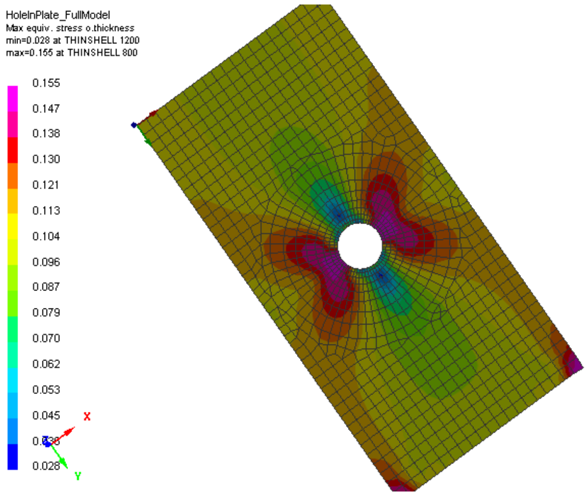
Tutorials 3 and 4

Elastic and Elasto-Plastic Loading of a Plate with a Central Hole



Problem description

Outline	The stress analysis a simple rectangular plate with a central hole is performed using Implicit and Explicit analysis. Different modelling methods, mesh refinements, elastic and elasto-plastic material laws are used.
Analysis type(s):	Implicit and Explicit, 2D analysis
Element type(s):	Shell
Materials law(s):	Elastic and Elasto-Plastic
Model options:	Linear and nonlinear analysis, full and symmetric models, mesh refinement
Key results:	Elastic and elasto-plastic stress distributions, deformations

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):** 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.

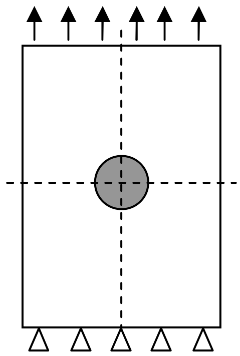
Problem data and description

Units: kN, mm, kg, ms

Description: Plate 200mm*100mm, thickness 2mm with a central hole of diameter 20mm.

Loading: Imposed total load = 20 kN (full model) with simple support at the base.

Material: Steel (E=210 kN/mm^2 (GPa), v=0.3 and density 7.8*10^{-6} kg/mm^3). For plasticity use a yield stress 190 MPa and a constant plastic modulus 0.2 GPa.



Supplied datasets

No datasets or meshes are needed for this problem; the meshes will be generated in the exercise.

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

For the <u>elastic</u> problem **HoleInPlate_Elastic.pc**

For the <u>plastic</u> problem **HoleInPlate_Plastic.pc**

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

Background information: Solution methods and geometry

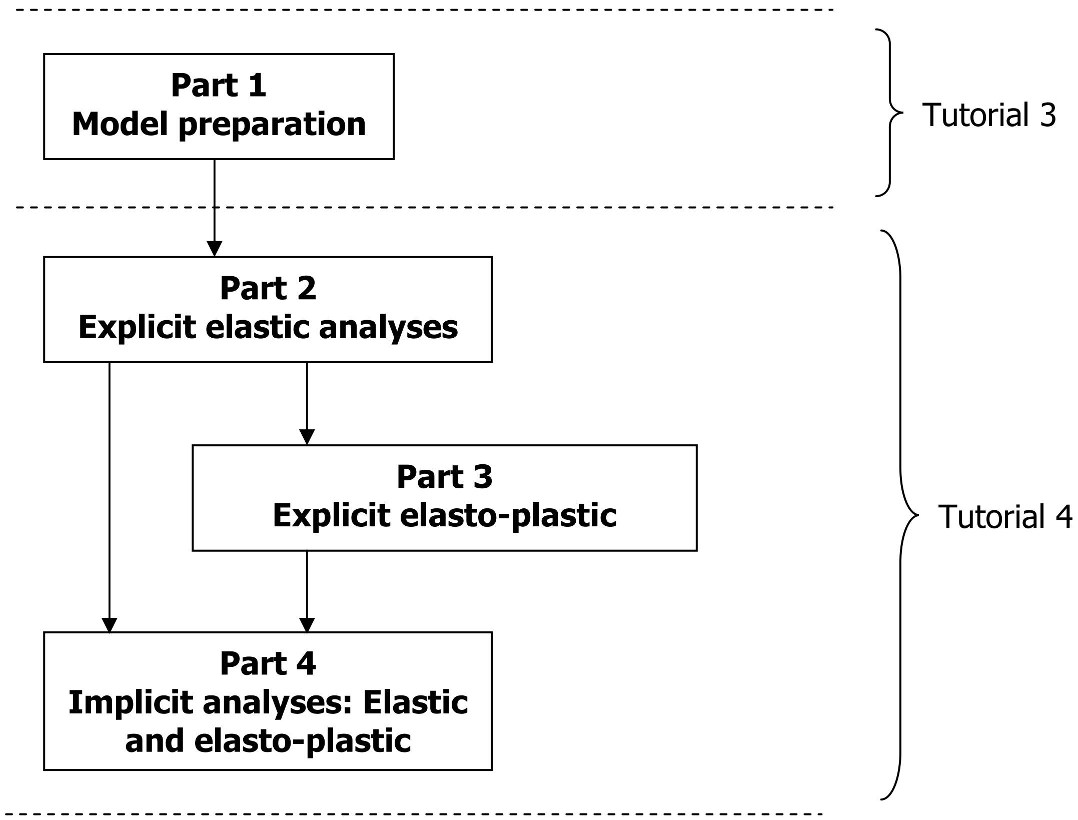
<u>Implicit versus explicit analysis:</u> The PAM-CRASH code is a general purpose three dimensional (3D) code for implicit and explicit FE analysis. This exercise performs implicit and explicit analyses of a simple two dimensional (2D) structure with applied force loading.

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. This exercise will include an implicit non-linear material analysis.

<u>Geometry:</u> 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 plate is analysed using shell elements with appropriate loading and boundary conditions to make it a valid 2D problem.

Contents

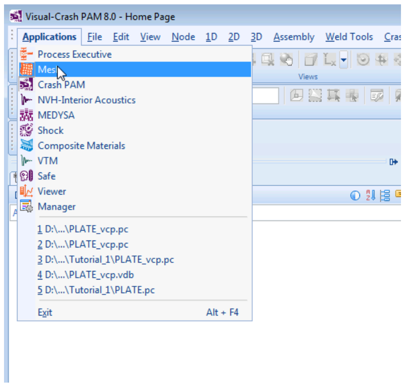

Part 1
Model preparation
Tutorial 3
Part 2
Explicit elastic analyses
Part 3
Explicit elasto-plastic
Part 4
Implicit analyses: Elastic and elasto-plastic
Tutorial 4

Part 1: Using VCP to construct the models

Preparing the mesh (Visual Mesh)

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

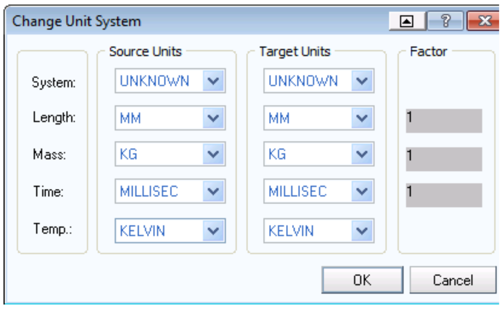
This will activate the meshing module (**Visual Mesh**) of the VCP program which is dedicated to generating finite element meshes.



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.

Meshing procedures and option for the plate with a central hole

The usually approach to mesh a plate with a hole would be to define key points in x,y,z coordinates to which lines and arcs (for the hole) are assigned. From this surfaces are defined and meshed. Usually some planning is required to make sure a sensible mesh is constructed with more elements around the hole, where the stress gradients are greatest.

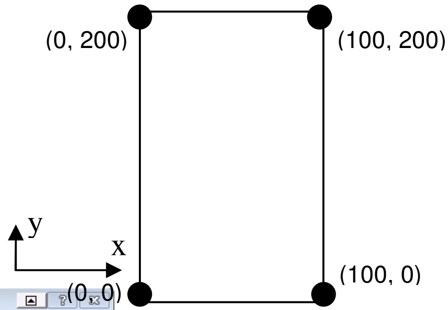
In Visual mesh there is a special option to automatically mesh a hole in a plate and we shall take advantage of this. The meshing procedures are:

- First create the plate with a suitable regular mesh (ignoring the hole).
- Then create the central hole and modify the mesh (there are special options for this).

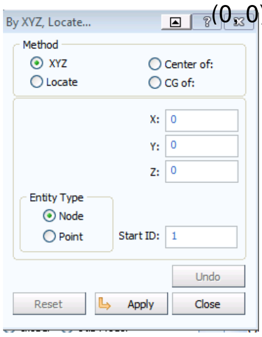
Either the full plate with the central hole can be analysed, or advantage can be made of problem symmetry to reduce computation costs. Taking advantage of symmetry may be less important today with the high speed of modern computers; however, for large problems and parametric/optimisation studies, where many analyses may be needed, it can still be a very useful option. We shall analyse both the full and quarter symmetry models in this exercise.

