

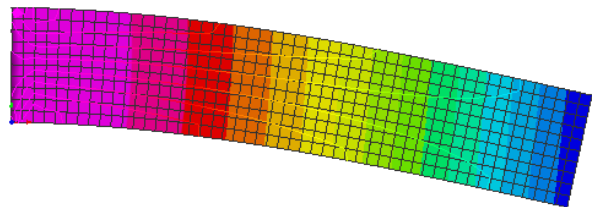
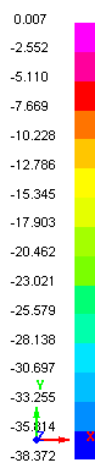
Tutorial 1 and 2

End Loaded Cantilever and

Shell Element

Studies

Cantilever_10mmMesh
Displacement_Y
min=-38.372 at NODE 8
max=0.007 at NODE 555



Problem description

Outline An end loaded 'built in' cantilever beam is analysed to demonstrate different analysis types (implicit and explicit) and different shell element types available in PAM-CRASH

Analysis type(s): Implicit and Explicit, 2D analysis

Element type(s): Shell: Quadrilateral (Belytchko-Tsay, Hughes-Tezduyar), Triangle (Belytchko-Tsay)

Materials law(s): Elastic

Model options: Boundary conditions, Nodal loads, Nodal damping, Hourglassing

Key results: Displacements, Convergence

Prepared by: Anthony Pickett, ESI GmbH/Institute for Aircraft Design, Stuttgart

Date: November 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 and Implicit):** To perform the Finite Element (FE) analyses.
- **Visual-Viewer:** Evaluating the results for contour plots, deformations, etc.

Prior knowledge for the exercise

No prior Visual or PAM-CRASH knowledge is required for working through this exercise.

Problem data and description

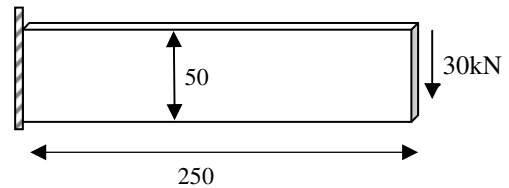
Units: kN, mm, kg, ms

Description: Plate 250mm*50mm, thickness 2mm

First analyse using a regular 10*10mm quadrilateral mesh (this will later be refined to 5*5mm and 2.5*2.5mm for the convergence study).

Loading: Imposed total load = 30kN (this is to be distributed at the loading end)

Material: Steel ($E=210\text{kN/mm}^2$, $\nu=0.3$ and density $7.8 \cdot 10^{-6}\text{kg/mm}^3$)



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 elastic cantilever 10 mm mesh use Cantilever_10mmMesh.pc

For the elastic cantilever 5 mm mesh use Cantilever_5mmMesh.pc

For the elastic cantilever 2.5 mm mesh use Cantilever_2.5mmMesh.pc

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

Background information: Solution methods and geometry

Implicit versus explicit analysis: The PAM-CRASH code is a general purpose three dimensional (3D) code for implicit and explicit FE analysis. This exercise performs implicit and explicit analyses of a simple two dimensional (2D) structure with applied force loading.

Explicit analysis treats the structure as a dynamic problem and solves dynamic equations of motion in the time domain; it is especially efficient to solve crash, impact and similar dynamic problems, particularly if material non-linearity (plasticity..), large deformations or contact occur. For explicit analysis you will find all nodal and element quantities given with respect to time.

Implicit analysis (usually) assembles the structure stiffness matrix to solve static loading problems that are independent of time. Material non-linearity, large deformations and contact are possible with non-linear solution methods. The exercise will include an implicit geometrically nonlinear analysis.

Geometry: The PAM-CRASH code is usually only applied to 3D structures; there are no special capabilities for 1D or 2D geometries. For this exercise the 2D cantilever beam is analysed using shell elements with appropriate loading and boundary conditions to make it a valid 2D problem.

Contents

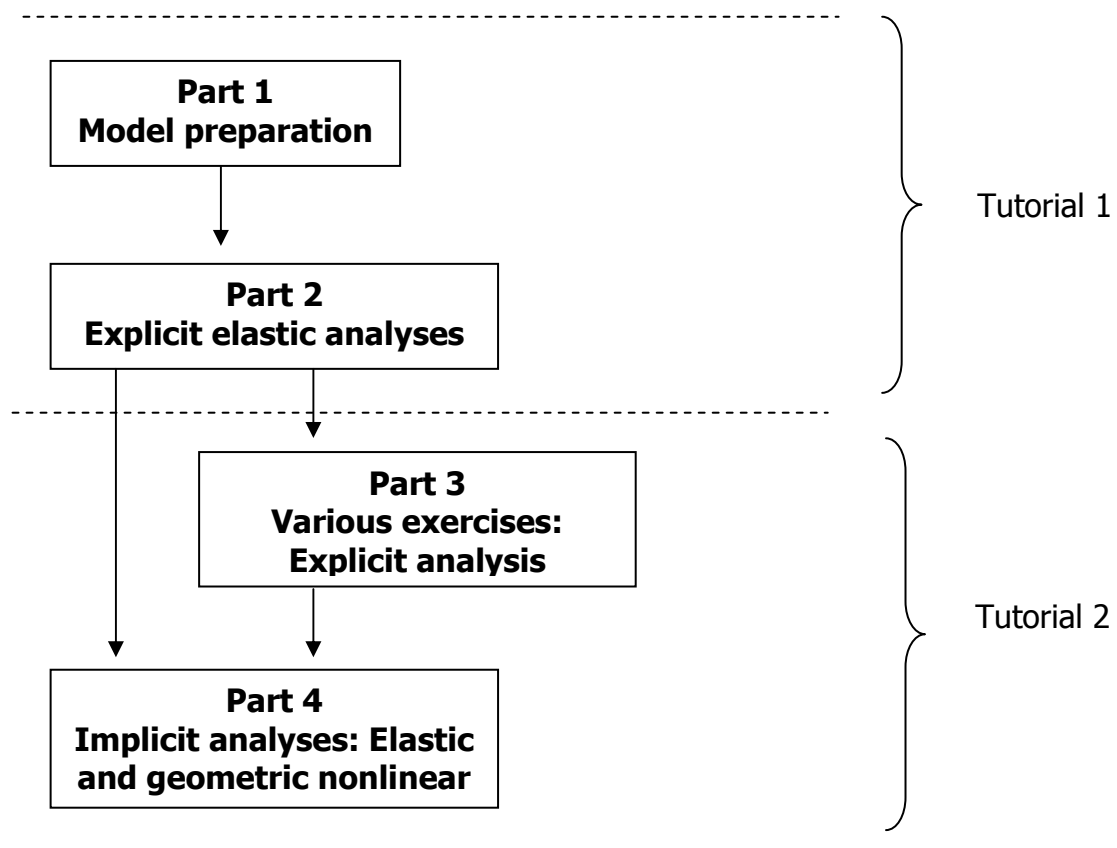
Tutorial 1 and 2	1
End Loaded Cantilever and Shell Element Studies.....	1
Problem description.....	1
Background information.....	2

[Tutorial 1 \(Model preparation and explicit analysis\)](#)

Part 1: Using VCP to make the cantilever beam model.....	3
--	---

[Tutorial 2 \(Further explicit problems and implicit analyses\)](#)

Part 2: Explicit analysis of the cantilever beam	14
Part 3: Exercises in explicit analysis.....	18
Part 4: Implicit analysis of the cantilever beam	25

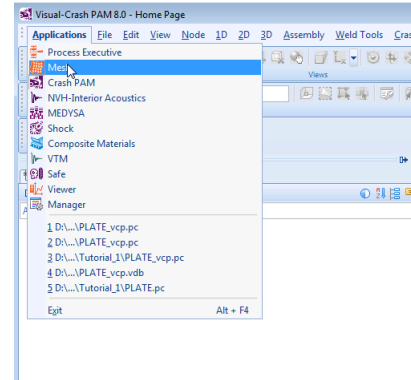


Part 1: Using VCP to make the cantilever beam model

Preparing the mesh (Visual Mesh)

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

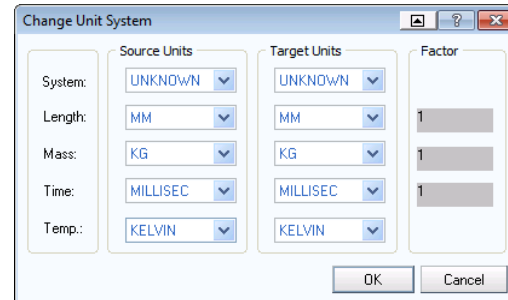
This will activate the meshing module (**Visual Mesh**) of the VCP program which is dedicated to generating finite element meshes.



Select the option **New file** then specify the model unit system:

- Set Source Units to **mm, kg, millisec, kelvin**
- The target units will be the same
- Click **OK**

This specifies the unit systems (kN, mm, kg, msec). A units conversion to a different system could be made if needed; we use the same for both.



Alternatively, an old or partially completed model could be read in for further working using **Open File** and specifying the name of the file.

Some general comments before starting

Most mesh generation programs either allow the mesh to be generated by construction information (points, lines, surfaces, volumes, etc.) or use surface and volume data from CAD packages such as IDEAS and CATIA.

We shall generate the mesh here using some simple construction features in Visual-Mesh. The usual procedure is:

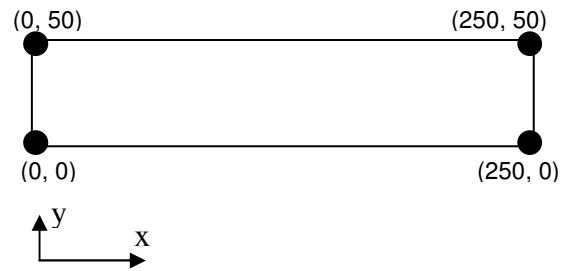
- Specify key points in x,y,z coordinates.
- Join these points with lines, arcs, circles, spline curves, etc.
- Use these lines as a basis to construct lines for 1D elements, surface (patches) for 2D elements, or volumes for 3D elements.
- Care has to be made during this process to make sure the generated mesh is appropriate to the loads, boundary conditions, part and material groupings that must be specified later in the process. **Some advanced planning is needed!**

Note: Visual-Mesh follows similar steps to make the mesh, but some useful tools are available to help simplify and speed the process.

Tutorials 1,2: End loaded cantilever and shell element studies

Planning and specification of (nodal) points

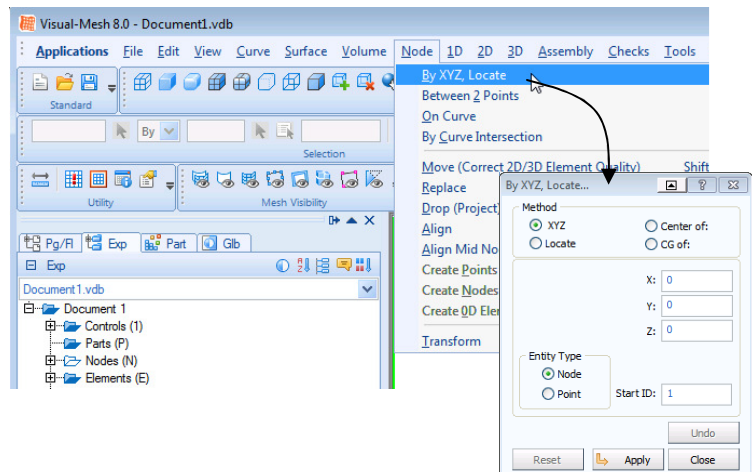
The cantilever has four points that must first be defined. It is arbitrarily chosen here that this model lies in the x-y plane with one corner at the origin ($x=0$, $y=0$) and the other points having the coordinates shown; ($z=0$ for all points).



