical fashion to patch 2 with the same mesh grading; again the smaller elements are at the adhesive end (i.e in the reverse direction to patch 2!).

Patch 7 is has a regular mesh and is meshed in the same way as previously done for patch 1.

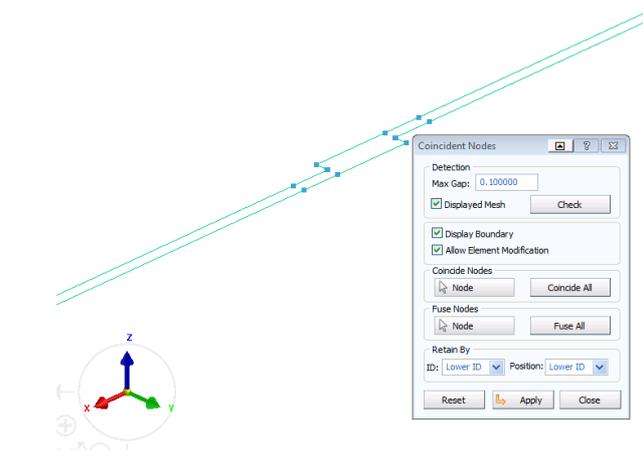


Combining the patches

In the meshing process so far seven individual patches have been created and meshed. At the boundaries of each patch the element spacings have been carefully chosen so there is mesh continuity. However, it is possible that each patch has separate nodes and the elements are discontinuous. Or other discontinuities occur.



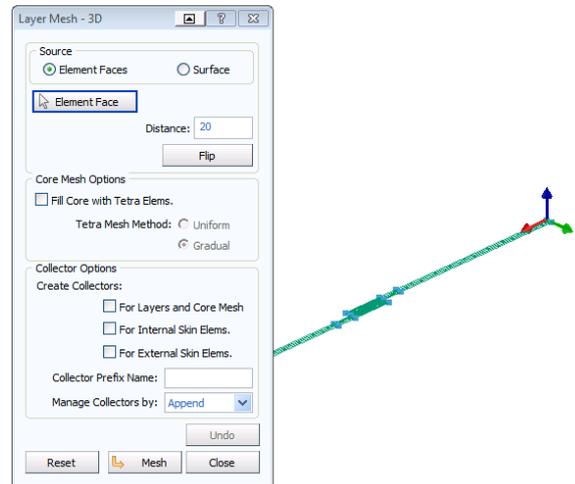
Check this using the coincident nodes option. Click **Checks > Coincident Nodes** and the option **Check**. This gives a view showing all duplicated nodes in the model (a suitable **Max Gap** has to be selected if the default is inappropriate). These duplicated nodes are merged with the **Fuse All** and **Apply** tabs to give the second view. Finish with **Close**.



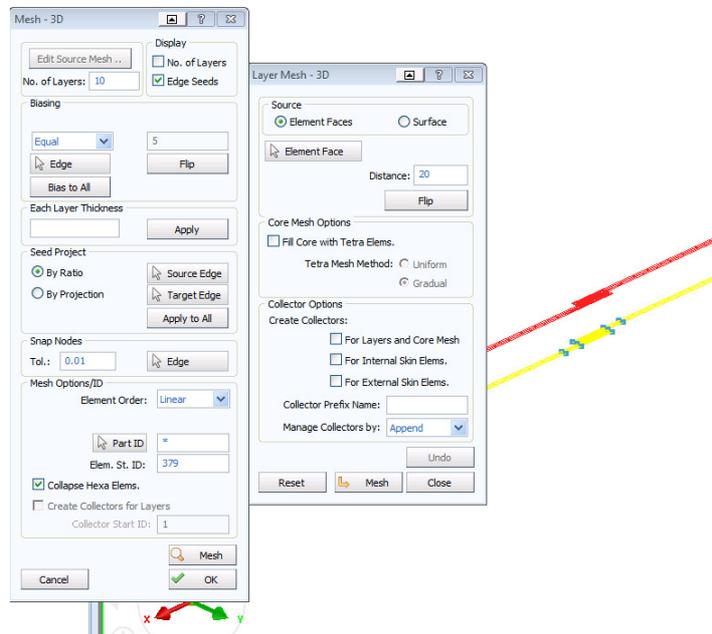
Tutorial 5,6: Single and Double lap adhesive joints

Creating the 3D Solid mesh

1. The 3D (solid) mesh is generated from the 2D (shell) mesh by clicking **3D > Layer Mesh (Grow)**. Enter a Distance of 20 (for a 20mm wide lap joint). The **Elements Faces** option is activated then select all elements by dragging a box over the model with the left mouse key and click the green arrow to select. The mesh (partially shown below) should appear, together with a new panel for defining details of the mesh to be created in the width direction.

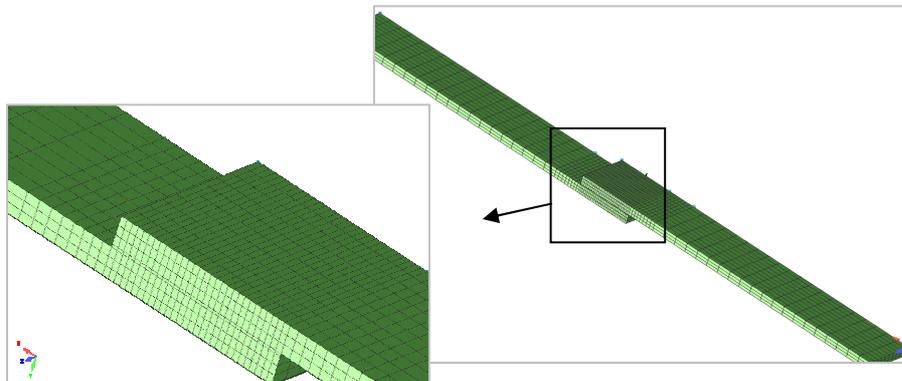


2. In the new panel enter a value for the No. of Layers which represents the number of elements in the width direction. Change the default 13 to **10**; click on **Mesh** to visualise and **Apply** to accept.



The final 3D solid mesh

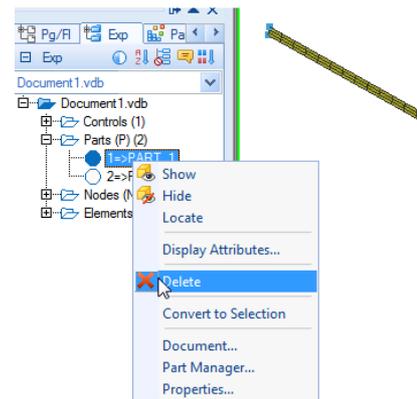
The following views using perspective, ploygon filling and zoom tabs show the mesh generated for the 3D solid adhesive joint.



Tutorial 5,6: Single and Double lap adhesive joints

Deleting the (now redundant) 2D shell elements

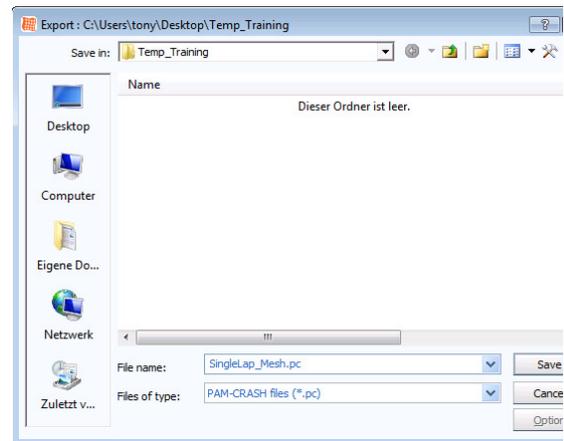
Before finishing the meshing part of the work the shell elements, which were used as a basis to generate the solid elements, must be deleted. Click on the **[+]** symbol of **Parts** using the left mouse key. Then click on **Part_1**, which is the part for the shell elements; with the right mouse key and activate the option to delete this part. The 378 shell elements are also eliminated with the part.