Meshing the plate (without hole)

The plate has four corner points that must first be defined. It is arbitarily chosen that the 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; (use z=0 for all points).



The four corner nodes are defined using the **Node > By XYZ, Locate…** panel in which the coordinates for each point 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.

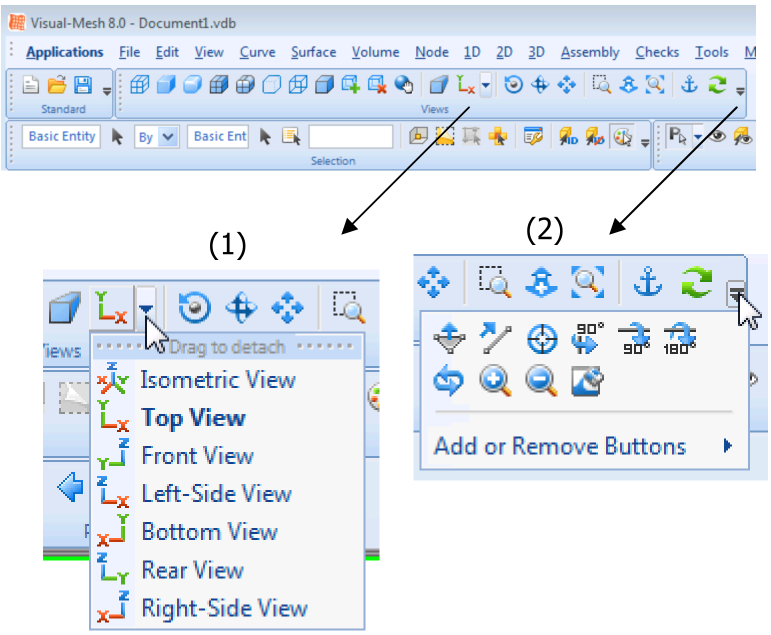


Positioning and viewing options:

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

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

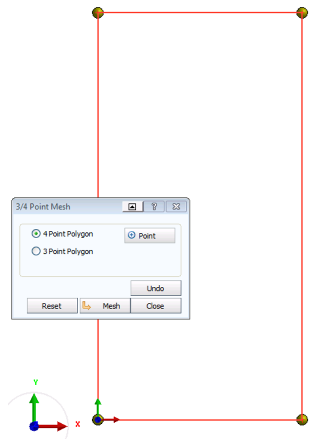
Spend some time to review these options, they are important.



Generating the mesh surface

- Position the model into a convenient view (e.g. using the **XY** tab and **Center** tab) so it appears in the center of the screen (see previous information).

- Activate the **2D > 3/4 Point Mesh** tabs with the **4 point polygon** option. Then click on the four corner nodes in sequence (clockwise or anti-clockwise). On clicking the last node the surface is generated as shown.

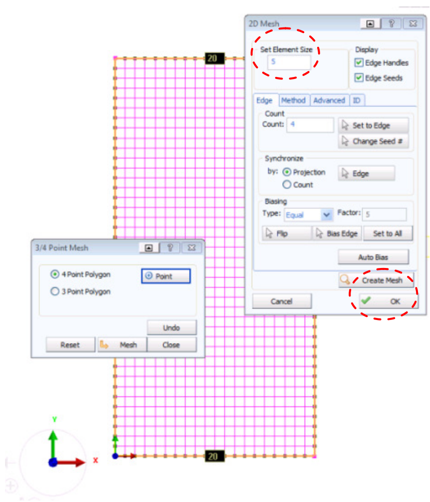


Generating the plate mesh

Click on the **Mesh** tab and a new panel opens to control details of the mesh to be generated.

Element sizes, grading and connection options (stitching) to possible adjacent meshes can be defined and controlled.

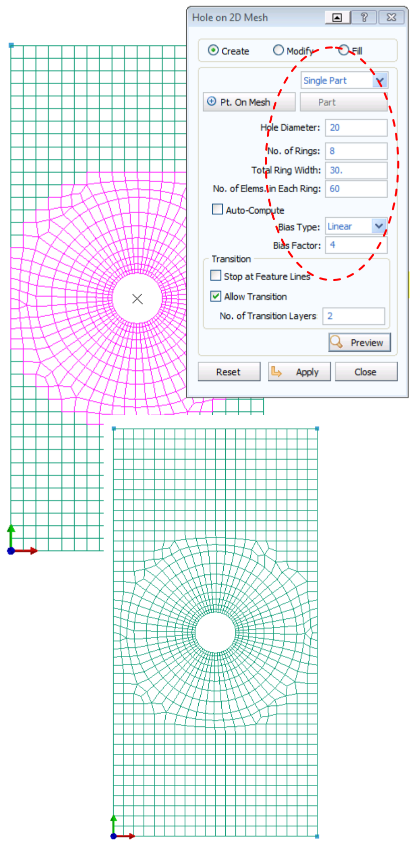
specify a 5mm element size and click the **Create Mesh** tab. The mesh will appear, accept this with the **OK**.



Generating the mesh with hole (Full model)

Click on the **2D > Hole on 2D Mesh** tabs and a new panel opens to control details for generating the hole in the plate.

Deactivate the **Auto-Compute** tab and specify a hole diameter of 20 mm with 8 rings to be meshed having a 'linear' bias of 4 (the outer element lengths are 4 times the inner ones). In this case the ring width is 30mm and 60 elements are specified in the circumferential direction. In addition VISUAL will ask you to select a node on the plate to which the hole center is located (use the left mouse key to select the plate center node). Click tab **Preview** to visualise the mesh. If this looks suitable click **Apply** and **Close** to accept it.



Remarks:

- The new mesh can be generated and cancelled untill you are satisfied with the results.
- Try to generate something that has sensible mesh grading bearing in mind the likely stress gradients of the loaded problem.

- Visual may have trouble to fit a meshed hole to the base plate mesh and not all combinations are feasible. Some experimentation of the input data may be needed before the mesh is generated.

Saving the Full model (mesh only)

This now completes meshing of the full model with **Visual Mesh**. Save the data with **File > Export** to a suitable folder with a suitable name (e.g. **HoleInPlate_FullMesh.pc**). Finally exit **Visual Mesh**.

The next steps look at modifying this model for quarter symmetry and mesh refinement.

Preparing the 'coarse' quarter model mesh

Creating the quarter symmetry model

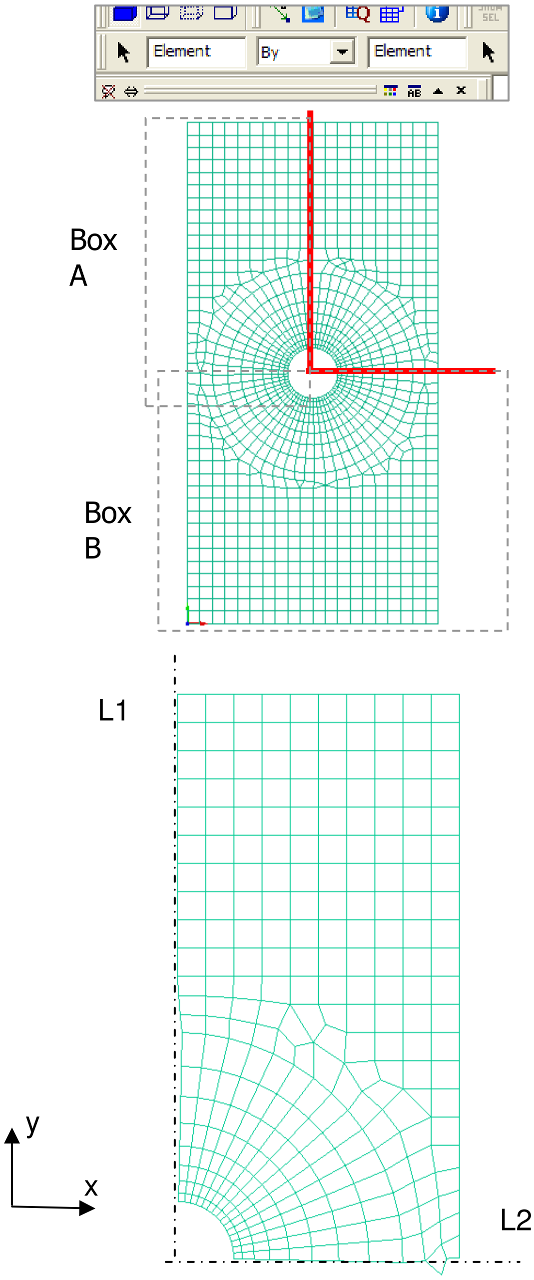
Probably the safest is to make a copy of the full model just created to HoleInPlate_QuarterMesh.pc.

Then start a new session of **Visual Mesh** in which the new dataset is read in for modification. Start **Visual Mesh** and use **Open File** then select the dataset HoleInPlate_QuarterMesh.pc to read it in.

Center the model in the x-y plane and visualise using the wireframe option to get the adjacent view.