Defining the (corner) points

The four corner nodes are defined using the **Node > By XYZ, Locate...** panel in which the coordinates for each node are entered and the **Apply** tab is clicked. The default ID numbers are used.

For the four nodes use a $z=0$ coordinate value. Note that the node ID numbers and their sequence are not important to the exercise.

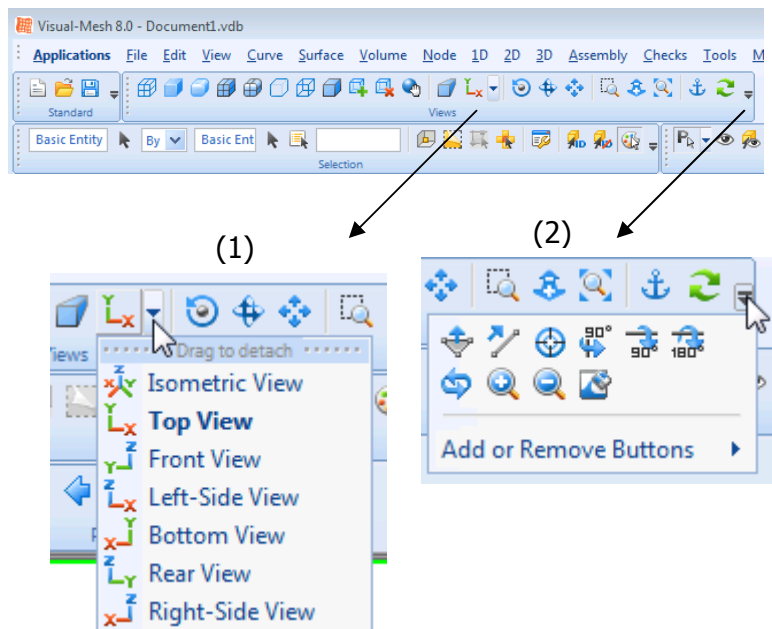


Positioning and viewing options:

Important options are available in the main (top) panel to position, center, zoom (in and out) and generally 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.

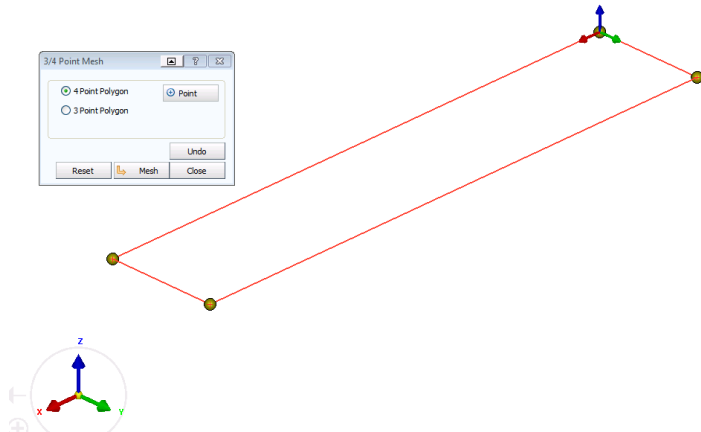
Spend 2-3 minutes to be familiar with these options; they are important. Use tab options (1) and (2) to center and position the model in isometric view.



Tutorials 1,2: End loaded cantilever and shell element studies

Generating the mesh surface

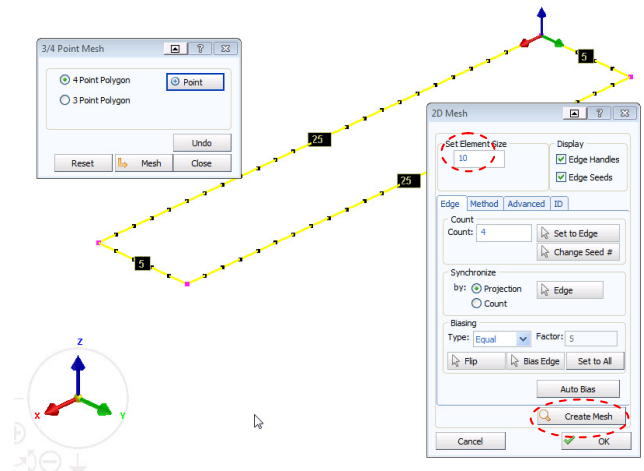
Activate the **2D > 3/4 Point Mesh** tabs with the **4 point polygon** option. Starting at any point click on the four corner nodes in sequence (clockwise or anti-clockwise). Note it may be necessary to use the previous positioning and zooming options to suitably position the nodes in the viewing frame. On clicking the last node the surface is generated as shown.



Generating the mesh

Click on the **Create Mesh** tab and a new panel opens to control the mesh to be generated. Element sizes, grading of meshed and connection options (stitching) to possible adjacent meshes can be defined.

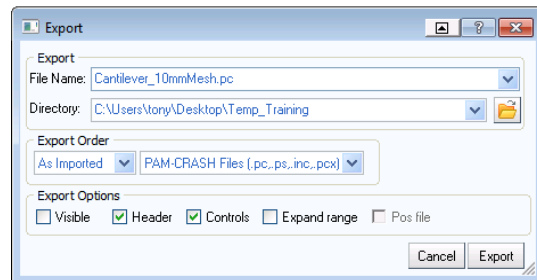
Use a 10mm element size and click the tab **Create Mesh**. The mesh will appear; accept this with the **OK** tab and then **Close** to close the meshing panel.



Ending the Visual-Mesh work

This now completes the meshing work with **Visual Mesh**. Save the data with **File > Export** to a suitable folder with a suitable name. Use the extension **.pc** which is the default name for a PAM-CRASH dataset.

E.g. a suitable name could be
Cantilever_10mmMesh.pc



Remark: It is possible to use the **File > Save** option to save data in a binary (.vcb) format, which could be useful for large models and stores additional detailed information such as CAD data and meshing information. However, PAM-CRASH only accepts the ASCII (.pc) format and so this is used as the basis to store data throughout these tutorial exercises.

Closing VCP-Mesh

Use **File > Exit** to close the meshing session.

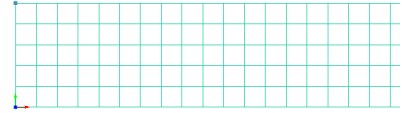
Remark: In principle you could directly continue with the next operations to define entities, materials, etc., by switching back to Visual Crash PAM (**Applications > Crash PAM**) and continuing. But it can be advisable stop, copy the dataset created to have a safe backup of the meshing work and then continuing; we shall do it this way.

Defining the model data (Visual Crash PAM)

Preparation of the model(s) for analysis

This involves:

- Starting **Visual Crash PAM** for definition of entities (Loads, boundary conditions, etc.,)
- Application of boundary conditions (fixed restraints at the built-in end).
- Application of loading at the opposite end.
- Definition of part (geometric) and material (mechanical) data.
- Definition of the PAM-CRASH control data (type of analysis – implicit or explicit) and other parameters.



Copying files and starting Visual-Crash PAM

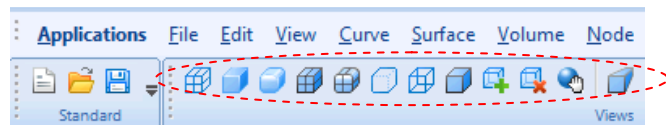
Copy the mesh file just created to a new name and work with the copied file. If things go wrong you could then go back to the mesh file and start again. Note there are no 'UNDO' capabilities in VCP; but you can always delete options and redefine them.

Copy **Cantilever_10mmMesh.pc** to **Cantilever_10mmModel.pc**.

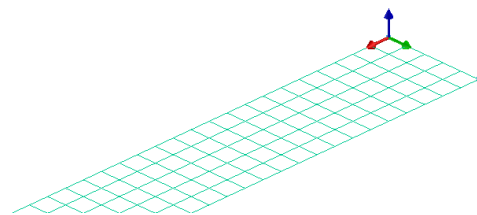
Start the **Visual Crash PAM** and read in the new file (**Cantilever_10mmModel.pc**) using **Open File**.

Further viewing options

Useful options are available in the main control panels for viewing the structure (wireframe, polygon...).

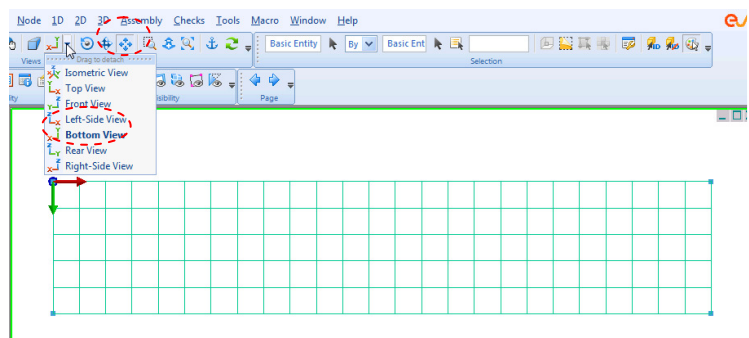


Try these and then activate the 'wireframe' option.



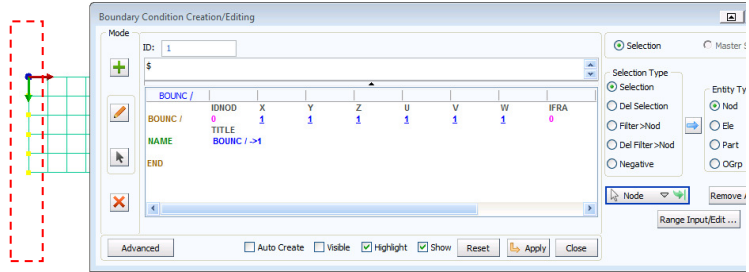
Positioning and centering of the model

Use the **x-y** and **centering** tabs to conveniently position the model as shown. This will make assignment of the boundary constraints, loadings and other data easier.



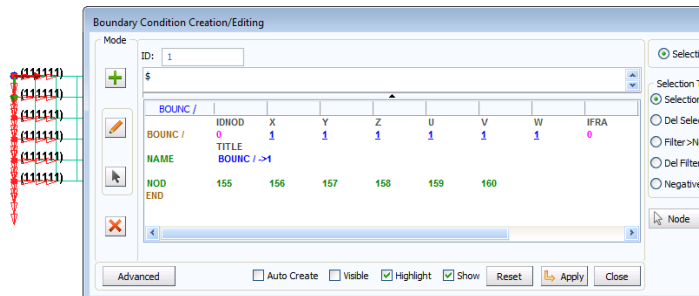
Boundary conditions (fixing the built in end to the wall and other constraints)

Click on the tabs **Crash > Loads > Displacements BC** to get the panel to assign nodes and boundary conditions. Use the left mouse key to drag a box over all the nodes at the left end (minus x direction) and click the **green arrow** to assign these. The selected nodes will now appear in the nodal list (see below; note your numbers could be different depending on initial meshing).

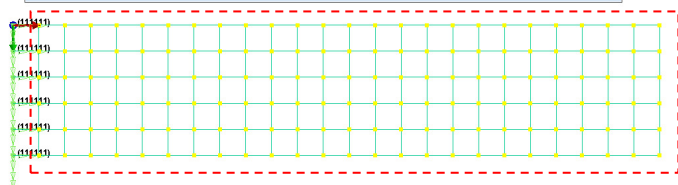
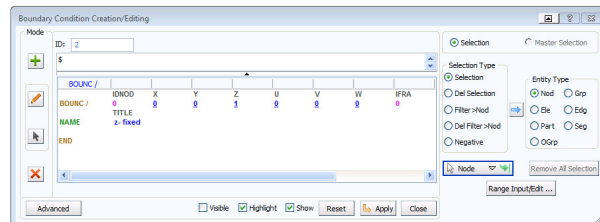


Now activate the required boundary conditions for these nodes:

- x,y,z are translational constraints
 - U,V,W are rotational constraints.
- Fix these nodes; in each case 1 means fixed, 0 means free. These are activated by clicking on each condition in the panel. Choose an appropriate title for the conditions; e.g. "fixed end". Save the selection with **Apply** and **Close**.