Saving the model (mesh only)

This now completes meshing of the single lap joint with **Visual Mesh**. Export the data with **File > Export** to a suitable folder with a suitable name (e.g. **SingleLap_Mesh.pc**). Then exit and close **Visual Mesh**.

The next steps apply boundary conditions, loading, materials data, etc.,



Defining the model data (Visual-Crash PAM)

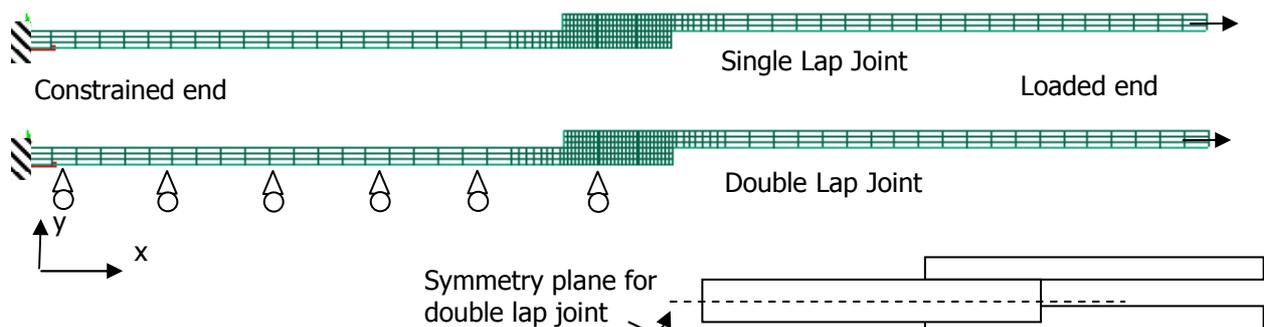
Preparation of the model(s) for analysis

Single Lap Joint:

- The constrained end (below) is fully fixed.
- The loaded end is constrained in all directions except x, and a velocity load curve is applied in this direction.
- The part and material data are applied: The adherends are Aluminum and the three cases will be considered for the adhesive; namely, elastic, elasto-plastic and elasto-plastic with failure.
- Application of the PAM-CRASH control data.

Double Lap Joint:

The single lap joint is also used for the double lap joint except that symmetric boundary conditions are applied to all nodes (y direction fixed) on the lower half of the joint.



Renaming the datasets

It is probably best to copy the meshed dataset to a new name and use this for the following work. This will allow you to start again with a new copy if things go wrong!

Copy:

SingleLap_Mesh.pc to **SingleLap_Elastic.pc** for the elastic analysis

The model **SingleLap_Elastic.pc** will be used to build the model for the elastic analysis of the single lap joint. This will then be used as a basis to build the elasto-plastic, failure, double lap, and implicit analysis models.

Preparing the Single Lap joint analysis model: Elastic study

Starting the Visual Crash PAM work

Start **Visual Crash PAM** and open the copied model file using **Open File**. Use the copied dataset from the previous section:

SingleLap_Elastic.pc

Positioning and centering of the model

Use the x-y and centering tabs to conveniently position the model. This will make assignment of the boundary constraints, loadings and other data easier. Also, in this case visualisation and assignment of data is easier if the 'wireframe' visualisation option is activated in the panel.



Boundary conditions

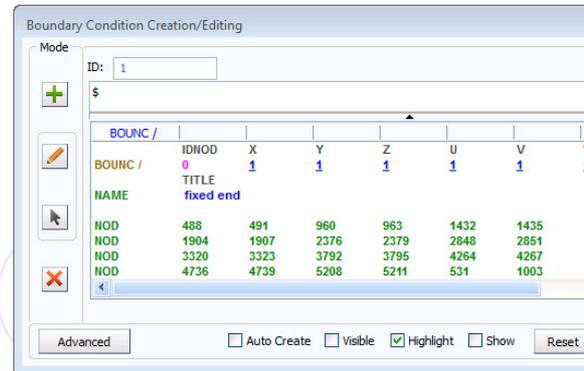
Constraining the fixed end

Click **Crash > Loads > Displacements BC** to get the Boundary Condition panel. Use the left mouse key to draw a box over all the nodes on the left end then click the **Green Arrow** to update the selection. The selected nodes will appear in the nodal list.



Fully constrain these nodes (x,y,z components = 1 for fixed). Note that the rotational degrees of freedom U,V,W are not present in Solid elements and a constraint applied to these has no meaning. Constraints are activated by clicking on each condition and then the desire constraint integer.

Choose an appropriate title; e.g. **"fixed end"** and save the selection with **Apply** and **Close**.



Constraining the loaded end

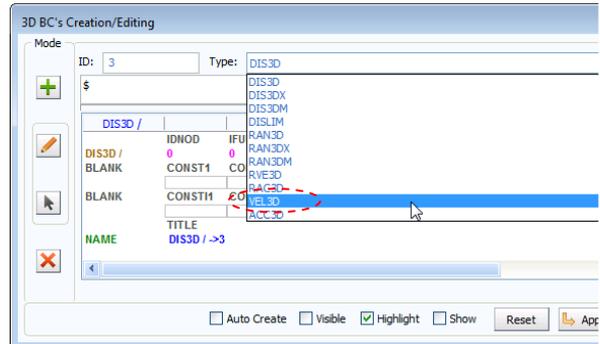
The same procedure is done at the other end where loading will be applied, except that the nodes are only constrained in the y and z directions (x=0 free).

Tutorial 5,6: Single and Double lap adhesive joints

Loading conditions (at the loaded end)

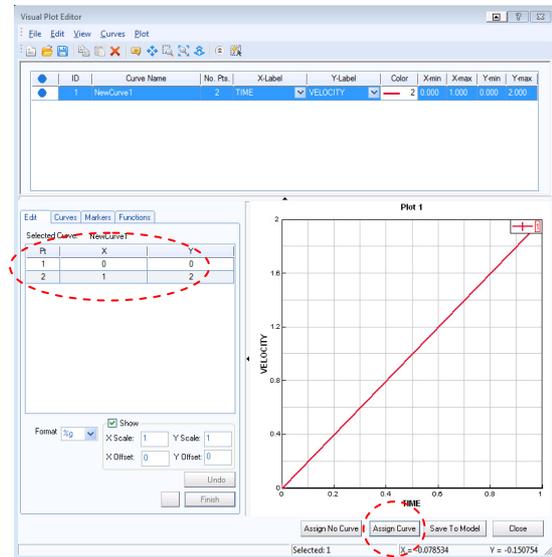
Different types of loading could be applied to the loaded end such as force, velocity or displacement control in the x-direction. Since failure will be modelled in a later exercise, it is probably best to use either velocity, or displacement control since force control can lead to rapid accelerations once failure is initiated. Velocity is used here.

Click on the tabs **Crash > Loads > 3D BC** and in the box Type activate **VEL3D** to get the panel to assign velocities.



Use the left mouse key to draw (drag) a box over all the nodes on the loading end and click the **Green Arrow**. The selected nodes will now appear in the nodal list.

The direction is identified by the IFUN number. Click IFUN1 for the x direction and use **File > New** to open a panel to define a velocity time history curve.

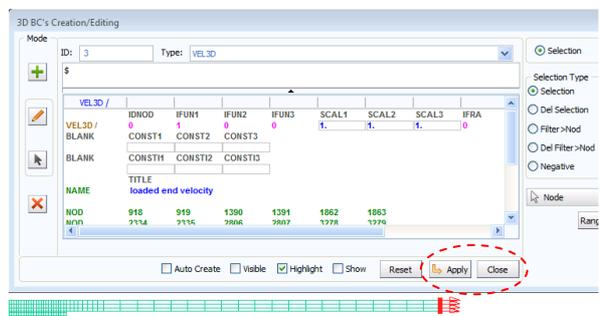


In the new panel the x-axis is time and y-axis is velocity. Assume the velocity increases linearly to 2mm/msec at 1msec: this equates to a total displacement of 1.0mm. Click **Apply** and **OK** to close this panel.