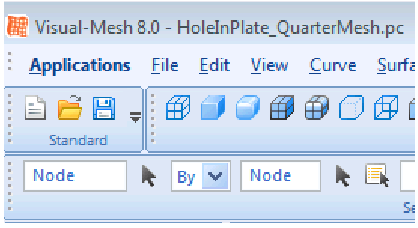
First elements outside the required quarter model are eliminated. The following steps are done:

1. Activate the selection option "Element – By – Element" in the top control panel.
2. Carefully draw boxes A and B over the required elements to be eliminated (drag the depressed left mouse key), then press the right mouse key and activate **Tools > Delete**; confirm with **Yes**. In practice this may be more easily done by repeating the procedure several times with smaller boxes and zooming into areas for better selection.
3. The aim is to delete all elements to the left of the line L1 (x=50) and below Line L2 (y=100). In the next step the irregular cut planes will be repositioned and straightened to lie on the required symmetry planes L1 and L2. *(Note: this may not be the best way to do this, but it does demonstrate some useful options!*)

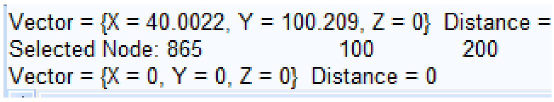


Finding nodal (and other) properties

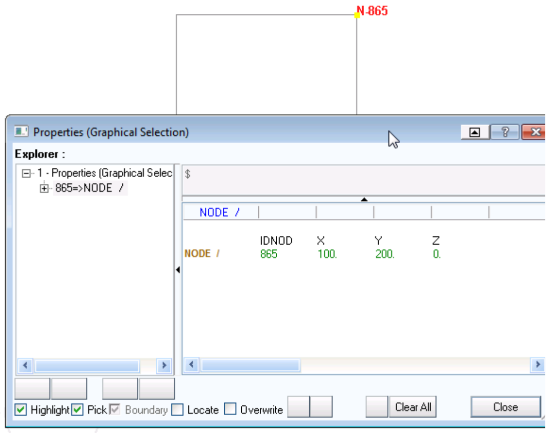
First make sure the selection option is active (e.g. Node-By-Node) in the top panel. Then click on the **required node** and it will be highlighted.



Coordinate information for the node will appear in the lower panel.



Alternatively, click the right mouse key and then **Tools > Properties** to open a new panel with the required nodal information will then appear.

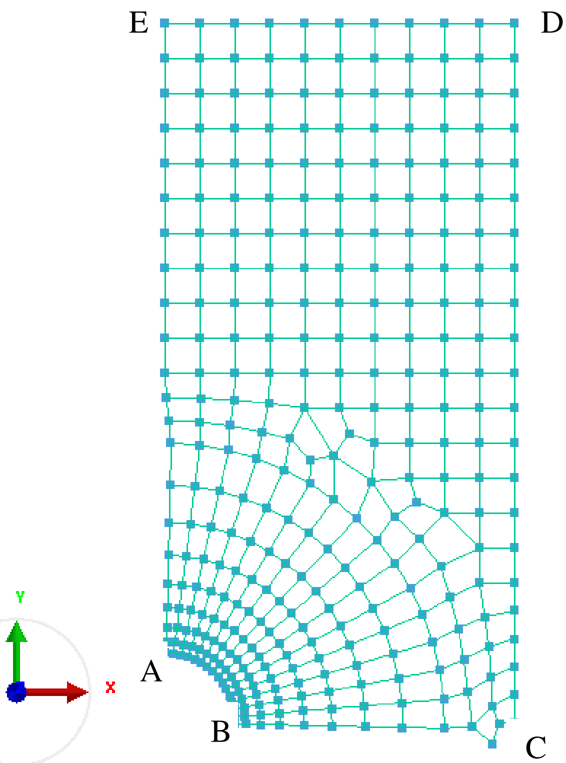


Similar operation are followed to get other information such as element numbers and properties.

Adjusting nodal coordinates (1/2)

Using the above operations it will be found that some of the key corner nodes do nor have the correct coordinates due to the automated meshing operations. The following table gives the required co-ordinates:

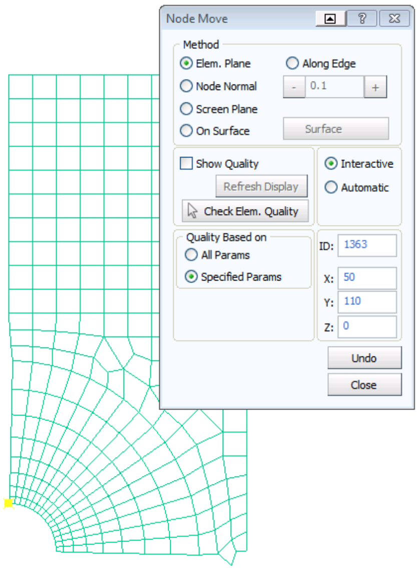
Point	X	Y	Z
A	50	110	0
B	60	100	0
C	100	100	0
D	100	200	0
E	50	200	0



The following operations will adjust nodal cordinates if necessary.

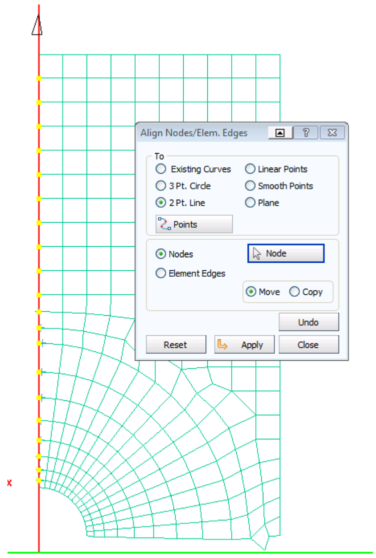
Adjusting nodal coordinates (2/2)

1. The option **Node > Move (Correct 2D/3D Element Quality..)** is used to adjust any corner nodes (A through E) that may be necessary. Use the options to check the nodal coordinates; if changes are needed enter the new valuse and click Close to modify them.



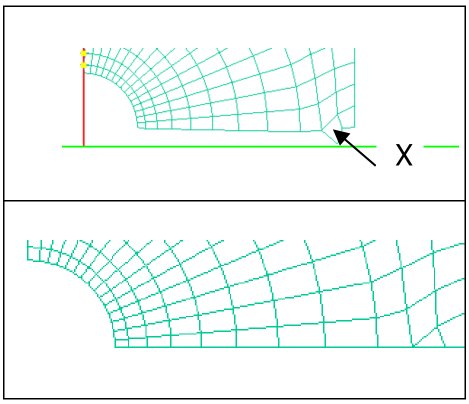
2. Next the nodes between B-C and A-E need to be aligned. Use the option **Node > Align** tabs and activate the **2 Pt. Line** option. For A-E alignment click on the first extremity node (A) and then the last extremity node (E), then click on the nodes to be alligned. A box can also be draged over the intermediate nodes. Click **Apply** and **Close** to impose the nodal coordinate changes.

3. Repeat this procedure for the nodes at B and C with the intermediate nodes between B-C. The upper and right most lines should not need adjustments.



Possible problems

Beware that generated meshes may vary depending on the automated meshing process. In this case the generated mesh and modifications made have created a 'bad' quadrilateral element with two sides being parellel (X)! This would not be accepted by PAM-CRASH. We shall elimiate it and replace it with a proper triangle.



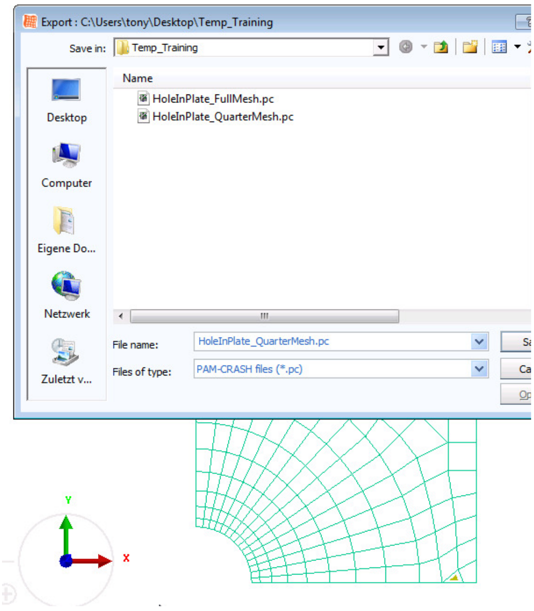
Activate **Element-By-Element** selection in the top panel, click on the element (X) and delete it with the (Entf/Del) key. Be careful that only this element is highlighted for deletion.

With option **2D > 3/4 Point mesh** and option **3 Point Polygon** a proper triangular element can be created by clicking the three corner nodes in sequence. Then click **Mesh** to open a meshing panel; use a large element size (e.g.10mm) so only one element is created and make sure the part ID is set to that of the other elements (Part ID = 1), so it gets the same number.

Saving the quarter model (mesh only)

This now completes meshing of the quarter model with **Visual Mesh**. Save the data with **File > Export** to a suitable folder using a suitable name (e.g. the original name **HoleInPlate_QuarterMesh.pc**).

Then exit Visual Mesh.