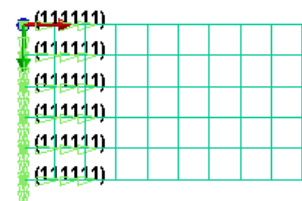
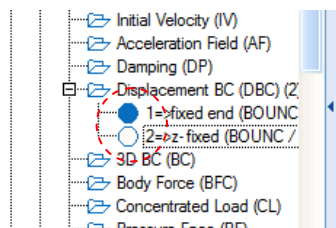
Further boundary conditions: The problem will use 3D shell elements, but we would like to treat this as a pseudo 2D problem with only deformation allowed in the x-y plane. Therefore constrain the other 'free' nodes in the z direction by repeating the above operations for the new selected nodes. Save with **Apply** and **Close**.



Visualisation (on/off) of entities

In the explorer panel (left tree directory) it is possible to edit entities and to switch on/off their visualisation.

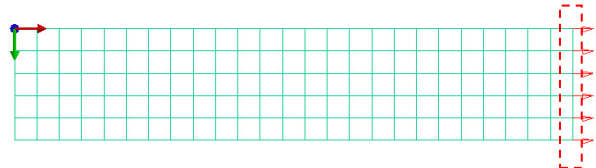
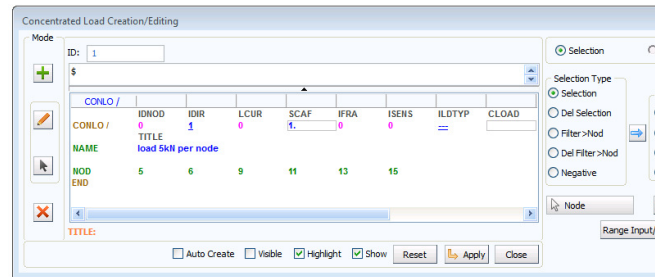
Click on the solid blue circles to switch off visualisation of the new boundary conditions.



Loading conditions (at the opposite free end)

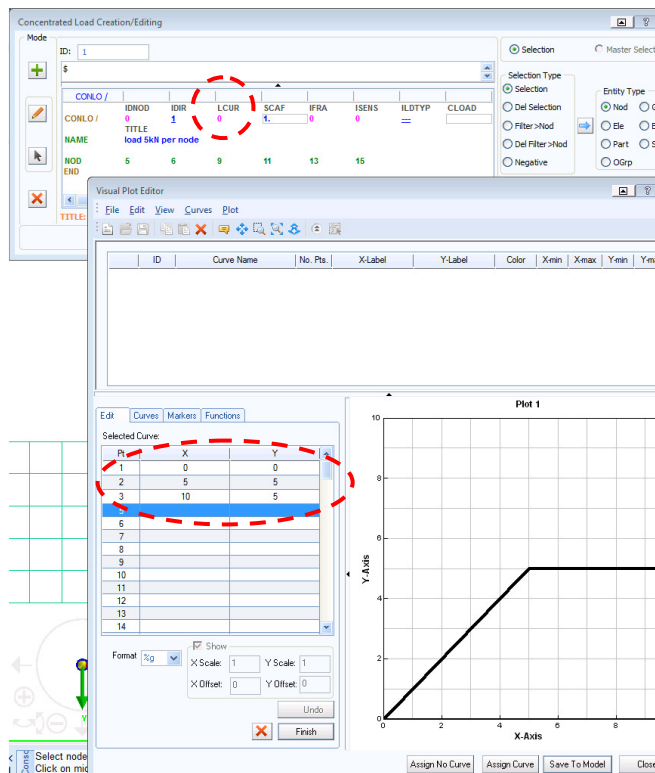
The aim now is to apply negative (y) loading at the free end.

Click on the tabs **Crash > Loads > Concentrated Loads** to get the panel to assign nodal loads. Use the left mouse key to drag a box over all the nodes at the right end (plus x direction of cantilever) and click the **Green Arrow** to assign these. The selected nodes will now appear in the nodal list. Give the nodal loads a suitable title; e.g "load 5kN per node" so it can be easily identified in the PAM-CRASH dataset file.

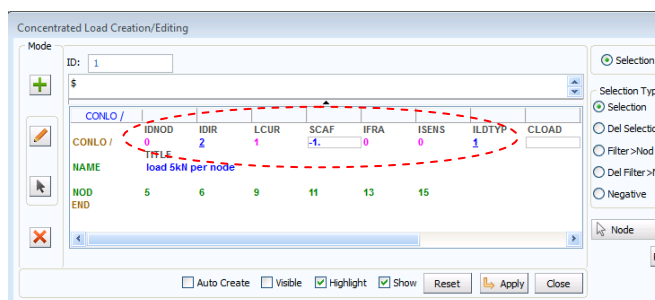


We now assign the loads to these nodes and a load direction. This is done using a load curve (which can have a constant value or a specific time history curve).

1. Click on **LCUR** and then **New** to open a load curve panel. Click on **File > New**.
2. In the new panel define the load curve function. The x-axis is time and y-axis is applied load. Assume the load starts at zero and increases to a constant 5kN at 5 msec, it is held at this value to the end of the simulation (=10 msec to be defined later).
3. Click **Assign Curve** and **Close**



4. Some last points:
 - The load so far defined is positive and we would like this to be in the negative y-direction. A convenient way is to set the load scale factor **SCAF = -1**. Also, set the load direction **IDR=2** (= y direction).
 - Make sure the new load curve (LCUR=1) is assigned to this set of nodes.
 - Use **ILDTYP = 1** option so the load is applied to each node (i.e. 5kN at each of the 6 nodes giving 30kN total applied load).
 - Note for a constant load the **CLOAD** option could be used.



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

Assigning the Material and Part data

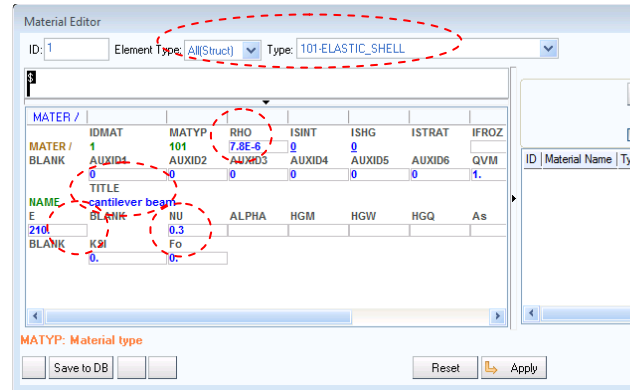
This is done using two entities which are linked together; namely:

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

1. Material data

Select **Crash > Material Editor** and then select the type of Material Model; in this case use a **101 – ELASTIC_SHELL**. Finally define the material data Steel ($E=210 \text{ kN/mm}^2$, $\nu=0.3$ and density $7.8 \cdot 10^{-6} \text{ kg/mm}^3$). Save with **Apply** and **Close**.

Remark: Make sure the units are consistent e.g. kN, mm, msec and kg; the analysis results will be in the same units.

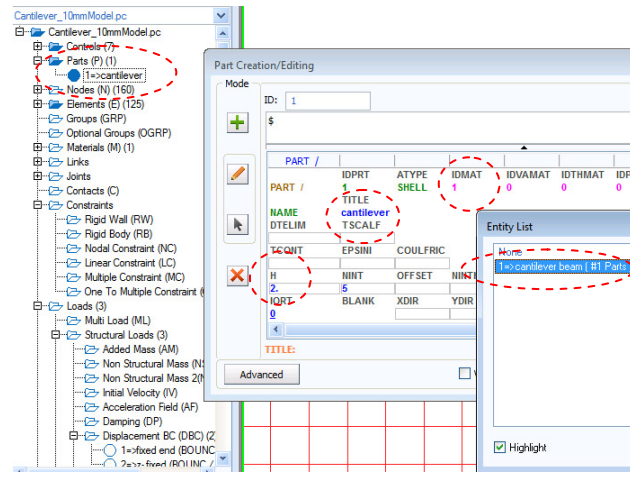


2. Part data

This is most easily done via the Explorer panel:

1. First click on **Parts** and the list of parts will open; select the required part. Press the left mouse key and activate the **Edit** option; the parts panel will open.
2. Set the shell thickness to 2mm. Click just below **H** and set this to 2 (=2mm).
3. Finally, the material is linked to this part. Click on **IDMAT** and the materials panel will open. Select the required material and click **OK**.
4. Give the part a sensible title (e.g. cantilever).

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

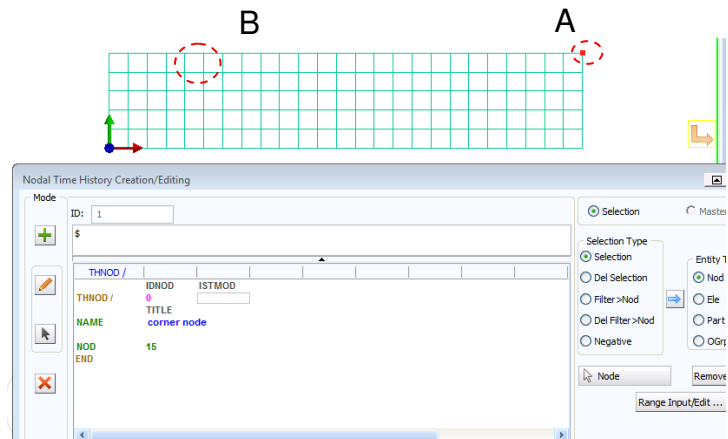


Output results

For later studies it will be useful to have details of time history information for the corner node (A) and an element (e.g. B 50mm from the wall).

For the node click **Crash > Output > Nodal TH** and select the corner node (A); click on it with the left mouse key. Then click the **Green Arrow**; give a suitable title and finish with **Apply** and **Close**.

For the element repeat the process using **Crash > Output > Element TH** and selecting the required element (B).



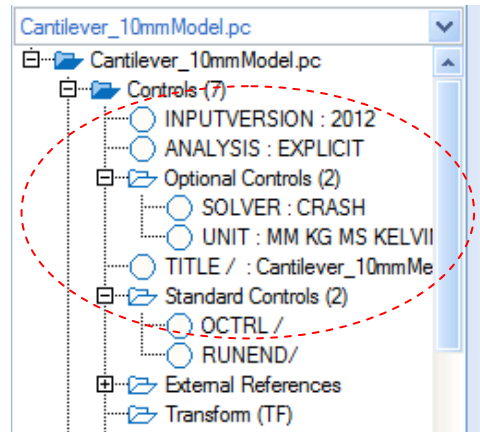
Tutorials 1,2: End loaded cantilever and shell element studies

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), specific output information and output intervals for the results.

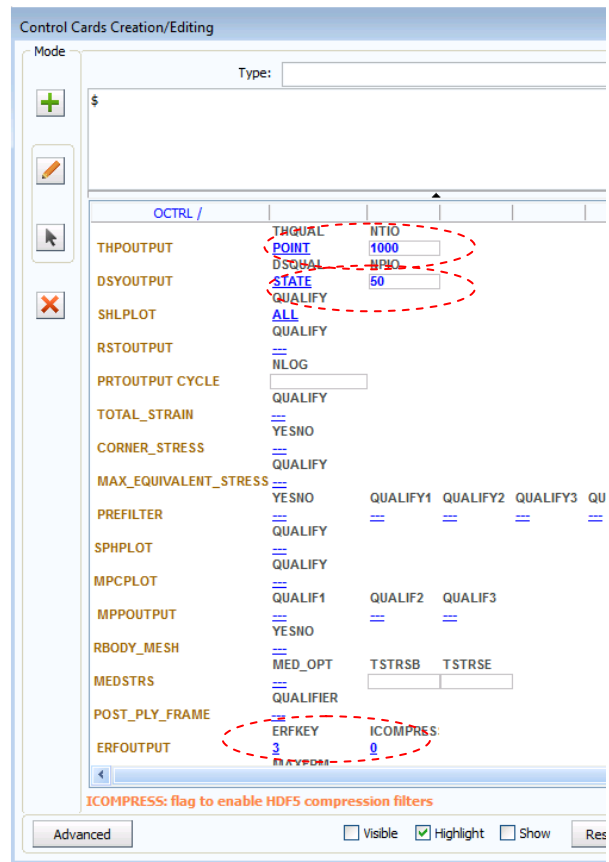
In the Explorer panel under controls the following controls can be opened and edited (click on the control and open with the right mouse key to edit):

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. Cantilever_10mmMesh); 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=10.0) 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 50 to give fifty 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**.