Some final points:

- The applied velocities could be scaled, if needed, but we shall use SCAL1 = 1
- Give the nodal velocities a suitable title so it can be easily identified .
- Make sure the velocity curve (IFUN1) is assigned to this set of nodes.

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



Tutorial 5,6: Single and Double lap adhesive joints

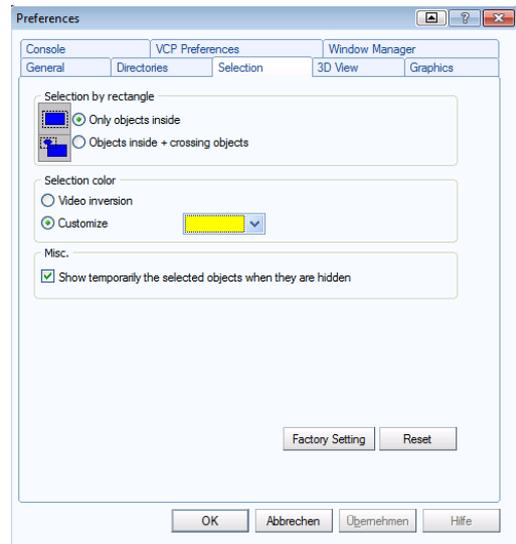
Assigning the Elastic Material and Part data

This is done using two entities which are linked together; namely:

1. The Material data for material data such as modulus, density and plasticity information.
2. The Part data for (geometric data such as thickness).

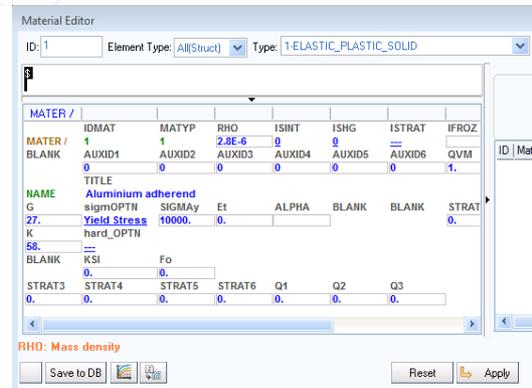
First, a new part is made for the adhesive elements (currently the model only has one PID number = 2):

- Click on the tabs **File > General Preferences > Selection** to open the adjacent panel. Make sure that the "Only objects inside" option is active. This will allow easier selection of the single row of adhesive solid elements.
- Make sure the entity selector "Solid-By-Solid" is active and then drag a box over the adhesive solids (left mouse key). These elements will be highlighted. Then, with the right mouse key, open the work tools option and **Change PID**. Give a new PID target number (e.g. 3) and finish with **Apply** and **OK**. The new part should get a new colour and a new part will appear in the explorer parts list.



1. Material data (Aluminium Adherend PID=2)

Select **Crash > Materials > Structural** and then select the type of Material Model; in this case use a **1 – ELASTIC_PLASTIC_SOLID**. Then define the material data for Aluminium ($E=70 \text{ kN/mm}^2$, $\nu=0.3$ and density $2.8 \cdot 10^{-6} \text{ kg/mm}^3$). The input requires the equivalent data $K=58 \text{ kN/mm}^2$, $G=27 \text{ kN/mm}^2$. Give a title and specify a high yield stress (e.g. $10,000 \text{ kN/mm}^2$) so that yielding cannot occur. Close the material assignments with **Apply** and **Close**.

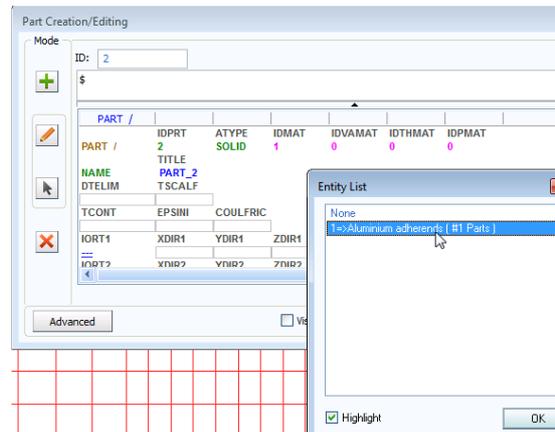


2. Part data (Aluminium Adherend PID =2)

This is easily done via the explorer panel:

1. First click on **Parts** and the list of parts will open; select the required part to open the parts panel.
2. The material to be linked to this part must be assigned. Click on **IDMAT** and a **List** option will open: Select the required material and click **OK**.
3. Give a suitable title.

Finish the assignments with **Apply** and **Close**.



Repeat for the Adhesive (PID=3): Use

$K=2.5 \text{ kN/mm}^2$, $G=1.15 \text{ kN/mm}^2$, density= $1.5 \cdot 10^{-6} \text{ kg/mm}^3$, yield stress= $10,000 \text{ kN/mm}^2$ and a new title.

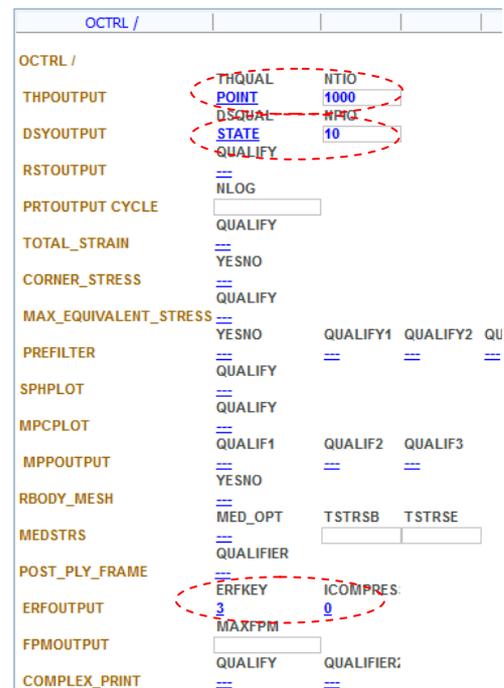
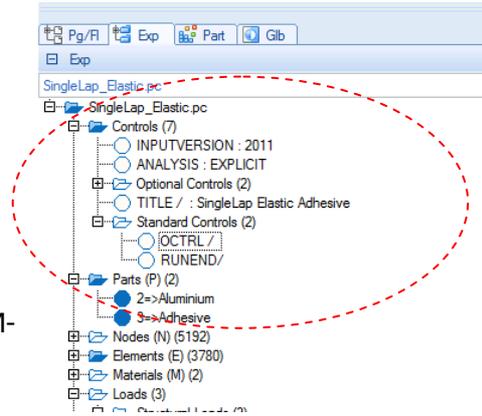
Tutorial 5,6: Single and Double lap adhesive joints

The PAM-CRASH control data must be set

This controls the analysis and allows specific data like the problem analysis time (in msec in this case), and output intervals for the results.

In the Explorer panel under controls the following controls can be opened and edited (click on the control and open with the right mouse key to edit):

1. INPUTVERSION: Set this (if necessary) to the latest PAM-CRASH version installed and used.
2. ANALYSIS: Open if necessary and set this to EXPLICIT.
3. TITLE: Open and give the project a suitable title (e.g. SingleLap Elastic Adhesive); this will appear on all results plots using Visual Viewer and is important to help identify the analysis.
4. RUNEND: Open and set this to (TIO2=1.0) for the solution time.
5. OCTRL:
 - a. Open and set the THPOUTPUT to POINT with 1000. This will save 1000 points of time history information for x-y graph plots in the .ERF results file.
 - b. For DSYOUTPUT set option STATE with 10 to give ten plots of the deformed structure in the .ERF results file.
 - c. In these exercises results will be saved in the .ERF file format. In the OCTRL panel activate type 3 without compression (ICOMPRES=0).
 - d. Save the information with **Apply** and **Close**.