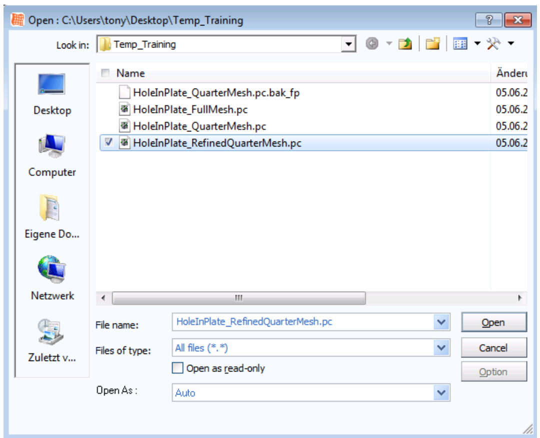
Preparing 'refined' version of the quarter model mesh

Again, probably the safest is to make a copy of the quater model just created to **HoleInPlate_RefinedQuarterMesh.pc**.

Then start a new session of Visual Mesh in which the new dataset is read in for modification. Start Visual Mesh and use **File > Open** then select the new dataset,

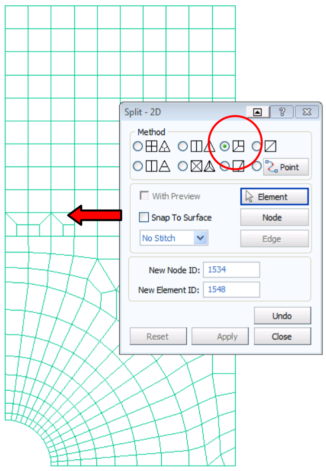
HoleInPlate_RefinedQuarterMesh.pc

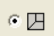
and read it in.



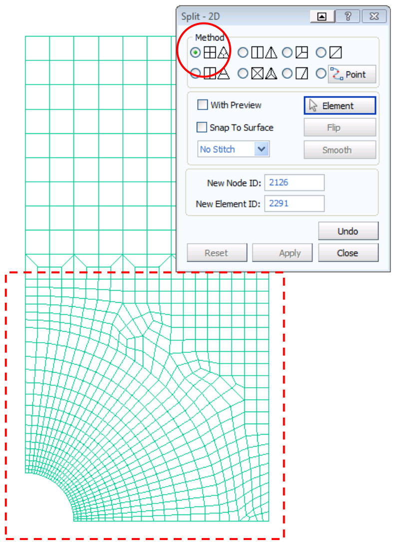
Remeshing the transition zone and refinement areas

The refirned model is to have mesh refinement in the lower half of the model for better prediction of stress and strain distributions around the hole.



1. For this the **2D > Split** option is used. First a line of elements for the mesh transition are remeshed using the option . The adjacent picture shows the line of transition elements that have been introduced (chose a similar line).

   The element to be remeshed is first selected with the left mouse key and then the corner node for the diagonal intersect is selected, again with the left mouse key. Note carefully the sequence to make sure there are no free nodes (except on the lower face where additional remeshing is made in the next step). If the remeshing is wrong select **Undo** before proceeding; otherwise continue the remeshing.



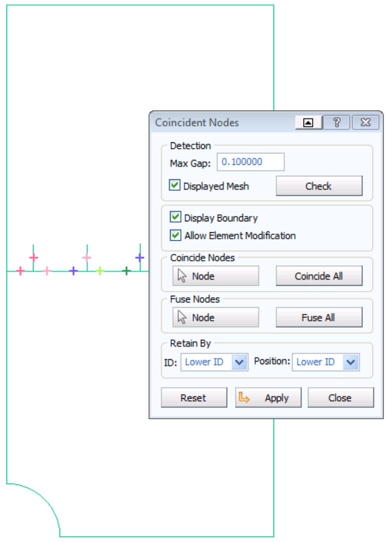
2. Finally the **2D > Split** tabs are selected with the remeshing option shown to split quadrilaterals and triangles. A box is drawn with the left mouse key over the elements to be refined and the new mesh is generated; accept with **Close**.

Combining the mesh

In the mesh refinement process duplicated nodes have been produced at the boundaries where new elements have been created. These duplicated nodes must be merged and removed.

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** tab. Click **Apply** and **Close** to finish.

If the Coincident nodes Check option is retried a view of the model showing only the outer free edges should be drawn.

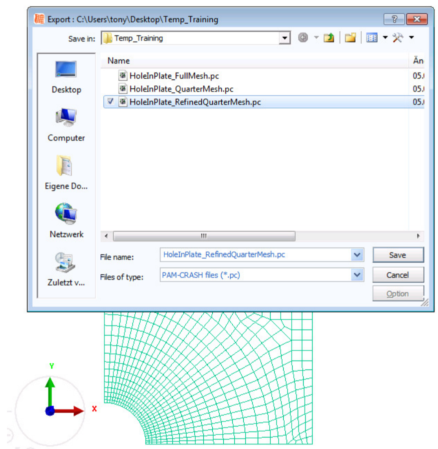


Saving the refined quarter model (mesh only)

This now completes re-meshing of the refined quarter model with Visual Mesh. Save the data with **File > Export** to a suitable folder using a suitable name; e.g. overwrite the existing file,

HoleInPlate_RefinedQuarterMesh.pc

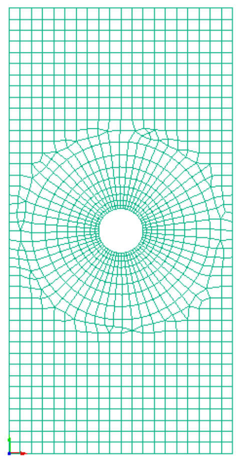
Then exit **Visual Mesh**.



Defining the model data (Visual-Crash PAM): Some general comments

Preparing the model(s) for analysis

Loaded end



Restrained end

Full model:

- Application of boundary conditions on the lower face.
- Application of loading on the upper face.
- Application of part (geometric) and material data for elastic and elasto-plastic solutions.
- Application of the PAM-CRASH control data.

Quarter (symmetry) models:

The same steps are followed as for the full model, except that specific boundary conditions must be assigned on the (x) and (y) symmetry planes to impose necessary symmetry.

Starting the Visual-Crash PAM work

The three meshed models (Full, quarter and refined quarter models) created so far will be modified for analysis. First elastic materials analyses will be undertaken and later these datasets will be copied and used for plastic analysis.

The safest way is to make copies of the three meshes and work with the new copies. This will allow you to go back and start again with a new copy if things go wrong!

Copy:

HoleInPlate_FullMesh.pc
to **HoleInPlate_FullModel_Elastic.pc** for elastic analysis } Full model

HoleInPlate_QuarterMesh.pc
to **HoleInPlate_QuarterModel_Elastic.pc** for elastic analysis } Quarter model

HoleInPlate_RefinedQuarterMesh.pc
to **HoleInPlate_RefinedQuarterModel_Elastic.pc** for elastic analysis } Refined quarter model

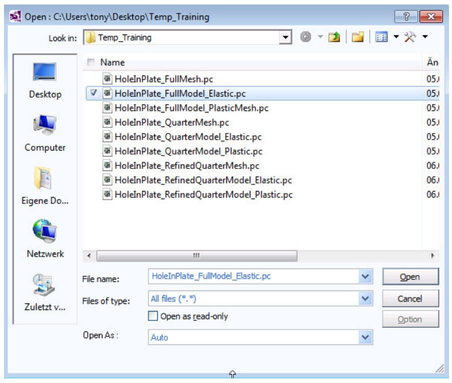
Preparing the full analysis model: Elastic analysis

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,

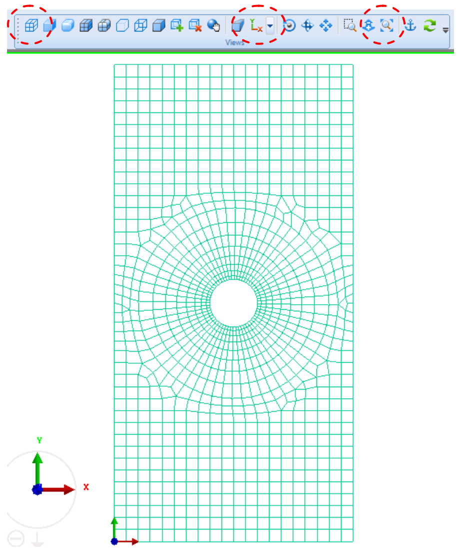
HoleInPlate_FullModel_Elastic.pc



Positioning and centering of the model

Use the x-y and centering tabs to conveniently position the model. This will make assignment of the bounday 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 top panel.