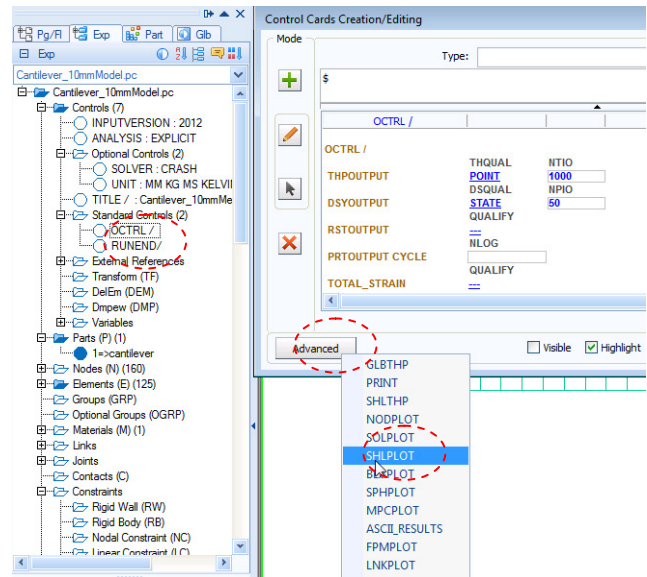


Tutorials 1,2: End loaded cantilever and shell element studies

Finally, special output options

By default only limited node and element time history information is stored in the ERF results file; for example, nodal displacements and velocities, and element local stresses.

A wide range of additional information is available, but this must be specified. For example in the Explorer panel click on **OCTRL** then the right mouse key and **Edit**. Click the tab **Advanced** followed by **SHLTHP > ALL > OK** to output all available shell contour variables to the result files. Finish with **Apply and Close**.



Saving the dataset (as a .pc file)

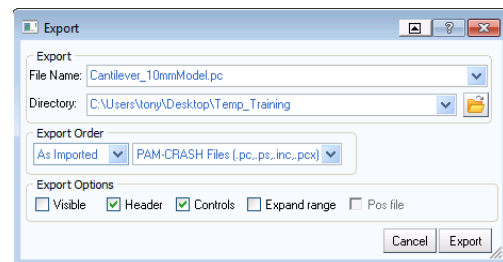
Click on **File > Export** and save the dataset in the required directory with a suitable name;

E.g. Overwrite the old **Cantilever_10mmModel.pc** file.

Be careful to make sure the export directory location is correct.

Remark 1: Files save in this way have the PAM-CRASH input file ASCII format and are readable. The **File > Save** option will save the model in a vcb internal binary format (it cannot be read by PAM-CRASH).

Remark 2: Some other dataset formats (e.g. DYNA3D and NASTRAN) are also possible to be saved: See the **Data type** options.



The PAM-CRASH (filename.pc) data file

The PAM-CRASH dataset (TOP)

Use a text editor to 'inspect' the input file:

- The top of the file should similar to this. →
- Work slowly through the dataset and try to identify the parameters you have set during the exercise.
- Do not worry if some nodes, etc., are slightly different.

```

INPUTVERSION 2012
ANALYSIS EXPLICIT
SOLVER CRASH
UNIT MM KG MS KELVIN
TITLE / Cantilever_10mmMesh.pc
OCTRL /
THPOUTPUT POINT 1000
DSYOUTPUT STATE 50
SHLPLOT ALL
END_OCTRL
RUNEND/
TIME 10.
END_RUNEND
$# IDNOD X Y
NODE / 1 0. 0.
NODE / 2 250. 0.
NODE / 3 250. 50.
NODE / 4 0. 50.
NODE 5 250. 0.
NODE 6 250. 10.
    
```

The PAM-CRASH dataset (BOTTOM)

- The end of the dataset should look similar to this: →
- Note the naming scheme is fairly logical, for example:
 - FUNCT – is a function curve (in this case the load curve)
 - CONCL – is the concentrated loads data and the list of nodes to which it applies
 - BOUNC – are the boundary conditions and applied nodes
 - THNOD – are the time history nodes which will be output to the results files for x-y plotting

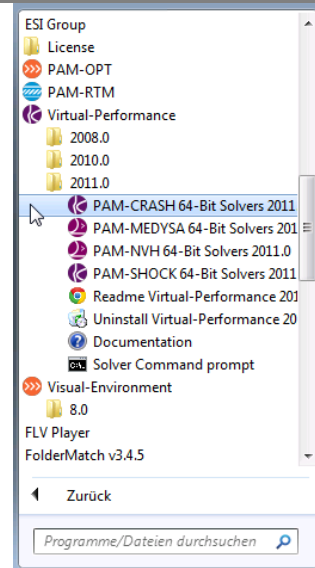
```

NAME cantilever beam
$# E BLANK NU ALPHA HGM
    210. 0.3
$#
$#
$# IDNOD ISTMOD
THNOD / 0
$#
NAME corner node
    NOD 15
    END
$# IDEL
THELE / 0
$#
NAME element 50mm from wall
    ELE 105
    END
**END OF DATA
    
```

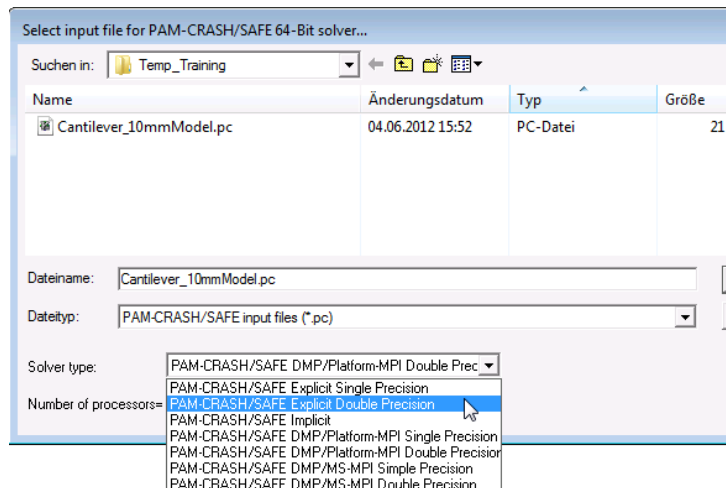
Part 2: Explicit analysis of the cantilever beam

Running a 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 a 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'.