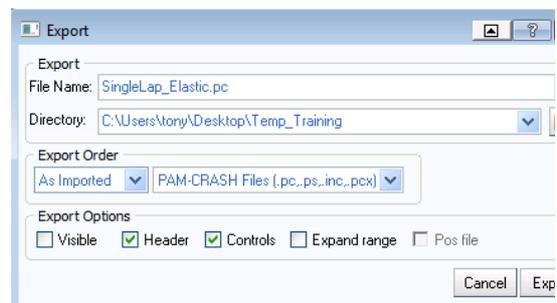


Saving the dataset (.pc file)

Save the model in a suitable directory and with a suitable name; e.g. **SingleLap_Elastic.pc**.

Click on **File > Export** and save the model as a PAM-CRASH input file (**SingleLap_Elastic.pc**).

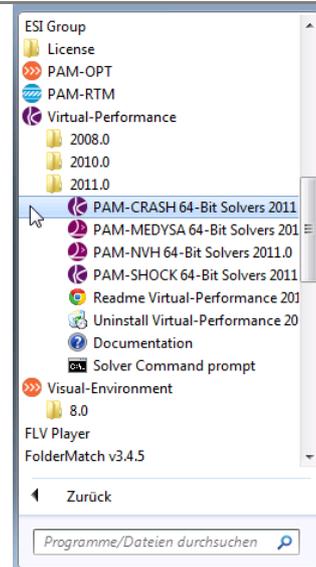
(**File > Save** will save the model in a VCP internal binary format which is more useful if CAD data is present).



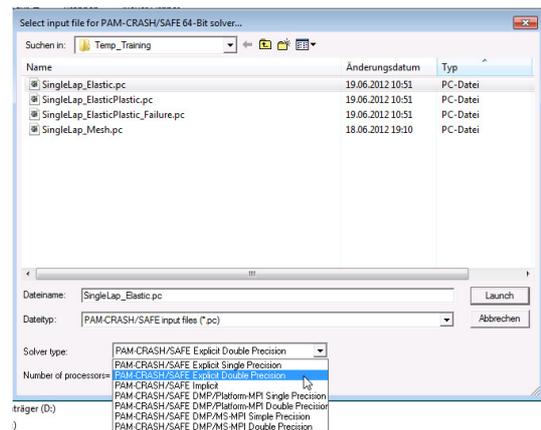
Part 2: Explicit analysis of the Single Lap joint (elastic)

Running the PAM-CRASH analysis

- Start the simulation run from the ESI Group Folder with the latest version of PAM-CRASH.



- Select the directory and your .pc file. It is also possible to select an SMP (shared memory) or DMP (distributed memory) version of the code for parallel processing and to select the number of processors you would like to run. The 'Explicit Double Precision' version is recommended for training problems.



If the dataset is good it will proceed through the dataset initialisation phase into the solution phase and terminate with 'NORMAL TERMINATION'.

- If there are data errors the run will stop with an abnormal termination message. Inspect the output/results files for errors (search for 'ERROR' and investigate). Correct the dataset; preferably in VISUAL CRASH PAM (or in the editor) and rerun the analysis.

NORMAL TERMINATION

```

TOTAL NUMBER OF CYCLES = 6262
AVERAGE TIME PER ZONE CYCLE = 0.9709E+02
NUMBER OF SHELLS ELIMINATED = 0
NUMBER OF SOLIDS ELIMINATED = 0
NUMBER OF BEAMS ELIMINATED = 0
NUMBER OF SPHS ELIMINATED = 0
    
```

	CPU (s)	ELAPSED (s)	CPU%	ELAPS%
INPUT/INITIALIZATION	1.0000E+00	1.3000E+01	1.32	34.21
INTERNAL FORCES	2.6000E+01	1.1000E+01	34.21	28.95
NODAL OPERATIONS	2.3000E+01	8.0000E+00	30.26	21.05
OUTPUTS	2.4000E+01	6.0000E+00	31.58	15.79
CONTACTS	0.0000E+00	0.0000E+00	0.00	0.00
CONSTRAINTS (RB, RW, BC...)	2.0000E+00	0.0000E+00	2.63	0.00
KJOINT/MBS SOLVERS	0.0000E+00	0.0000E+00	0.00	0.00
ADAPTIVE MESH (STAMP)	0.0000E+00	0.0000E+00	0.00	0.00
TOTALS	7.6000E+01	3.8000E+01	100.00	100.00

```

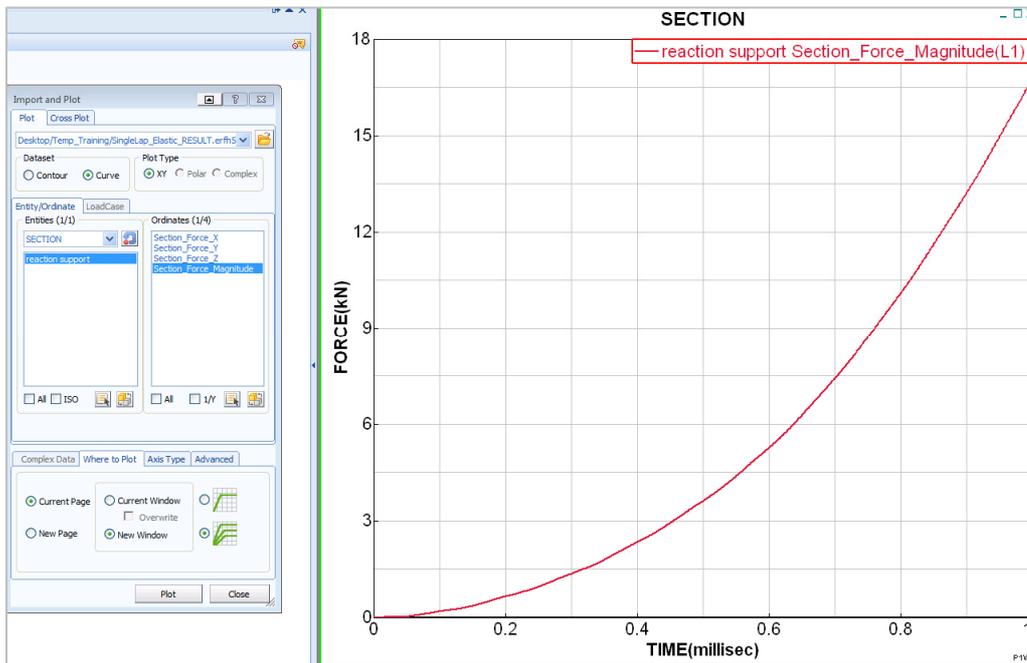
CPU TIME 7.600E+01 (s)
ELAPSED TIME 3.800E+01 (s)
RATIO CPU/ELAPSED TIME 200.00 %
    
```