Boundary conditions

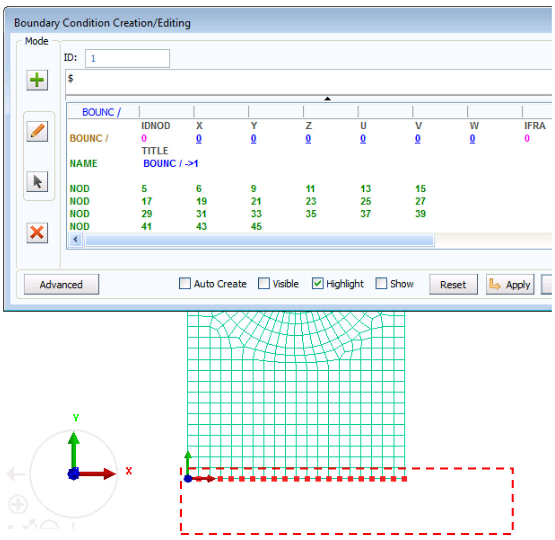
Fixing the constrained end

Click on the tabs **Crash > Loads > Displacement BC** to get the panel to assign nodes and conditions. Use the left mouse key to draw a box over <u>all the nodes</u> on the lower face, then click the **Green Arrow** and the selected nodes will now appear in the list.

Activate the required boundary conditions for these nodes (1=fixed, 0=free), in this case fix:

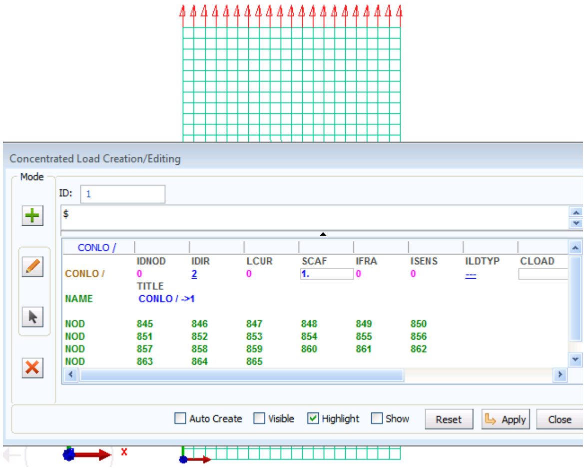
- Fix the y,z translational directions (=1),
- leave x free (=0) so contraction in the width direction is possible.
- Fix the U,V,W rotations (=1).

Choose an appropriate title for the boundary condition; e.g. "boundary conditions at holding end". The save the selection with **Apply** and **Close.**



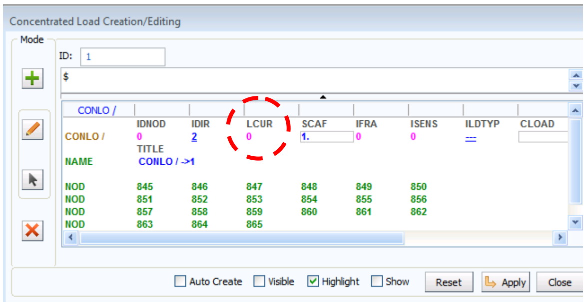
Loading conditions (on the top face)

Click on the tabs **Crash > Loads > Concentrated Loads** to get the panel to assign nodal loads. Use the left mouse key to draw (drag) a box over all the nodes on the top face and click the **Green Arrow**; the selected nodes will now appear in the nodal list.

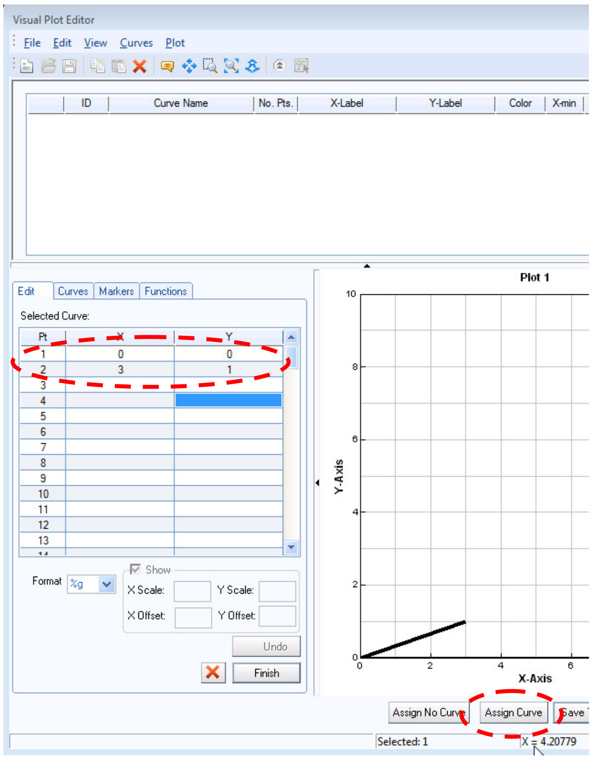


We now set the load direction (set IDR = 2) and assign the loads to these nodes via a load load curve (in this case we shall use a linearly increasing curve to 1kN over 3 msec for each node).

1. Click on **LCUR** and activate **New.** A new panel to define the load curve will open. Click on **File** and **New** for the possibility to define the data.



2. In the new panel define the load curve function. In this case x-axis is time and y-axis is applied load. Assume the load starts at zero and increases to a constant load of 1kN at 3 msec. The simulation time for this problem is 3 msec (to be defined later). Note that the applied loading is 1kN at each of the 21 nodes (total load =21kN). Finish with **Assign Curve**.



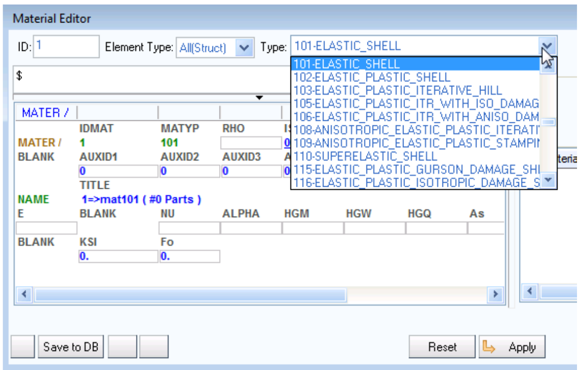
3. Some final points:
   - The applied load could be scale if needed, but we shall use SCAF = 1.
   - Set ILDTYP = 1 for load to be applied to all nodes (this is the default).
   - Give the nodal loads a suitable title so it can be easily identified in the PAM-CRASH dataset file.
   - Make sure the load curve (LCUR) is assigned to this set of nodes.

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

Assigning the Elastic Material and Part data

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

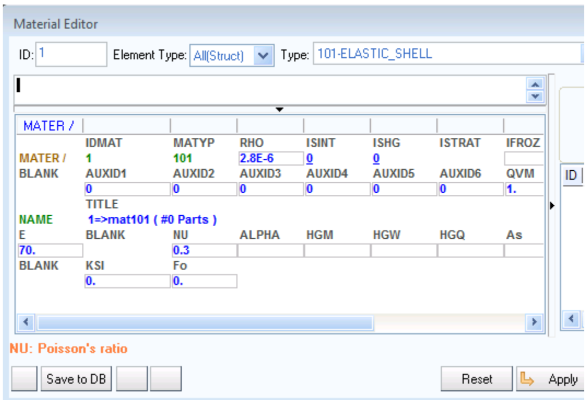
1. The Material data for material data such as modulus, density and plasticity information.
2. The Part data for (geometric data such as thickness).



1. Material data

Select **Crash > Materials Editor** and then select the type of Material Model; in this case use a **101 – ELASTIC_SHELL.** Then define the material data for Aluminium (E=70 kN/mm^2, v=0.3 and density 2.8*10^{-6} kg/mm^3).

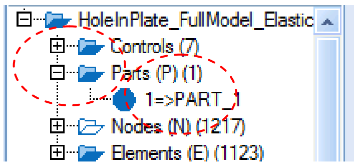
The material assignments are closed with **Apply** and **Close**.

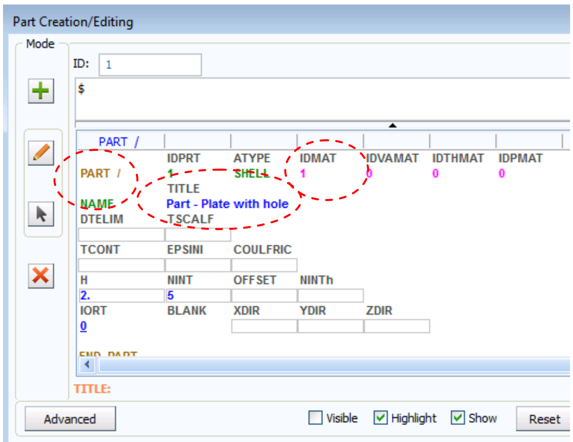


2. Part data

This is easily done via the explorer panel:

1. First click on **Parts** and the list of parts will open; select the required part (=PART_1) and with the right mouse key select **Edit**. The parts panel will open.
2. Set the shell thickness to H=2mm. Click just below **H** and set this to 2.
3. Finally, the material to be linked to this part must be assigned. Click on **IDMAT,** then **List** and the materials panel will open. Select the required material and click **OK**.
4. Give the part a sensible name.