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
    
```

- 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.

	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

Evaluating the results (Visual Viewer)

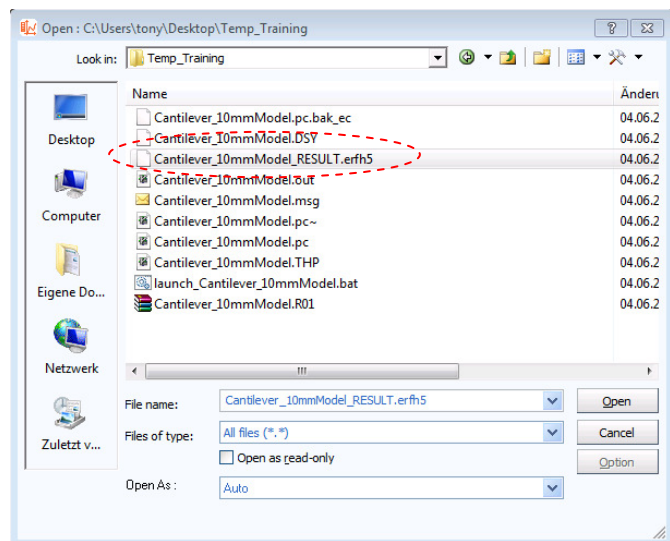
First, we shall look at selected results of the deformed structure and some information that are available for contour plots of the complete structure; these are stored in the .ERF file (extension .erfh5) at selected time intervals (states). Following this we shall look at typical x-y plot information which is stored in the same file. The time output intervals for structure deformations and contours are different to x-y plot information and were specified in the **Visual Crash PAM** session.

- Variable THPOUTPUT in the PAM-CRASH dataset controls frequency of time history plot information at selected nodes, elements, etc.
- Variable DSYOUTPUT in the PAM-CRASH dataset controls frequency deformed states and contour information for the full structure.

Mesh plot results

Start the **Visual-Viewer** program (from the same location as Visual Crash PAM); you will see that the layout of the program and most options to visualise and manipulate the part are the same as those used in **Visual-Mesh** and **Visual-Crash PAM**.

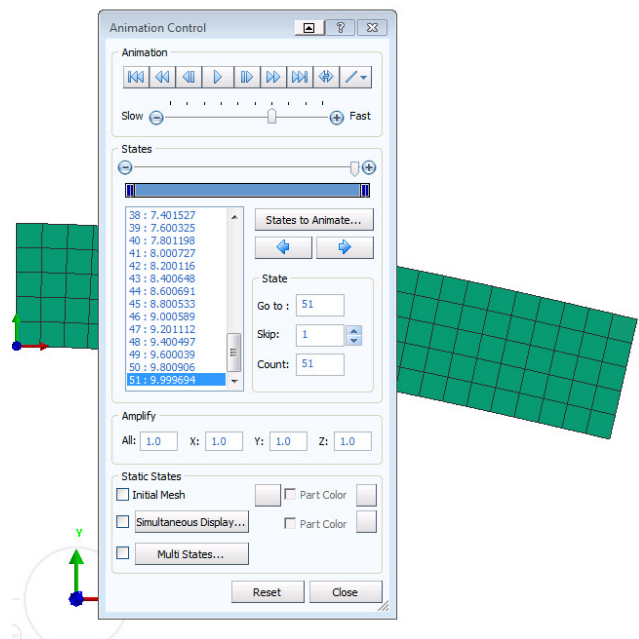
1. Open the results file using **Open File** and select the .ERF file (i.e. Cantilever_10mmModel_RESULT.erfh5). Note the new file extension is _RESULT.erfh5.



2. Position the structure; for example center and view in the x-y plane.
3. Click **Results > Animation Control** and you will be able to visualise the model and use the adjacent panel to examine deformations, either at a certain time, or as a continuous animation.

Use the **Speed Control** option to change the viewing speed in the animation option.

Additional tabs are available for overlaying the **Initial Mesh** or **Simultaneous Display** for viewing selected states simultaneously. Click **Close** to exit this option.

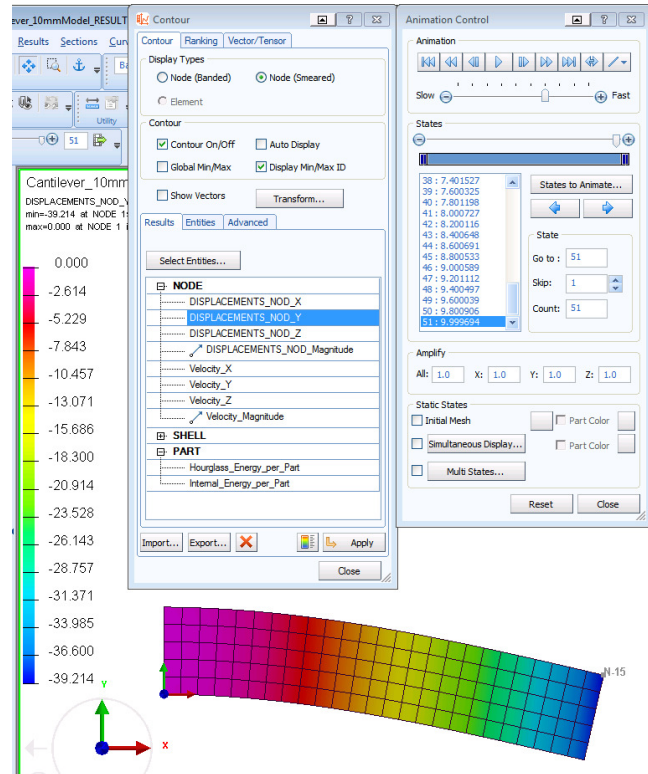


Tutorials 1,2: End loaded cantilever and shell element studies

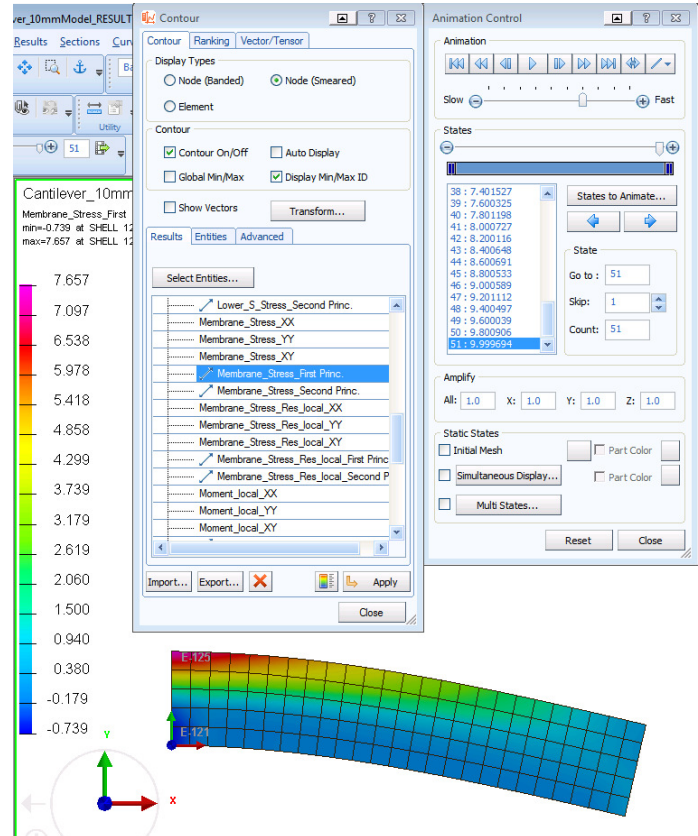
- Click **Results > Contour** and under **Select Entities** activate **Displacements_Nod_Y** and **Apply**. The contour distribution of nodal displacement (component Y) is shown with information on max/min values.

You will see various display options are available and the possibility to change contour limits if required.

This option, together with the previous animation control panel, allow different contours for both nodal and element variables to be visualised at certain time states, or as an animation.



- In the **Contour** panel the entity type **SHELL** can be selected and, for example, the variable **Membrane Stress First Princ.** selected to plot the maximum principle stress distribution for the structure. The adjacent view is for the last state (≈ 10 msec).



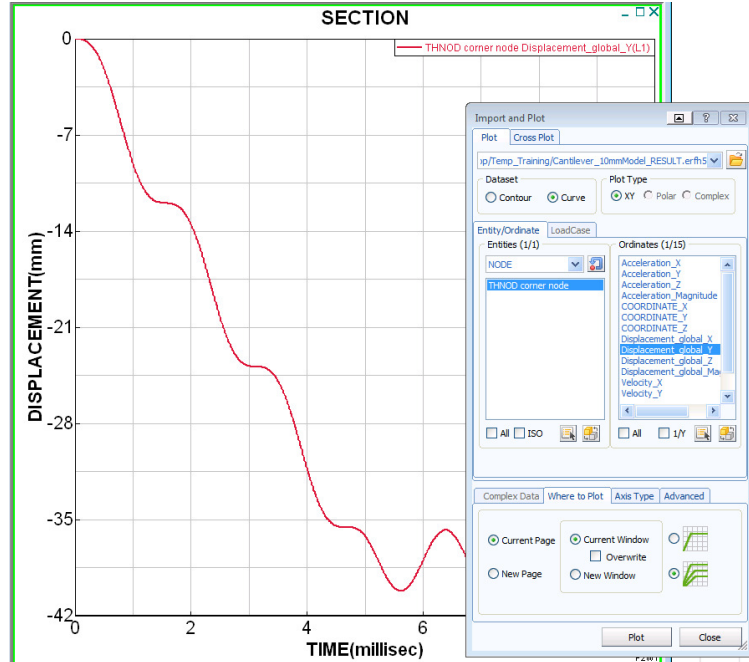
Tutorials 1,2: End loaded cantilever and shell element studies

Time history results

From the original Visual Crash PAM session two specific points of information were stored for x-y type plots; namely, the element located on the upper surface at 50 mm from the wall and the top corner node at the loaded end.

To collect this information click **File > Import and Plot** which will open the adjacent panel. Under Entities click **NODE** and a list of stored nodal time histories will appear. In this case there is only the '**corner node**'. Click this and also the information to be plotted **Displacement_global_Y** and **PLOT**.

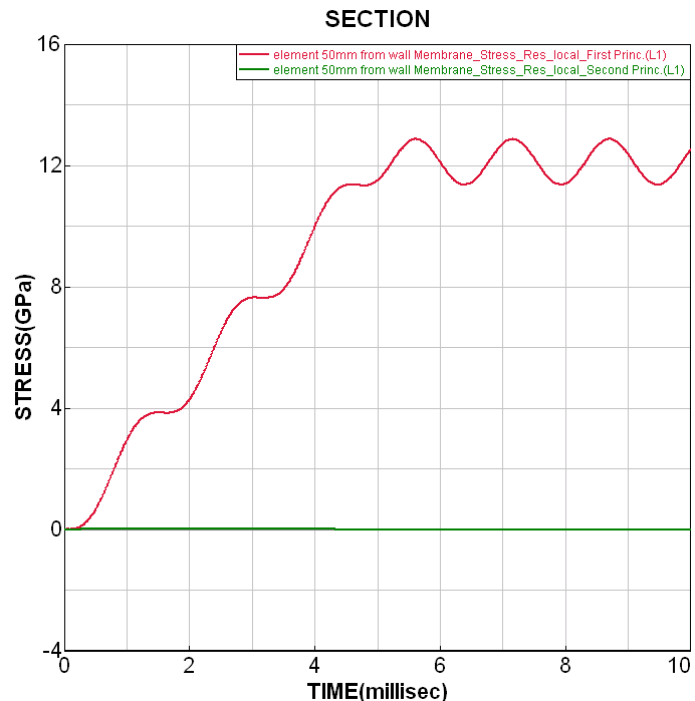
The red (oscillating) curve plotted is the displacement time history (component Y) for the upper corner node. The oscillations are typical and a consequence of the undamped dynamic explicit FE solution method.



Using similar steps time history information for stress (and other) variables in selected element can be plotted.

In the **Import and Plot** panel activate the **Entity** Type **SHELL**:
Membrane_Stress_Resultant_Local_First Princ. and **PLOT**,
then
Membrane_Stress_Resultant_Local_Second Princ. and **PLOT**.

The two curves (one is nearly zero) are plotted for these principle stresses in the selected shell element.



Part 3: Exercises in explicit analysis

3.1 Quasi-static explicit analysis and nodal damping

The oscillations of the structure observed in the previous analysis are a consequence of the explicit integration method and the dynamic way loading is imposed over a short duration. For this simple beam structure these oscillations can be removed using nodal damping; a full description is given in the PAM-CRASH user and theory manuals.

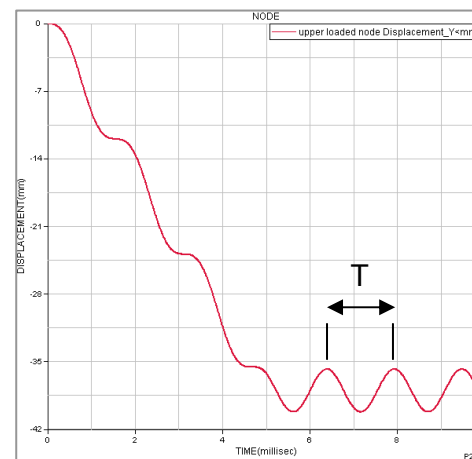
Nodal damping should only be imposed once the structure is fully deformed under constant loading and in a state of dynamic equilibrium; in this case after 5msec. It should not be imposed during the loading phase as this will damp the loading deformations and in the case of elasto-plastic analysis may change the plastic distribution. The formula to imposed critical nodal damping is given in the PAM-CRASH manuals as,

$$q_{crit} = 2\omega = \frac{4\pi}{T},$$

where T is the period of oscillation of the frequency to be critically damped; in this case T= 1.5 msec. Critical damping will damp the selected frequency over one cycle.

Damping is applied from 5 msec onward using two cases:

1. Critical damping $q_{crit} = 4\pi / 1.5 = 8.38$
2. 20% of critical damping $q = 0.2 * 8.38 = 1.68$

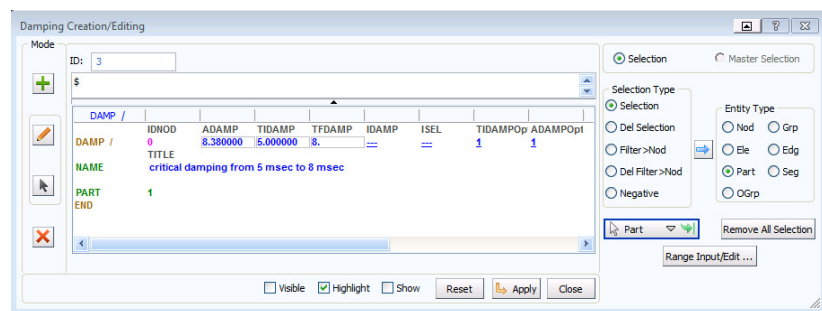


To apply nodal damping open the adjacent panel using

CRASH > Loads >

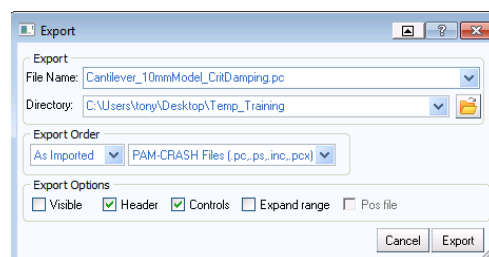
Damping. In the selection type click **Part** and then click on the structure in the main window and **Update Selection** to select the full structure for damping. The

critical damping value is entered, together with a start and end time for damping of 5msec and 8msec respectively. Chose a suitable title and click **Apply** and **Close** to finish.



The modified dataset is saved using **File > Export** in directory with a new file name, e.g.

Cantilever_10mmModel_CritDamping.pc



Tutorials 1,2: End loaded cantilever and shell element studies

Repeat the same operations using 20% of critical damping ($=1.68$), with all other data remaining the same. Export the dataset as **Cantilever_10mmModel_0.2CritDamping.pc**

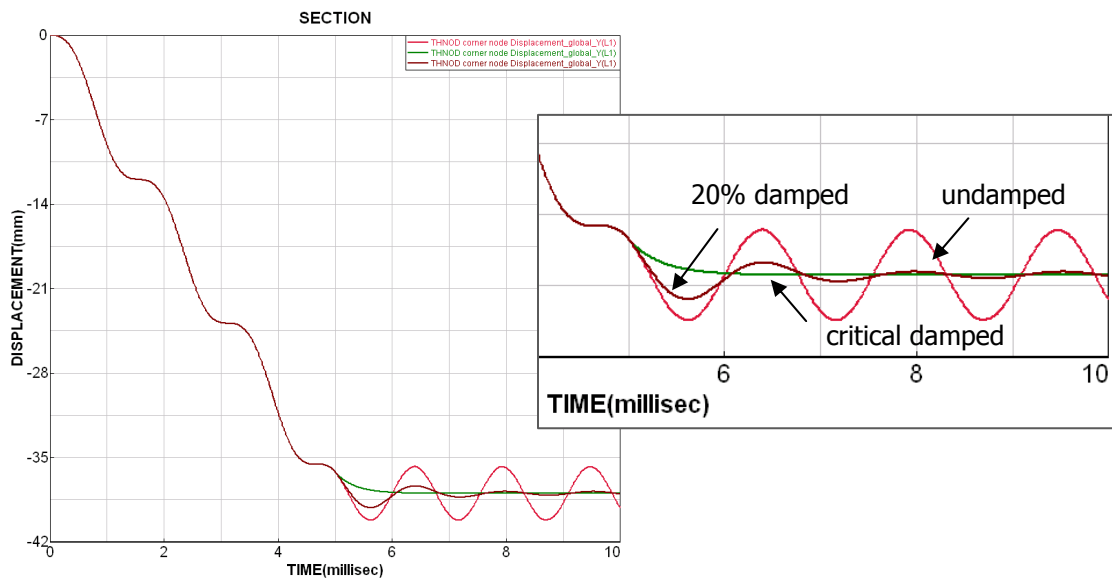
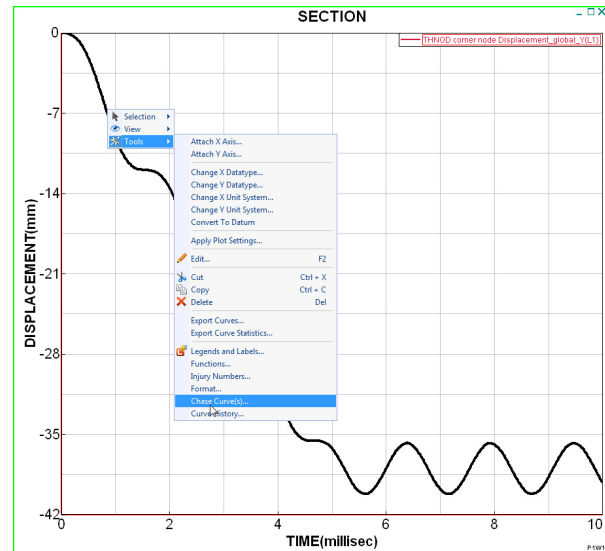
The two damping cases are run with PAM-CRASH to generate new .ERF results files:

Cantilever_10mmModel_CritDamping_RESULTS.erfh5

Cantilever_10mmModel_0.2CritDamping_RESULTS.erfh5

These results are now compared with the undamped solution for the corner node at the loaded end. Start Visual-Viewer and open the previous undamped results file (**Cantilever_10mmModel_RESULTS.erfh5**); using the previous procedure plot the vertical displacement time history (**Displacement_global_Y**).

A convenient method to superimpose the damped results is to use the **Chase Curve(s)** option. Click on the current time history curve and it will change colour to black. With the right mouse key new options will open; click **Tools** and then **Chase Curve(s)**. Using the windows folder options the damped results files (do one after the other) can be loaded and automatically plotted. Note there are options that appear to allow curve settings to be changed, for example the colour.