NORMAL TERMINATION, EXITO MESSAGE

Tutorial 5,6: Single and Double lap adhesive joints

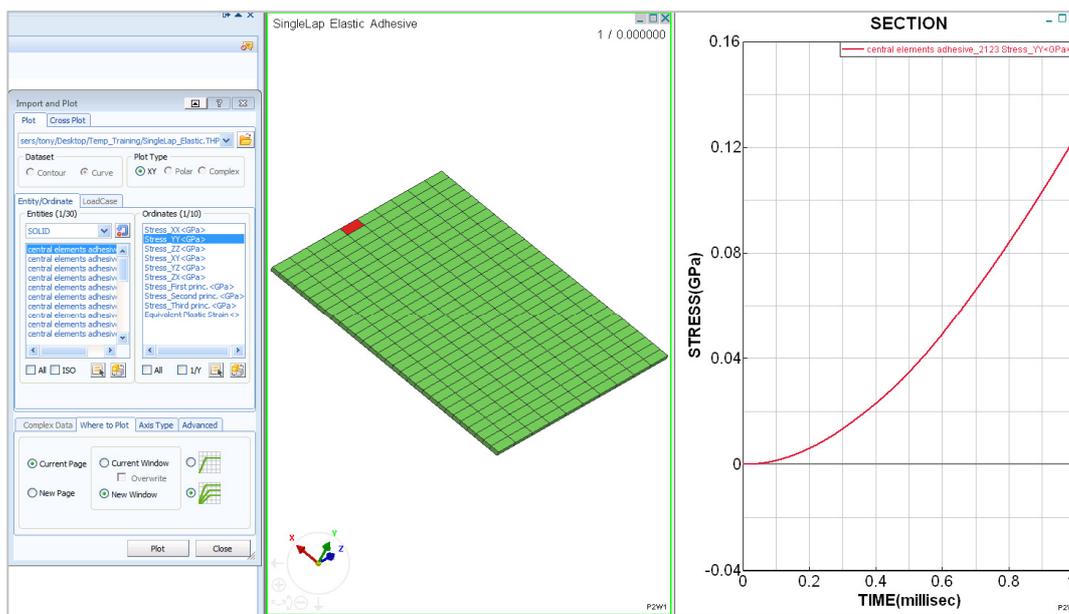
Time history plot results

In **Visual Viewer** time history information can also be extracted from the .ERF file. For example select entity **SUPPORT** and component **Section_Force_Magnitude**.



In the **Import and Plot** panel it is also possible to plot time history information for each of the solid elements that were saved in the **Visual Crash PAM** session.

The plot below shows the adhesive peel stress (stress components YY) for the centre edge solid element shown. You will need to have opened all files with the **File > Open Project** option in order to visualise the element being selected.



Part 3: Explicit analysis of the Single Lap joint (elasto-plastic and elasto-plastic with failure)

This section will perform an explicit simulation of the Single Lap joint using:

1. An elasto-plastic law for the adhesive.
2. An elasto-plastic with failure law for the adhesive.

Copy the files:

SingleLap_Elastic.pc to **SingleLap_ElasticPlastic.pc** for the elasto-plastic analysis.

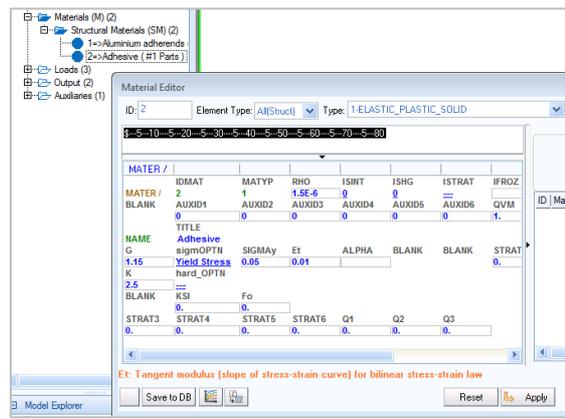
SingleLap_Elastic.pc to **SingleLap_ElasticPlastic_Failure.pc** for the failure analysis.

For the two new models the only changes that need to be made are to the material laws:

1. Material modifications for the elasto-plastic adhesive model

Start **Visual Crash PAM** and read in **SingleLap_ElasticPlastic.pc**. In the explorer panel open Materials (click on [+]) and then click on the elastic adhesive material with the right mouse key; select option **Edit**.

Enter 0.05 (MPa) for the yield stress and a small plastic tangent modulus (= constant hardening slope) of 0.01 MPa. Finally, enter a new title for the material and close with **Apply** and **Close**.

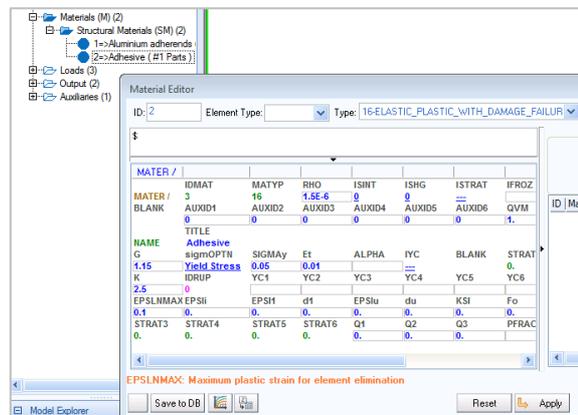


A better problem title should be used (**Explorer / Controls > Pam Controls > Title**) such as "SingleLap_Elastic Plastic adhesive". The modified dataset is exported with the new name **SingleLap_ElasticPlastic.pc** and close **Visual Crash PAM**.

2. Material modification for the elasto-plasticity with failure model

Repeat the above steps with dataset **SingleLap_ElasticPlastic_Failure.pc**.

In the explorer panel open Materials (click [+]) and then click the elastic adhesive with the right mouse key; select option **Edit**. In this case under Type select the material **type 16** which allows elasto-plastic behaviour and material failure. The same data as above is entered, together with a maximum elasto-plastic shear strain failure criteria (EPSLNMAX=0.1). Finally, enter a new title and close with **Apply** and **Close**.



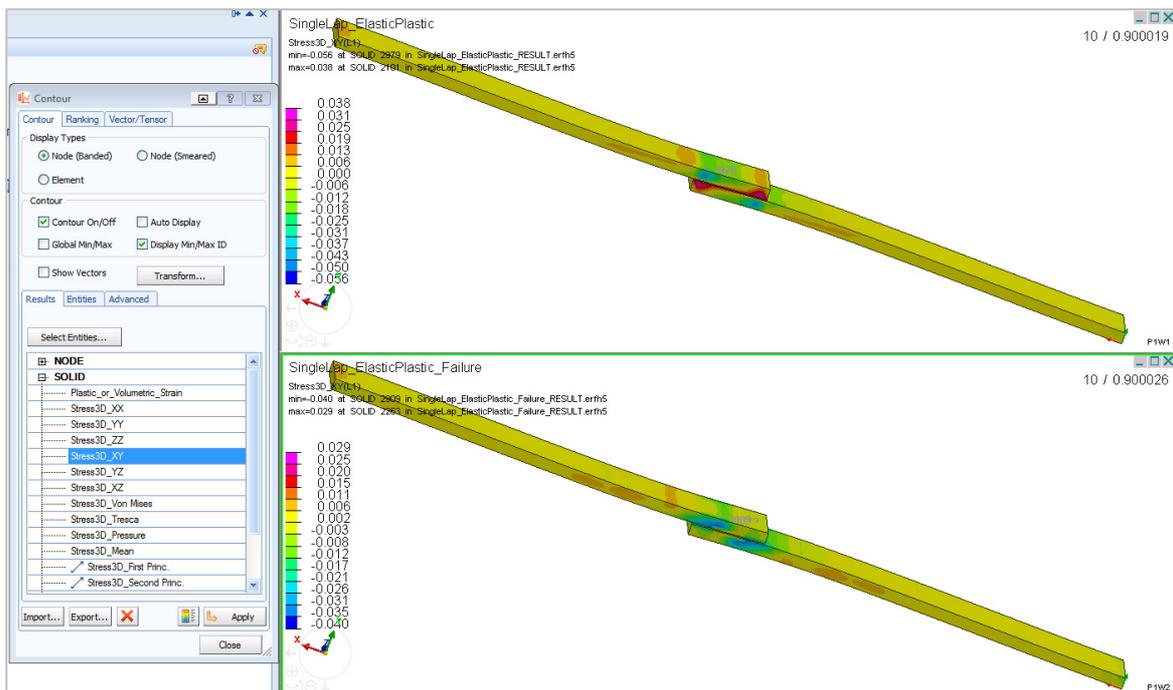
Beware, a new material model has been defined and it will be necessary to re-assign this material to the adhesive part. In the explorer panel select Parts and then the adhesive material in Edit mode; reassign the adhesive material to this part. The problem title should be updated to mention failure and the modified dataset exported with the new name **SingleLap_ElasticPlastic_Failure.pc**. Finally close **Visual Crash PAM**.