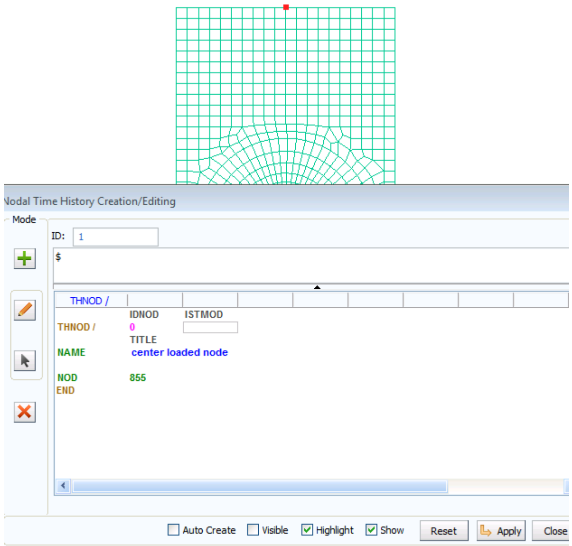
Finish the material/part assignment with **Apply** and **Close.**

Output results

Some useful output could be selected (this is optional)

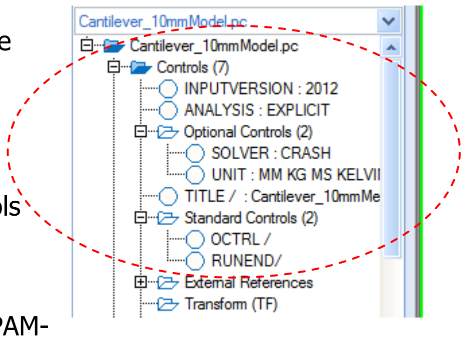
For time history at a node click **Crash > Output > Nodel TH** and select a node (e.g. the center node at the loaded end) and complete the selection by clicking the **Green Arrow.** Give a suitable title and finish with **Apply** and **Close.**

For the element repeat the process but using **Crash > Output > Element TH** and selecting the required element (for example some elements around the hole) which will have the maximum stresses.



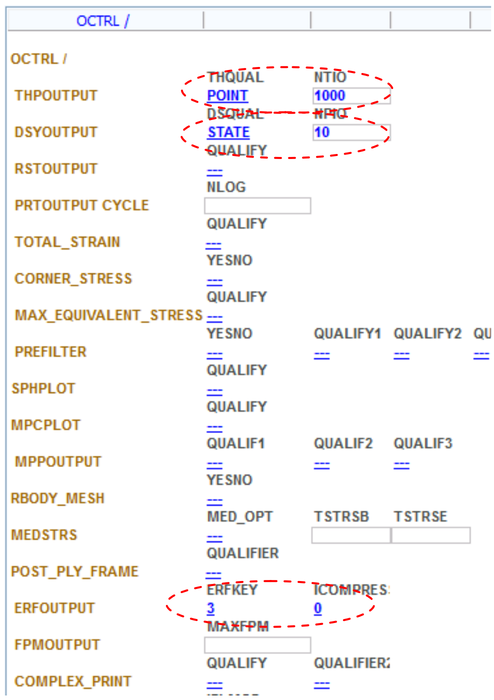
The PAM-CRASH control data must be set

This controls the analysis and allows specific data like the problem analysis time (in msecs in this case), and ouput 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 tiltle (e.g. HoleInPlate_FullMesh Elastic); 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=3.0) 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**.



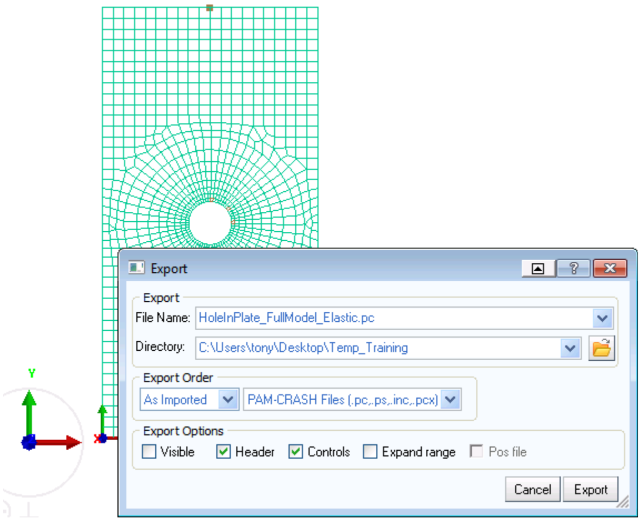
Saving the dataset (.pc file)

- Save the model in a suitable directory/name; e.g. **HoleInPlate_FullModel.pc.**

- Click on **File > Export** and export the model as PAM-CRASH input dataset.In this case overwrite the file,

  HoleInPlate_FullModel_Elastic.pc

This dataset should be complete and ready for a PAM-CRASH analysis.



Preparing the quarter (coarse and refined) analysis models for elastic analysis

Essentially the same steps are followed for preparation of the quarter models (coarse and refined meshes) for elastic analysis. A brief outline and differences to the previous full model preparations are detailed below. These models will be analysed and results evaluated before moving on to the elasto-plastic models.

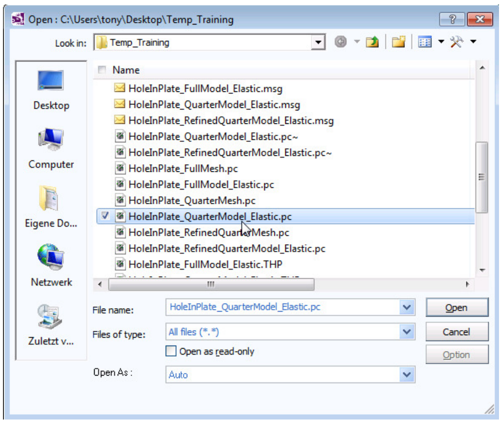
The datasets (already copied) and to be used here are:

HoleInPlate_QuarterModel_Elastic.pc
HoleInPlate_RefinedQuarterModel_Elastic.pc

For each dataset the following modifications are made.

---

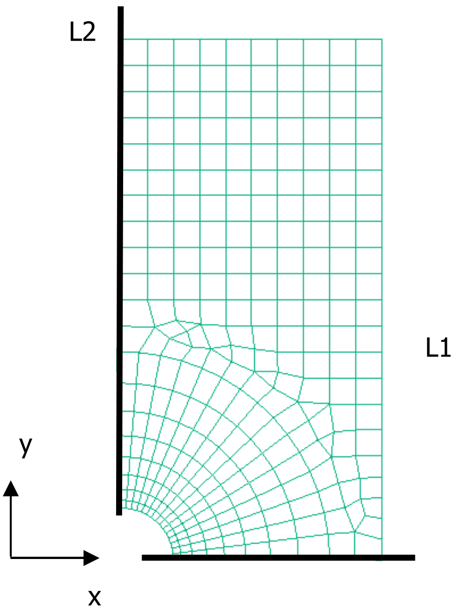
1. For each dataset start **Visual Crash PAM** and read in the file with **Open File**.



2. For boundary conditions:

   Symmetry on line L1 - use 011111 conditions
   Symmetry on line L2 - use 101111 conditions

   where 0 and 1 are standard FE conventions meaning (0=free) and (1=fixed) in the x,y,z,U,V,W directions respectively.



3. Loading on the top face is the same as the previous full model. But it should be noted that strictly the centerline node should carry only half the load (=0.5kN) due to the symmetry conditions.

4. Materials and Parts information are the same as for the previous full model.

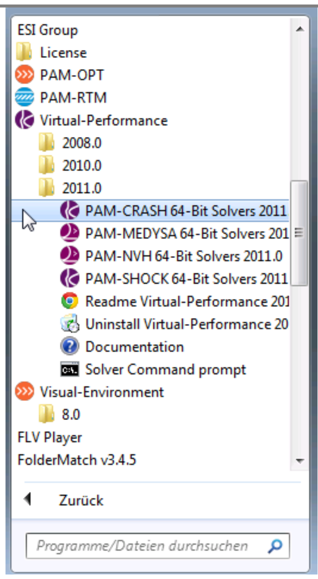
5. Control information's are all the same as for the previous full model. But you should use different names for the Title control card. Remember to save results in the .ERF file format; set this in the OCTRL panel - activate type 3 without compression (ICOMPRES=0).