The displacement time history curves for the free end corner node shows that critical nodal damping has rapidly damped the structure within one oscillation, and that 20% has nearly managed this in the selected time window of 5 msec to 8 msec. In both case the final quasi-static deflection of the beam is approximately -38mm.

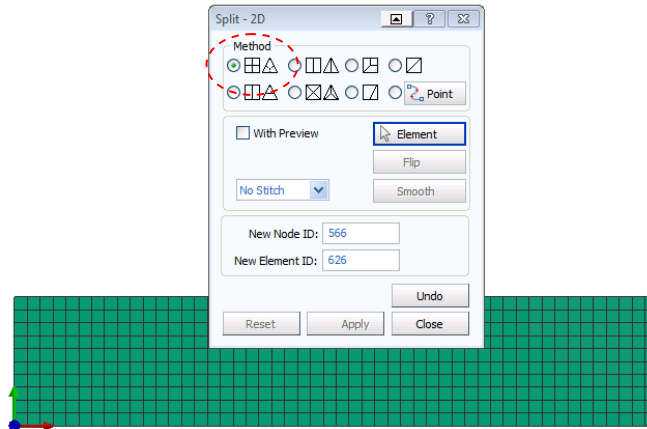
Remark: Generally, critical damping is not recommended as this can lead to stability problems; instead 10% to 20% of critical damping over 2-3 oscillation is preferable. In this study the damping was deliberately switched off after 8 msec to see if further oscillations occur, or if steady state equilibrium has been reached; it more-or-less has.

3.2 Mesh refinement and further analyses

In this section the 10mm FE mesh model is refined to give equivalent models having 5mm and 2.5mm elements sizes, and a model using triangular elements. These new models will be used to study several effects including hourglassing and solution convergence with refined meshes.

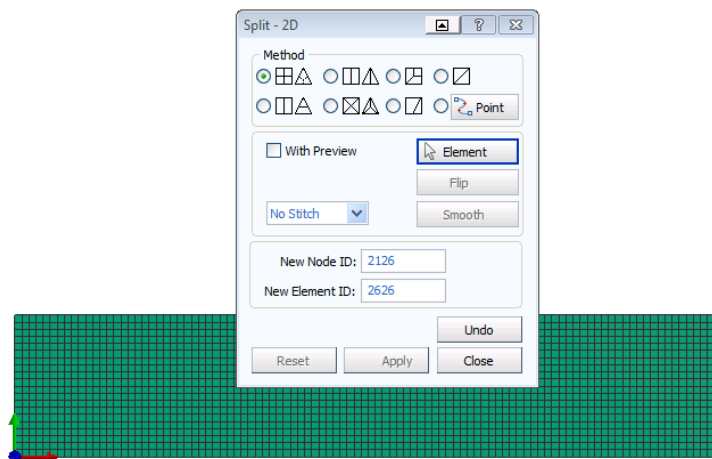
The following steps briefly outline the steps to prepare the new models.

1. Start **Visual Crash PAM** and read in the 10mm meshed model (the one without nodal damping) that we have been using.
2. This mesh is easily refined using the option **2D > Split** and the splitting option shown below. All elements in the model must be selected. The new mesh has four times more elements with a new side length of 5mm. Finish with **Close**.



3. Check the model: The material data and control data are good; but the boundary conditions and loading are not strictly correct (have a look) since new generated nodes have no boundary conditions or loads. However, the model is valid and we can it for now. Note that the total loading is still the same as the 10mm meshed model, although poorly distributed.
4. Export the dataset with a new name (e.g. **Cantilever_5mmModel.pc**) and run it using **PAM-CRASH**.

Repeat the above procedure and refine the 5mm mesh model one level further (element side length = 2.5mm); again use a new file name when exporting (e.g. **Cantilever_2.5mmModel.pc**).

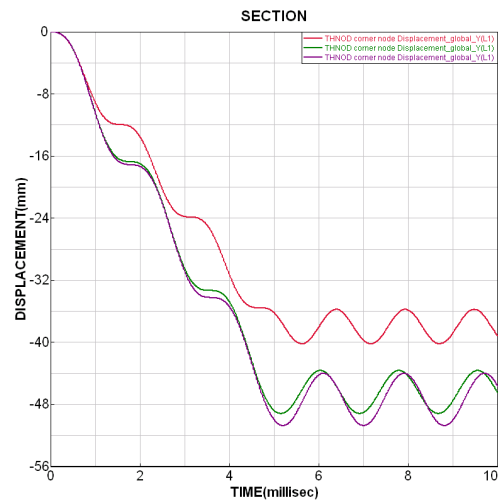


Tutorials 1,2: End loaded cantilever and shell element studies

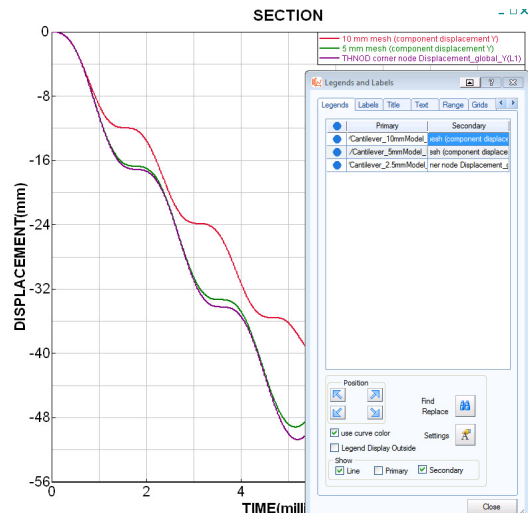
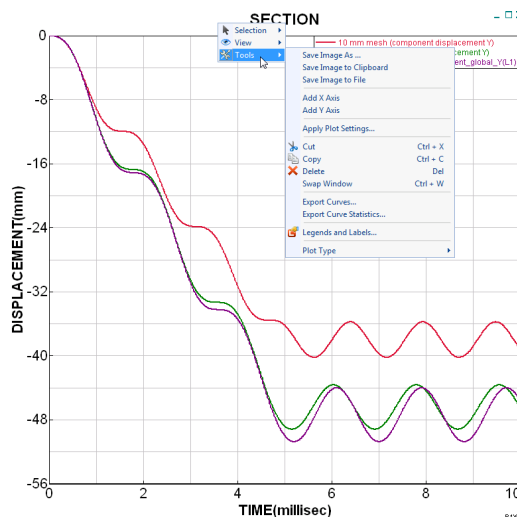
The two refined analysis models are run with **PAM-CRASH** and are now compared with the previously analysed 10mm meshed model. Only the undamped results are used in this comparison.

- Start **Visual Viewer**, open the .erfh5 results file for the undamped 10mm meshed model. Then plot the nodal displacement (**Component Y**) for the loaded end corner node.
- Use the Chase Curves(s) method describe in the previous section to superimpose the same displacement time history for the 5mm and 2.5mm meshed models; you should get the adjacent plot.

The 10mm element gives the lowest 'average' end displacement ($\approx -38\text{mm}$), whereas the 5 mm and 2.5 mm element mesh models both show greater average end displacements ($\approx -47\text{mm}$).

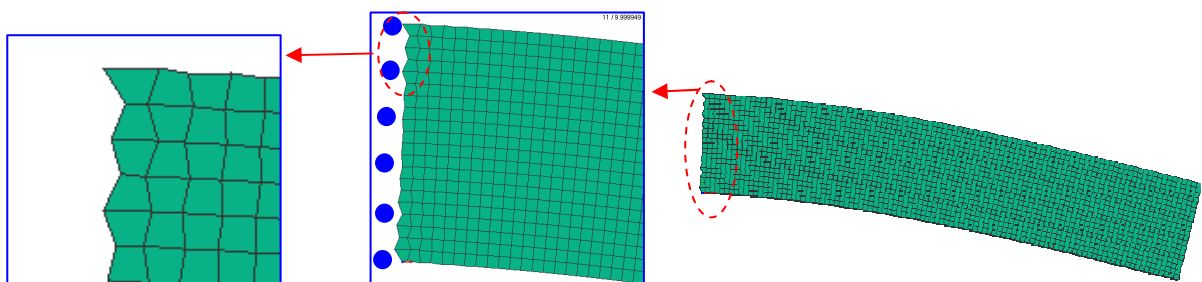


Titles and graph legends can be modified. Anywhere on the graph view click with the right mouse key to open Tools and then Legends and Labels. From the new panel new labels, font sizes and colours can be defined.



A close inspection of the deformed model at the restrained end, below, shows poor deformations and local hourglassing; this has led to (incorrect) greater end point deflection.

The boundary condition nodes come from the original 10mm meshed model, so only every second node in the 5mm mesh and every fourth node in the 2.5mm meshed model is restrained. This is bad practice and all nodes should be restrained! However, it is an interesting result and the opportunity is taken in the next section to discuss hourglassing and methods to treat it.



3.3 Element types and hourglass control

Hourglassing is a problem of under-integration techniques used in the numerical integration of element stiffness (or other quantities) and is especially common in 'one point' integration elements used in explicit FE codes. It is a phenomena that can be easily excited in regular meshed structures, or poor loading. A full description of hourglass theory and its control is given in the PAM-CRASH theory manual.

Essentially, hourglass modes are certain element deformation modes that erroneously predict zero deformation energy at the element integration point(s). For one point under integrated elements this point is at the centre of the element. Three methods could be used here to control, or eliminate hourglassing:

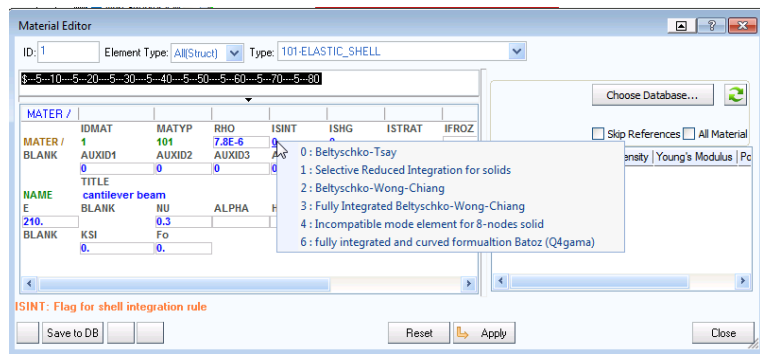
1. Fully constrain all nodes at the boundary; this would be the best modelling approach.
2. Use a different element type that does not have hourglass modes, for example 'fully' or 'reduced' integrated elements, or triangular elements.
3. Apply increased hourglass control parameters (this rarely is very effective).

For this study to control hourglassing the 5mm mesh model with the poor end restraints is used. Also, for easier comparison of solutions use the previous parameters for 20% critical damping for a quasi-static solution. Open the file **Cantilever_5mmModel.pc**, add damping and export as,

Cantilever_5mmModel_0.2CritDamping.pc

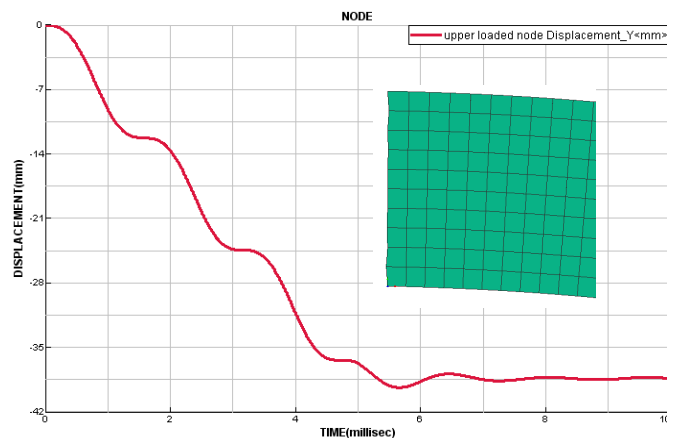
Method 1: Using the Hughes-Tezduyar (reduced integration) element

The Belytschko-Tsay shell element is the default PAM-CRASH shell element and was used in all previous studies. It is CPU efficient, but is under-integrated and can suffer hourglass problems. Instead the **Hughes-Tezduyar element**, which uses a reduced (2*2) integration scheme, is analysed. This is activated via the Material panel in **Visual-Crash PAM**



(Shell element type 1 under the ISINT parameter). Make this change to **Cantilever_5mmModel_0.2CritDamping.pc**, export the file under a new suitable name and run with **PAM-CRASH**.

Analysis results show that hourglass modes are eliminated and deflections (= -38.5 mm) compares well with the previous 10mm mesh case using the Belytschko-Tsay element.

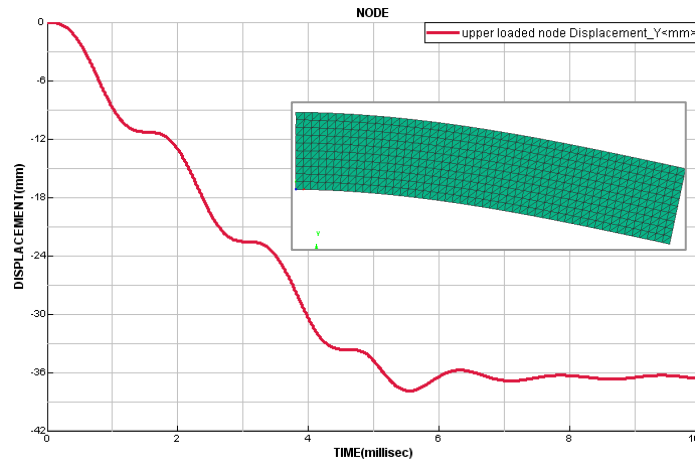


Tutorials 1,2: End loaded cantilever and shell element studies

Method 2: Using triangle elements

The Belytschko-Tsay element is used (set parameter ISINT in the material cards) and the 5mm mesh model **Cantilever_5mmModel_0.2CritDamping.pc** is modified to have triangle elements. This is done using the Split option in **Visual Crash PAM**. Export the file under a new suitable name and run with **PAM-CRASH**.

The maximum deflection is about -36.5mm as shown below.



Note in this case the poor boundary conditions have been used but there are no signs of hourglassing. Triangle elements do not have hourglass modes. Also, it can be seen that the end deflection is slightly less ($\approx 5\%$) than the quadrilateral element mesh result; this is due to this element formulation being slightly stiffer.

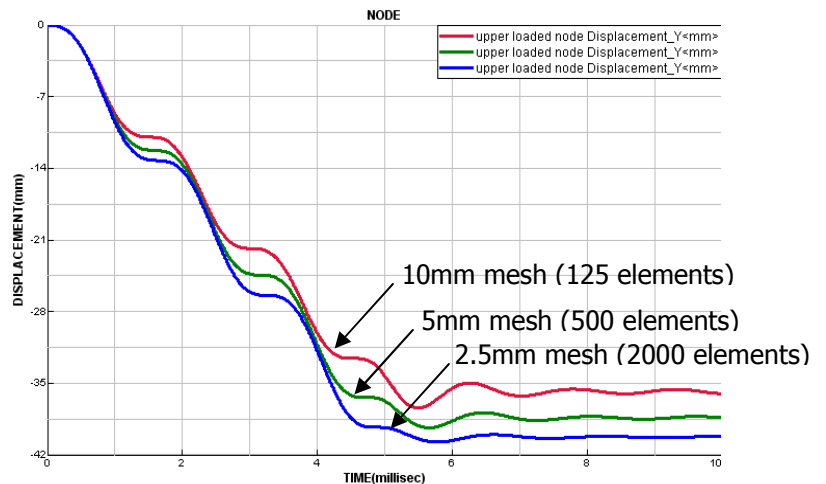
Remark: Hourglassing is actually very rare in analyses and should not occur if the standard PAM-CRASH hourglass controls are used in under-integrated elements.