Tutorial 5,6: Single and Double lap adhesive joints

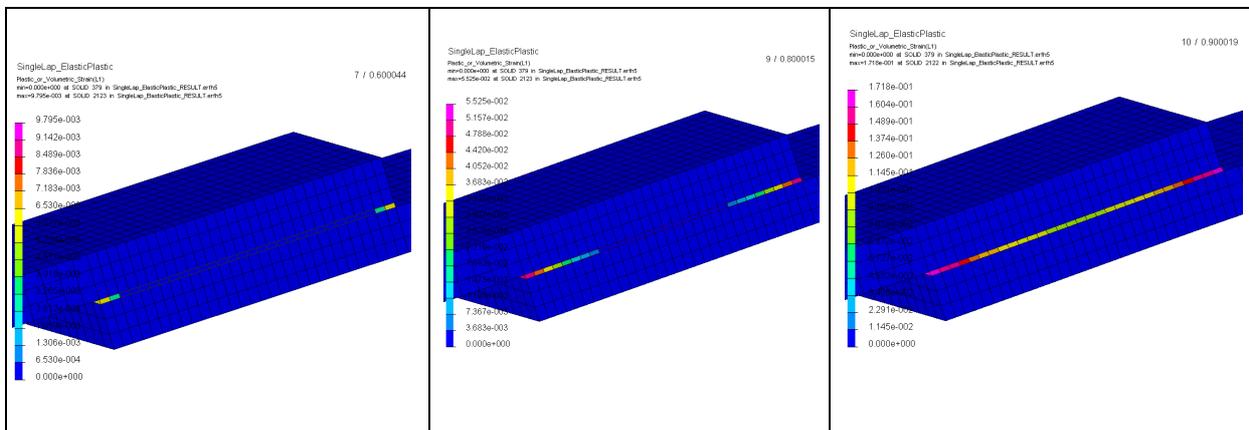
Mesh plot results (Elasto-plastic only: Dataset SingleLap_ElasticPlastic.erfh5)

Start the **Visual Viewer** and use the split screen viewing option (2 screens split horizontally). Open the elastic plastic model to the top screen and the elasto-plastic with failure results to the lower.

Synchronise the two screens/models and position them both to a suitable viewing angle. The following plot illustrates a typical view that may be made, for example shear stress component xy at 0.9 msec. Careful examination of the two models shows results are identical to 0.8 msec. Thereafter the joint with failure starts to break and the two solutions differ. Another useful contour to check for potential adherend failure is first principle stress.



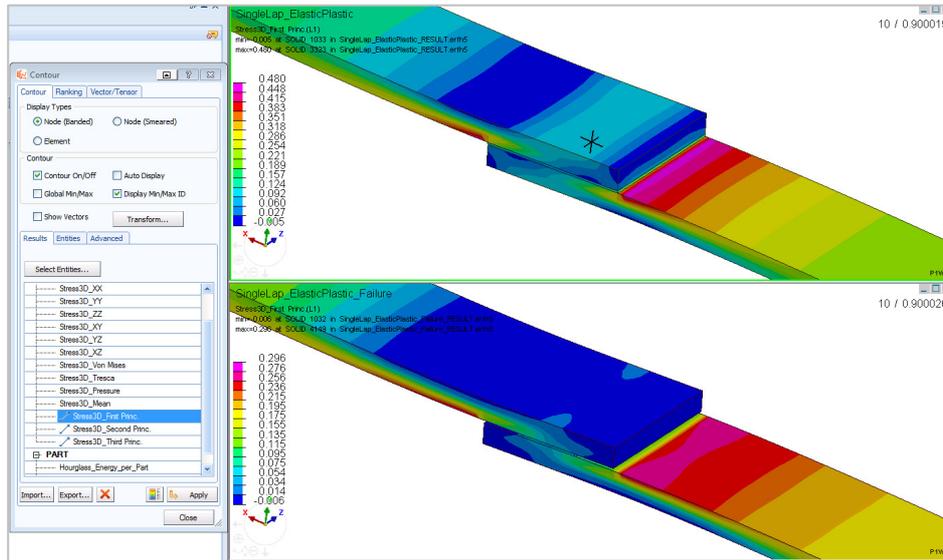
The distribution and growth of plastic strain can be seen from contours of Plastic (or volumetric strain) which start at about 0.6msec cover the full glue line at 0.9msec.



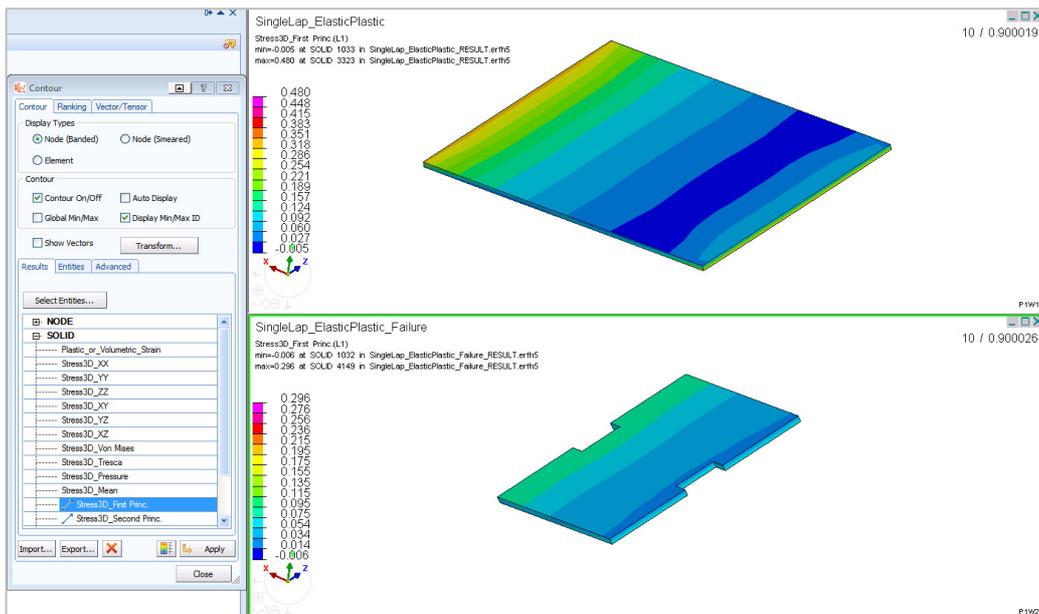
Tutorial 5,6: Single and Double lap adhesive joints

Mesh plot results (Elasto-plastic with failure: Dataset SingleLap_ElasticPlastic_Failure.erfh5)

The following plots show contours of first principle stress at 0.9 msec. At this time the adhesive is fully plastic and for the model with failure elements have started to reach the failure criteria (Von Mises plastic strain = 0.1 or 10%) and are being eliminated. The stresses in the joint without failure (top) are high (ca. 480 MPa); whereas the lower joint with failure has much lower stresses due to relaxation of loads as adhesive fails.