6. Save (overwrite) the completed models.

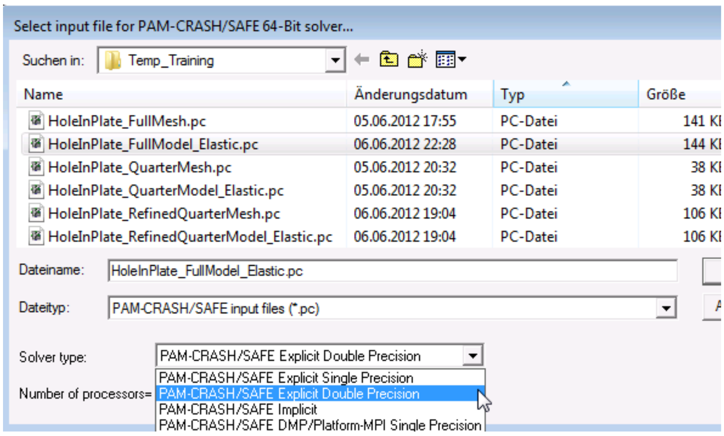
Part 2: Elastic explicit models: Analysis and results

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 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'.

```
N O R M A L   T E R M I N A T I O N

     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
```

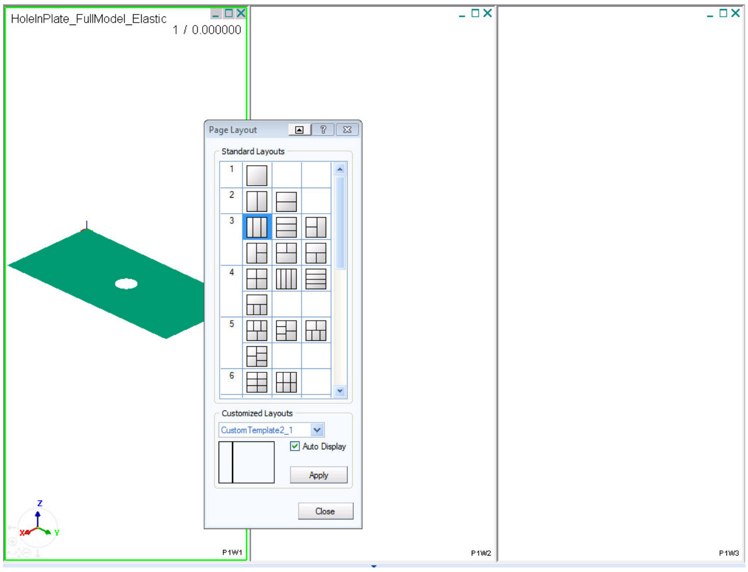
- If there are data errors the run will stop with an abnormal termination message. Inspect the output/results files for errors (search for 'ERROR' and investigate). Correct the dataset; preferably in VISUAL CRASH PAM (or in the editor) and rerun the analysis.

Evaluating the results

Execute the three elastic analysis models:

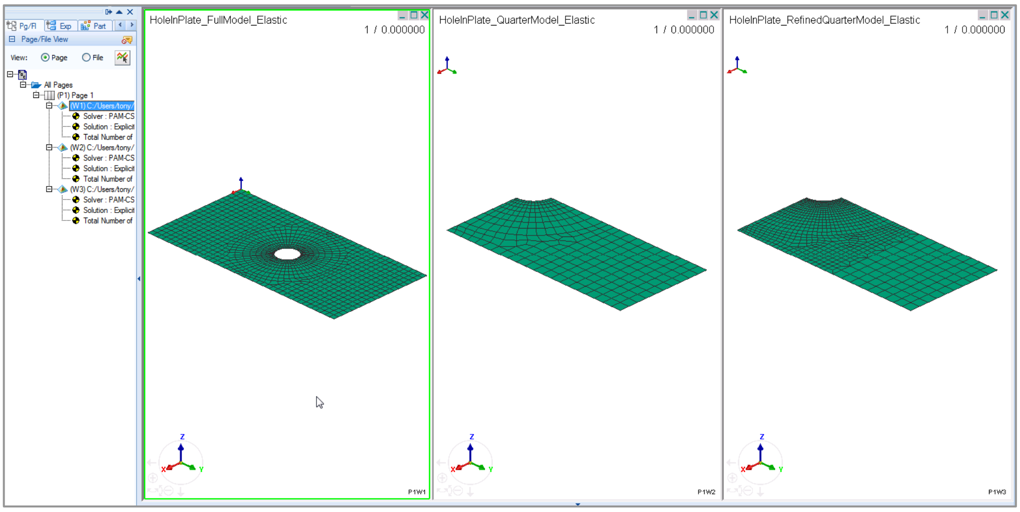
HoleInPlate_FullModel_Elastic.pc
HoleInPlate_QuarterModel_Elastic.pc
HoleInPlate_RefinedQuarterModel_Elastic.pc

Check the analyses finish with 'NORMAL TERMINATION' (if not check for errors, correct and rerun the models). For the three models it is convenient to visualise results simultaneously in one viewing frame. Start **Visual Viewer** and open results file **HoleInPlate_FullModel_Elastic_RESULT.erfh5.** Then click on windows > Page Layout and the variation shown below below.



This will open three viewing windows with the current results occupying the first window. Click on each of the new windows in turn (left mouse key) to activate it and then read in the remaining results files to each window using **File** and **Open**,

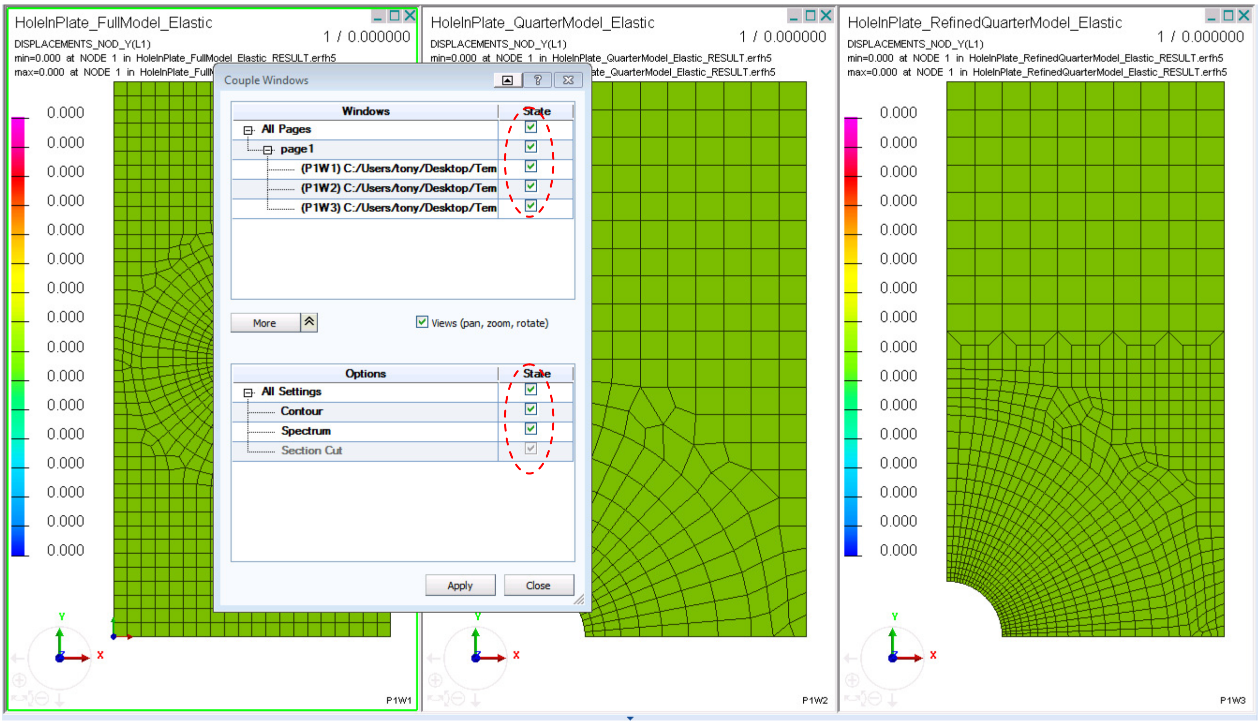
1. **HoleInPlate_QuarterModel_Elastic_RESULT.erfh5**
2. **HoleInPlate_RefinedQuarterModel_Elastic_RESULT.erfh5**



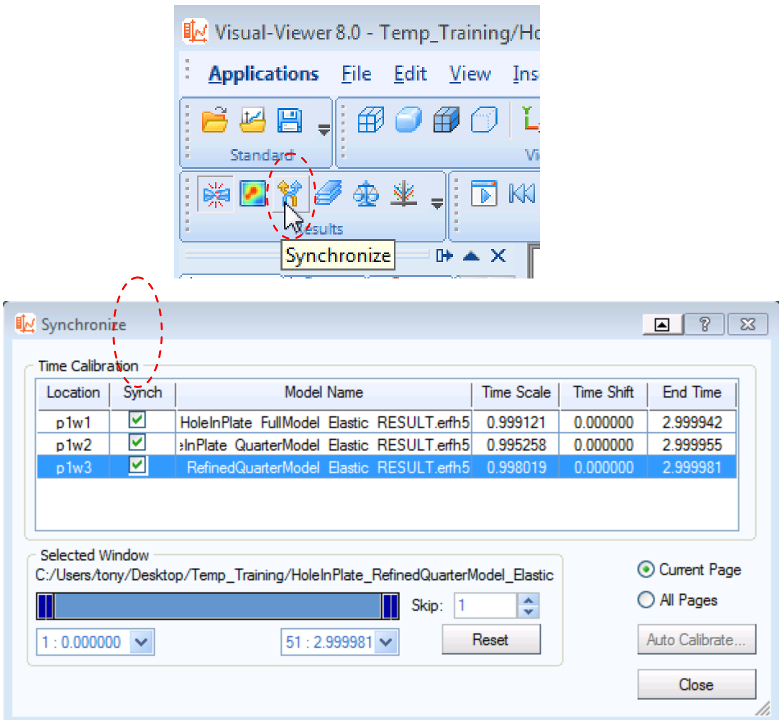
The next step is to synchronise the three viewing frames. Ths is done on two levels:

1. For coupling of viewing information
2. For coupling of state (time) plots

For viewing coupling click on **Window** > **Couple Windows** to open the synchronisation panel. Then activate the three models and also the **Views** box and the options under **More**. This will now force all windows to be syncronised for zooming, viewing, contour plots, etc. Finish with **Apply** and **Close**.