3.4 Element convergence (explicit analysis with the Hughes-Tezduyar element)

It is expected that a 'well behaved' finite element should show a convergence trend. That is, analyses using increasingly refined meshes should tend toward a converged solution.

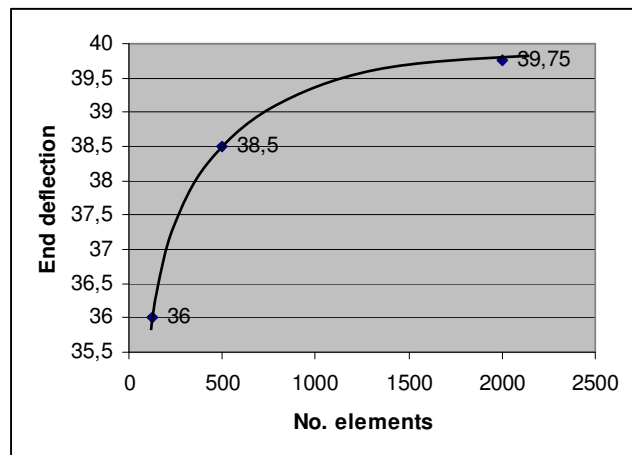
The following is a convergence exercise to try. Use the Hughes-Tezduyar shell element (to eliminate hourglassing) with 20% critical damping for a quasi-static solution. Compare maximum deflection at the corner node for the 2.5mm, 5mm and 10mm meshes.

Results for deflection:



It can be concluded that this element has a good convergence trend and the final deflection is about 40mm (for this element type).

A similar study for triangle elements should also show convergence with refined meshes, but with slightly stiffer behaviour.



Part 4: Implicit analysis of the cantilever beam

The following briefly shows the simple steps needed to convert a PAM-CRASH Explicit dataset to an Implicit dataset. The changes are to the control cards only and are:

1. To specify that an **Implicit** analysis will be undertaken.
2. To specify that a **Static** analysis will be performed. In this case geometrically **Linear** analyses will be performed; in the next section this is changed to geometrically **Non-Linear**.
3. To specify the element formulation to be used; in this case **Element type 6**.

There are two ways to do this, either:

Read the explicit dataset into **Visual Crash PAM**, make the necessary modifications outlined below, and export the modified dataset to a suitable new name, for example for each case:

Read in **Cantilever_10mmModel.pc** and export to **Cantilever_10mmModel_Implicit.pc**

Read in **Cantilever_5mmModel.pc** and export to **Cantilever_5mmModel_Implicit.pc**

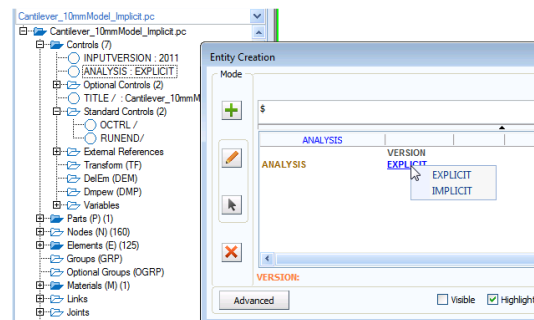
Read in **Cantilever_2.5mmModel.pc** and export to **Cantilever_2.5mmModel_Implicit.pc**

Or

Make a copy of the datasets in advance and work with the copy. This is the safest way to do it as there is no risk to loose, or accidentally modify the original.

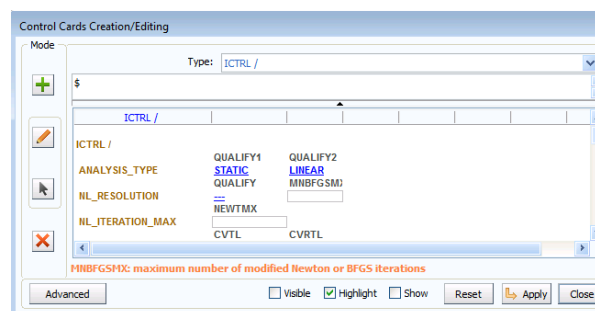
1. The implicit control card

In the explorer panel under **Controls** the type of analysis can be changed from EXPLICIT to IMPLICIT. Then click **Apply** and **Close**.



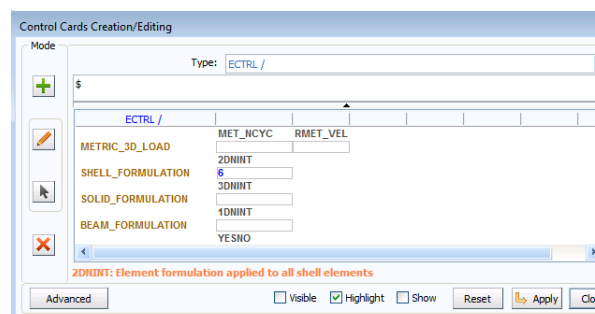
2. The type of implicit analysis

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**. Then click **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 SHELL FORMULATION activate type **6**. Then click **Apply** and **Close**.



4.1 Static linear implicit results

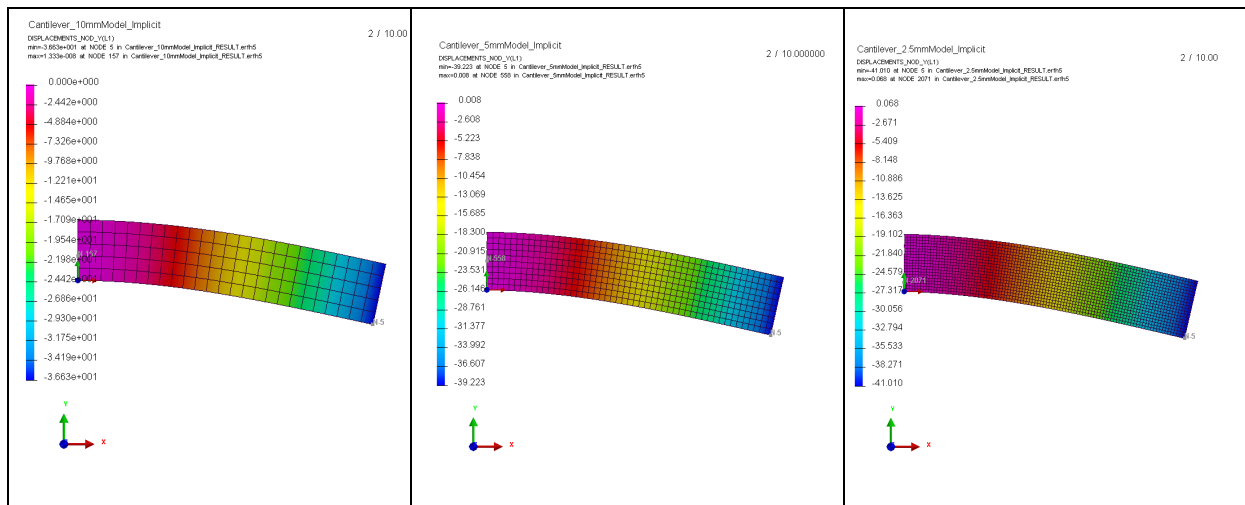
Use **PAM-CRASH Implicit** to run the three implicit cantilever beam problems; namely,

Cantilever_10mmModel_Implicit.pc

Cantilever_5mmModel_Implicit.pc

Cantilever_2.5mmModel_Implicit.pc

Results from the analysis are processed in exactly the same way as the previous explicit analyses using **Visual Viewer**; all results are stored in the respective .ERF files. The following plots show contours of deformations for the three models.



The three implicit analyses show similar results and trends to the explicit analyses with maximum y-deflections being:

for the 10mm mesh = 36.63 mm

for the 5mm mesh = 39.22 mm

for the 2.5mm mesh = 41.01 mm

Remark 1: Generally, the element shows a good convergence trend; that is, the more elements used the more accurate is the result and the results tend to a converged value.

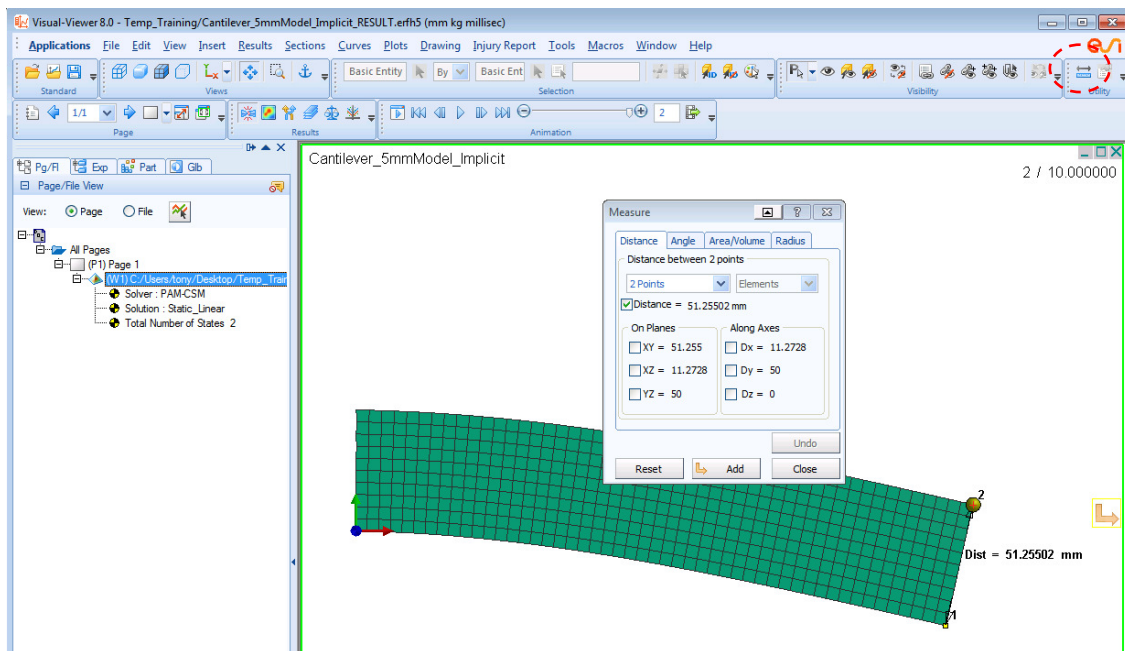
Remark 2 : There is slightly greater displacement indicating this element type is less stiff than the previous element used in the explicit analyses.

Remark 3: For this case the linear implicit analysis is much faster to analyse than the explicit analysis. This is not always the case and implicit analysis can become CPU expensive if significant material and/or geometric nonlinearity occurs and is included in the analysis; see the next section.

4.2 Static geometric non-linear implicit analysis

Generally a static linear analysis will provide good results for most problems if the material is linear and displacements are small. In this case, however, displacements are large and a close inspection of deformations (look at the width of the beam along its length) suggests something might be wrong; indeed the beam gets wider!

Using **Visual Viewer**, open the 5mm mesh linear static analysis results. The deformed width of the beam at the loaded end can be measured using the 'measure' tab shown below. A new panel opens and then click on the two point to be measured. It is seen that the beam width increases from 50mm (undeformed) to 51.25mm which cannot be correct under this applied loading. A contour plot of nodal displacements (x-Component) also shows strange results that the beam does not shorten in the x-direction!



Remark: Linear static analysis is valid for most problems involving small displacements. In this case the derivation of the element stiffness matrix assumes a linear relationship between element displacements and strains and other simplifications are made regarding the interaction of element deformations and the material strains.

In effect these simplifications lead to a constant structure stiffness matrix that assumes a linear relationship between loads and displacements. This is not valid in this analysis as large deformations occur and a full geometric nonlinear analysis must be performed. In this case the structure stiffness matrix is a function of the displacements and a non linear iterative (e.g. Newton Raphson) solution is necessary.

A geometrically non-linear analysis can be activated in **PAM-CRASH Implicit** using the following changes; we shall do this arbitrarily for the 5mm mesh model.

Tutorials 1,2: End loaded cantilever and shell element studies

1. Copy the 5mm mesh case **Cantilever_5mmModel_Implicit.pc** to a new file; e.g. **Cantilever_5mmModel_Implicit_GNL.pc**.
2. Change the "Type of Analysis". In the Explorer window locate the existing ICTRL control card and change the Qualify2 from **LINEAR** to **GEOMETRIC NON LINEAR**, click **Apply** and **Close** to save the changes. Note it is still a static analysis problem.
3. Finally, nonlinear analyses are performed as a series of load increments, with iterations being made at each load increment to an acceptable level of convergence. Information of the load steps must be given. This is controlled by the time value defined in **RUNEND** (=10) and the time increment which is set under **Crash > Controls > Standard Controls** and in the window **Type** for **TCTRL**. Set parameter DT2USR=0.2. These two values mean that 50 load increments will be made, which is a reasonable number.
4. Note that the RUNEND parameter (=10) was the duration time used for all explicit analysis work. In the previous implicit analysis it had no meaning and was ignored.
5. Export the model to the file **Cantilever_5mmModel_Implicit_GNL.pc** and run the analysis with **PAM-CRASH Implicit**.

Some results for the analysis are briefly presented

First, it is useful to look at a typical step in the output file. The time is specified and this increases from 0.0 to the end time 10.0 in intervals of 0.2. The number of Newton Raphson iterations for convergence is given together with output on solution progress for each load increment.