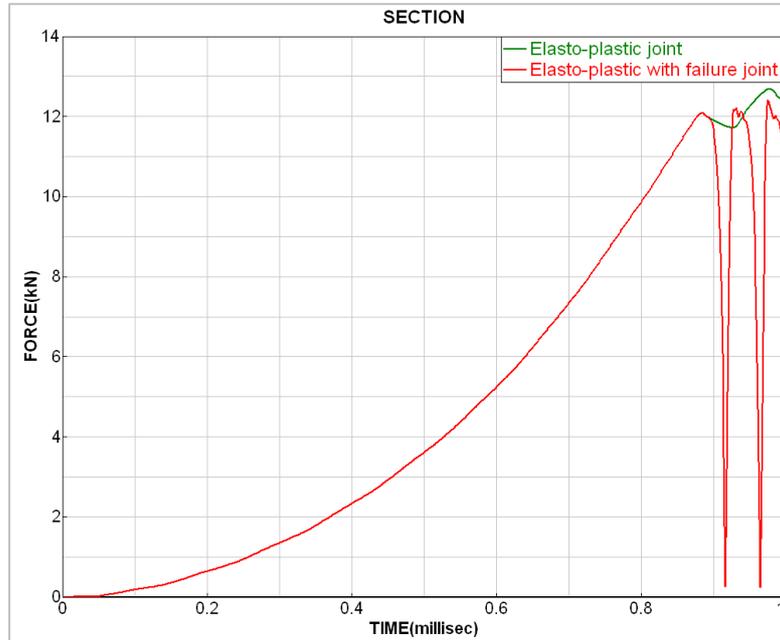
The adhesive layer can be viewed on its own by clicking on the adherend (make sure the selection entity Part-By-Part is active) and then using the **Hide Sel** tab. The eliminated elements in the model with failure (bottom) at 0.9msec are now clearly visible.



Tutorial 5,6: Single and Double lap adhesive joints

Time history results

The plot below compares the reaction force time histories (**Section > Section_Force_Magnitude**) for the elasto-plastic (no failure) and the elasto-plastic with failure joints. At about 0.9msec both joints become fully plastic causing the change in the loading slope. The joint with failure also breaks at about this time causing dynamic oscillations as elements are eliminated.



Part 4: Explicit analysis of the Double Lap joint (elasto-plastic with failure)

In order to demonstrate the method only the Single Lap joint using an elasto-plastic law with failure will be analysed as a Double Lap joint; the other joints with other material laws could be treated in the same way. Copy the file:

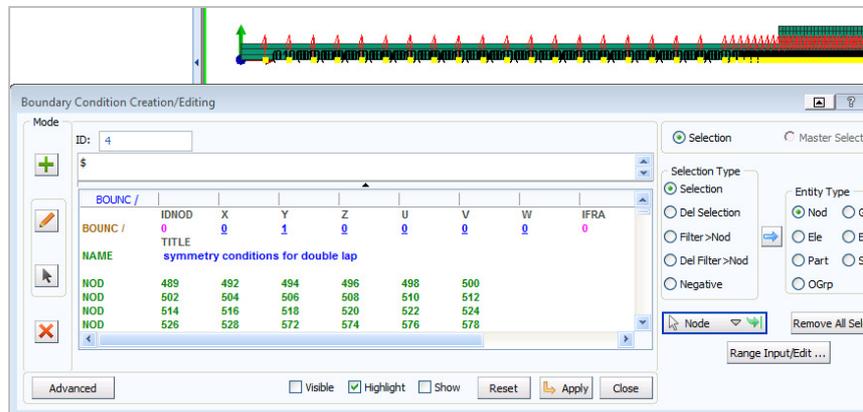
SingleLap_ElasticPlastic_Failure.pc to **DoubleLap_ElasticPlastic_Failure.pc**

The only change needed is to the boundary conditions and to give a new title for the analysis.

New boundary condition

Start **Visual Crash PAM** and read in the new file **DoubleLap_ElasticPlastic_Failure.pc**.

Suitably position the joint and then open the boundary condition panel using **Crash > Loads > Displacement BC**. Select all nodes on the lower face (except the constraint one) and click **Update Selection**.



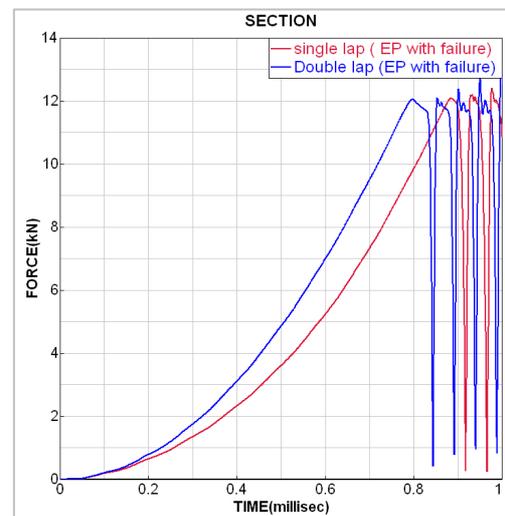
Constrain these nodes in the y direction only (for yz symmetry) and give a suitable title. Finish with **Apply** and **Close**.

The problem title should be updated to mention the Double Lap joint and then the modified dataset is exported with the new name **DoubleLap_ElasticPlastic_Failure.pc** and **Visual-Crash PAM** is closed.

Time history results (SingleLap... and DoubleLap_ElasticPlastic_Failure.erfh5)

A comparison of the Single and Double Lap joints with failure are shown using the component **Section > Section_Force_Magnitude**.

It can be seen that the results are similar with the double lap joint (blue) having a slightly stiffer response to failure due to the suppression of joint rotation. The results show similar failure loads, which should not really be the case. Generally single lap joints are weaker due to joint rotation which causes higher adhesive peel stresses that promote earlier failure. This effect has not been captured here due to the simple maximum plastic strain failure criterion that has been used.



Part 5: Implicit analysis of the Single Lap joint (static and non-linear)

Implicit analysis the Single Lap joint is demonstrated using elastic and elasto-plastic laws; the techniques used could be easily applied Double Lap joint. Failure modelling is currently not possible in the implicit code. Three implicit analyses model will be performed:

Case 1: Static linear analysis of the Single Lap joint with elastic adhesive.

Case 2: Static non-linear analysis of the Single Lap joint with elasto-plastic adhesive.

Case 3: Static geometric non-linear analysis of the Single Lap joint with elasto-plastic adhesive.

Case 1: Static linear elastic study

Copy the file:

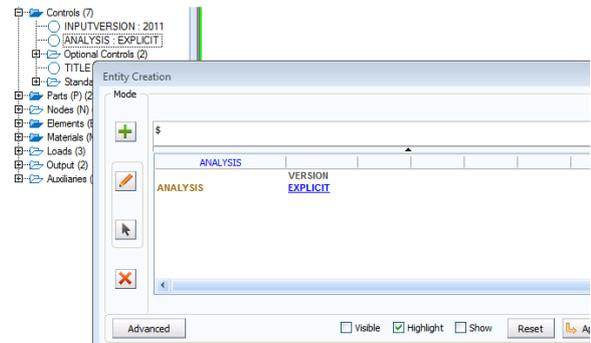
SingleLap_Elastic.pc to **SingleLap_Elastic_Implicit.pc**

Open the dataset with **Visual Crash PAM** and make the following changes:

1. The Implicit control card

In the explorer panel under **Controls > Pam Controls > Analysis** change the type of analysis from EXPLICIT to IMPLICIT.

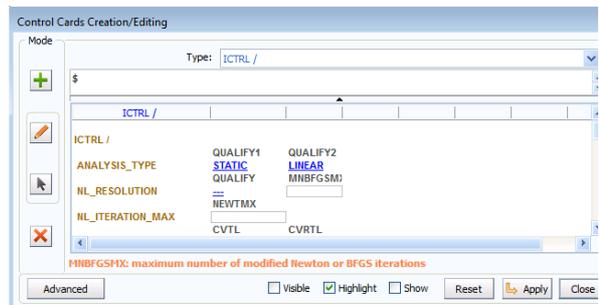
Finish with **Apply** and **Close**.



2. The type of implicit analysis

In the main window under **Crash > Controls > Standard Controls** look in the window **Type** for ICTRL. Click this and the adjacent panel opens. In the line ANALYSIS TYPE activate **STATIC** and **LINEAR**.

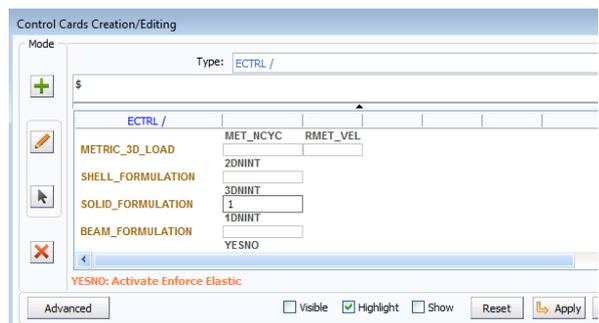
Finish with **Apply** and **Close**.



3. The element type

In the main window under **Crash > Controls > Standard Controls** look in the window **Type** for ECTRL. Click this and the adjacent panel opens. In the line SOLID FORMULATION activate type **1**.

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

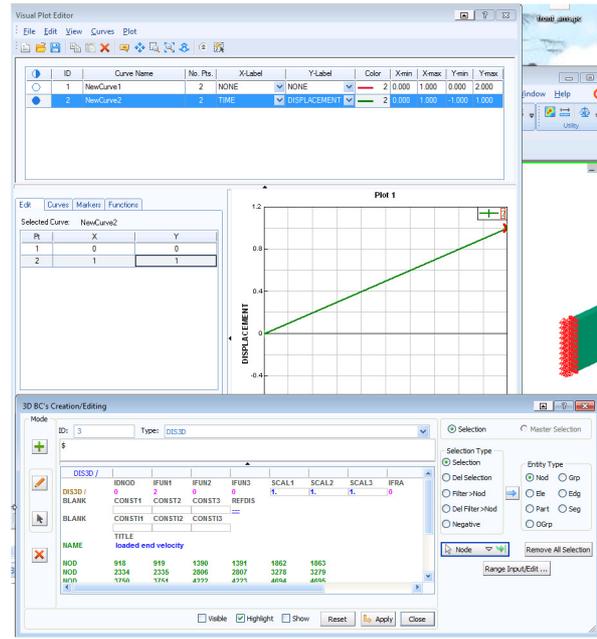


Tutorial 5,6: Single and Double lap adhesive joints

3. Changing the loading

Velocity loading (which we used in the previous explicit analyses) is not possible in Implicit.

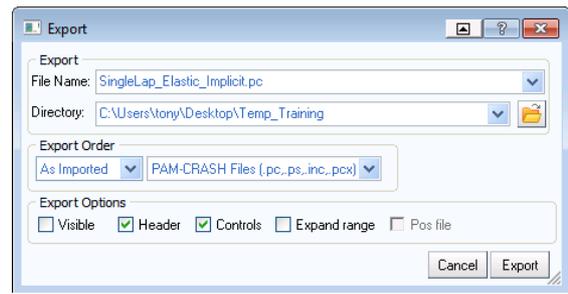
Remove the VEL3D loads and define an equivalent DIS3D (imposed displacement). Give a curve function of 0,0 to 1,1.



5. Title and exporting the PAM-CRASH dataset

In explorer **Controls > Pam Controls** open the **Title** option in edit mode and give an appropriate name; e.g. "SingleLap_Elastic adhesive IMPLICIT".

Finally export the dataset and close the **Visual Crash PAM** session.



Case 2: Static non-linear elasto-plastic study

Copy the file:

SingleLap_Elastic_Implicit.pc to **SingleLap_ElasticPlastic_Implicit.pc**

Open the dataset with **Visual Crash PAM** and make the following changes:

1. In the control cards (Type ICTRL) change the QUALIFIER" from LINEAR to NON LINEAR.
2. In the material cards for the adhesive material change the yield stress (SIGMA_Y) to 0.05 kN/mm² and the plasticity hardening slope Et to 0.01 kN/mm².
3. Give a new appropriate name for the analysis title in the control cards; e.g. "SingleLap_ElasticPlastic adhesive IMPLICIT".
4. Under **Crash > Controls > Standard Controls** look in the window **Type** for TCTL and specify the increment parameter DT2USR = 0.1. This would impose 10 load increments for the specified 1.0 msec analysis time.
5. Export the file to **SingleLap_ElasticPlastic_Implicit.pc** and close the **Visual Crash PAM** session.

Case 3: Static geometric non-linear elasto-plastic study

Copy the file:

SingleLap_ElasticPlastic_Implicit.pc to **SingleLap_ElasticPlastic_Implicit_GNL.pc**

Open the dataset with **Visual Crash PAM** and make the following changes:

1. Give a new appropriate name for the analysis title in the control cards; e.g. "SingleLap_ElasticPlastic adhesive IMPLICIT GNL".
2. Under **Crash > Controls > Standard Controls** look in the window **Type** for ICTRL and change the qualifier 2 from NON LINEAR to GEOMETRIC NON LINEAR.
3. Export the file to **SingleLap_ElasticPlastic_Implicit_GNL.pc** and close the **Visual Crash PAM** session.

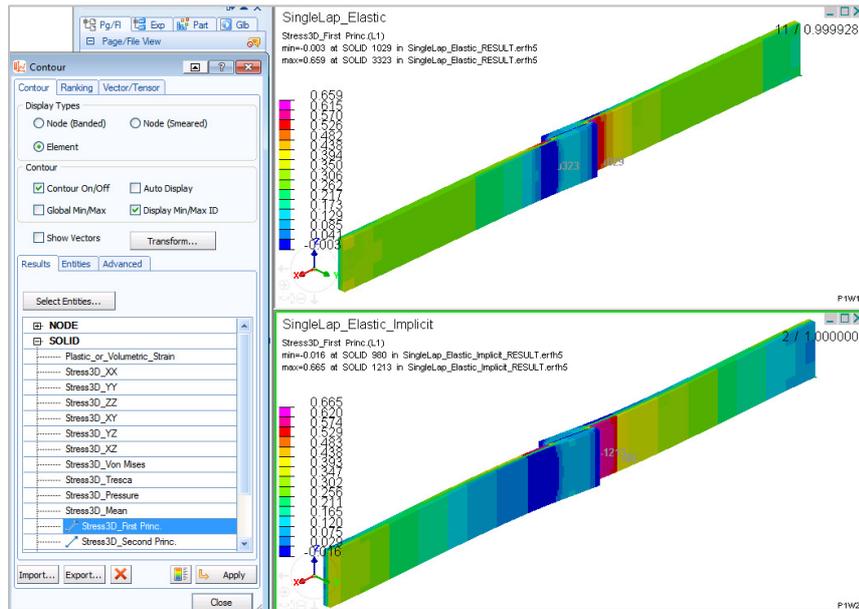
The three analysis models are run using **PAM-CRASH Implicit**.

Comparing analysis results

The following illustrates typical results that can be investigated from the analyses in this exercise.

Comparison of the explicit (top) and static linear implicit (bottom) analyses

The following shows that a good agreement between the implicit and explicit results (single lap - linear adhesive). The implicit results tend to slightly over predicted stresses due to errors in assuming this problem is geometrically linear.



Comparison implicit non-linear (top) and implicit geometric non-linear (bottom) analyses

Contour plots of first principle stress are shown below for the non-linear and geometric non-linear implicit solutions at the last deformation state. The first principle stresses differ by about 20% and show the importance of using the geometric non-linear option for this joint type. The Double Lap joint does not undergo large joint rotations and therefore does not require a geometric non-linear analysis.