```
TIME..... 1.0000E+01
TIME STEP..... 2.0000E-01
INCREMENT..... 50
NEWTON ITERATION..... 1
*****

REFERENCE FORCE : 5.1462E+00 (MODEL AVG. : 6.7866E+00 ON 65.05% ZONE)
LARGEST CURRENT RESIDUAL FORCE : 3.9702E-13 (NODE : 555 DOF 1)
CURRENT RELATIVE FORCE : 7.7148E-14 (THRESHOLD : 1.0000E-02) CONVERGED
AVG. MAX. FORCE : 7.3342E+01 (MODEL MAX FORCE : 9.6927E+01)
REFERENCE DISPLACEMENT : -3.4895E-06 (NODE : 15 DOF 2)
LARGEST CURRENT DISP. CORRECTION : -3.4895E-06 (NODE : 15 DOF 2)
CURRENT RELATIVE DISPLACEMENT : 1.0000E+00 (THRESHOLD : 1.0000E-02)
CONVERGENCE WAS ACHIEVED DUE TO LINEAR FORCE INCREMENT

REFERENCE MOMENT : 4.2228E-04 (MODEL AVG. : 5.5466E-04 ON 33.30% ZONE)
LARGEST CURRENT RESIDUAL MOMENT : -3.4694E-18 (NODE : 551 DOF 6)
CURRENT RELATIVE MOMENT : 8.2160E-15 (THRESHOLD : 1.0000E-02) CONVERGED
AVG. MAX. MOMENT : 1.3599E-02 (MODEL MAX MOMENT : 1.8109E-02)
REFERENCE ROTATION : -1.7653E-08 (NODE : 555 DOF 6)
LARGEST CURRENT ROT. CORRECTION : -1.7653E-08 (NODE : 555 DOF 6)
CURRENT RELATIVE ROTATION : 1.0000E+00 (THRESHOLD : 1.0000E-02)
CURRENT TOTAL RESIDUAL ENERGY : 3.5687E-20
CURRENT RHS RESIDUAL NORM : 3.1293E-12
CONVERGENCE WAS ACHIEVED DUE TO LINEAR MOMENT INCREMENT

NUMBER OF ITERATIONS IN INCREMENT : 1
```

The following contour plot shows nodal displacements for the cantilever beam. The geometric non-linear analysis has now led to a correct prediction of structural deformations. In this case overall deflection of the beam is similar to the linear results; however, the width is now correctly predicted to be 49.986mm (~ 50mm). The stress distribution and x-component of displacements will also be correct.

Tutorials 1,2: End loaded cantilever and shell element studies

Cantilever_5mmModel_Implicit_GNL

51 / 10.000000

DISPLACEMENTS_NOD_X(L1)
min=-8.675 at NODE 5 in Cantilever_5mmModel_Implicit_GNL_RESULT.ert5
max=3.503 at NODE 339 in Cantilever_5mmModel_Implicit_GNL_RESULT.ert5