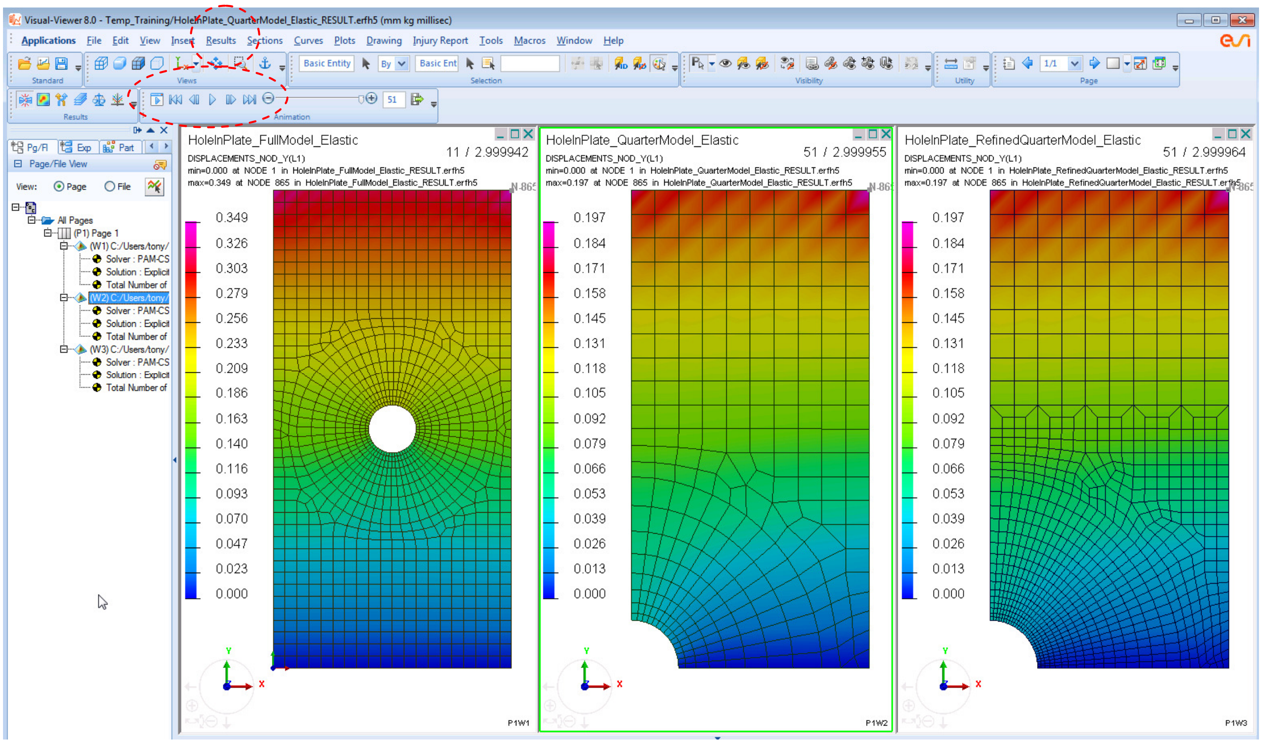


For state coupling click on the **syncronisation** tab shown below and then activate the three models to be syncronised. Finish with **Close**.

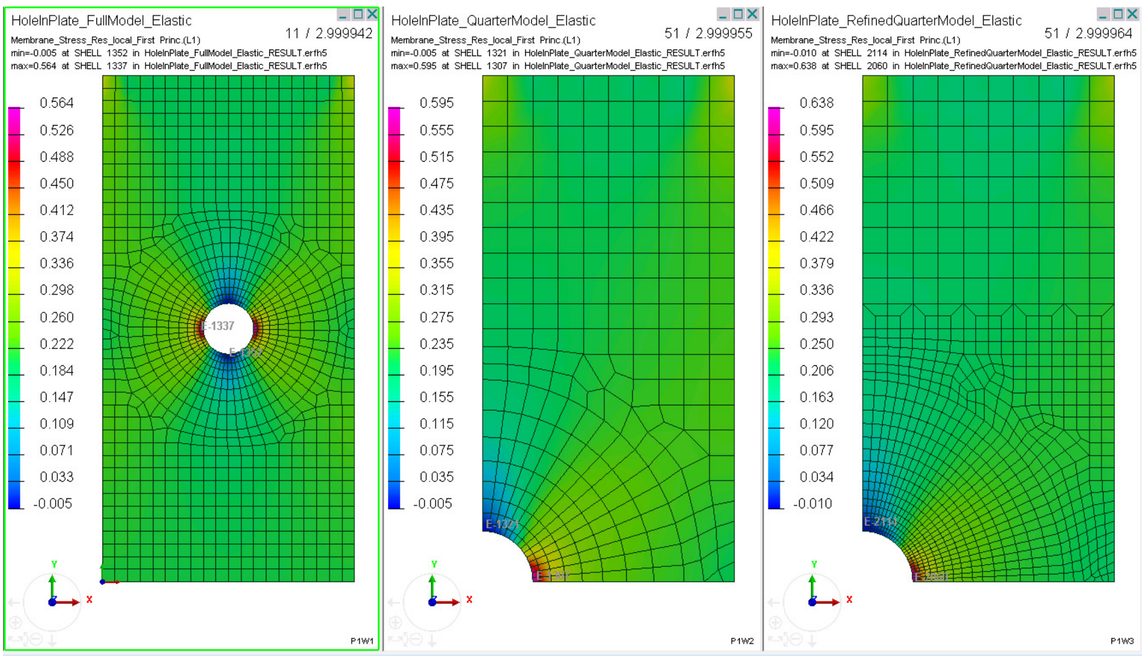


<u>Tutorials 3,4: Elastic and elasto-plastic loading of a plate with a central hole</u>

The following is a typical set of plots that may be produced. In this case the last state (≈3msec) is selected together with the contour variable (Displacement_Nod_Y) to show nodal displacements in the loading direction. The states can be selected via **Results** > **Animation Control**, or the tabs shown below. The contours are selected via **Results** > **Contour**.



Finally, contour plots for example of membrane stress (resultant first principle) are obtained by clicking **Results > Contour** and selecting entity type **SHELL** and this contour. For the full plate the loading resultant is total applied load/width = 21kN/100mm = 0.21kN/mm. The maximum stress resultant from the plot is 0.564 giving a stress concentration factor for the notch of 0.564/0.21 = 2.7, which is a realistic value. As expected the finer meshed model shows higher stresses.



Part 3: Elasto-Plastic explicit models: Modifications, analysis and results

Preparing the elasto-plastic analysis models

The previous elastic model are now modified for elasto-plastic analysis. The only differences are made to the material law (elastic model 101 is changed to elasto-plastic model 103). The datasets to be copied and modified are:

Copy **HoleInPlate_FullModel_Elastic.pc**
to **HoleInPlate_FullModel_Plastic.pc** for plastic analysis } Full model

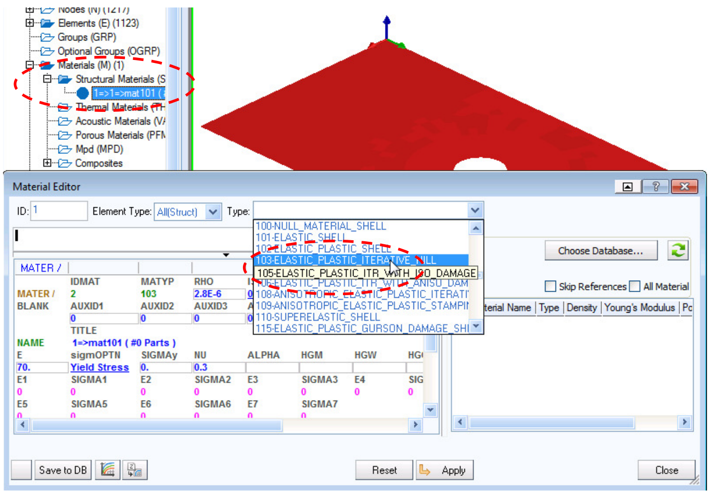
Copy **HoleInPlate_QuarterModel_Elastic.pc**
to **HoleInPlate_QuarterModel_Plastic.pc** for plastic analysis } Quarter model

Copy **HoleInPlate_RefinedQuarterModel_Elastic.pc**
to **HoleInPlate_RefinedQuarterModel_Pastic.pc** for plastic analysis } Refined quarter model

The following gives details of the changes to be made to each of the above _Plastic datasets. Start **Visual Crash PAM**, open each file in turn and make the changes.

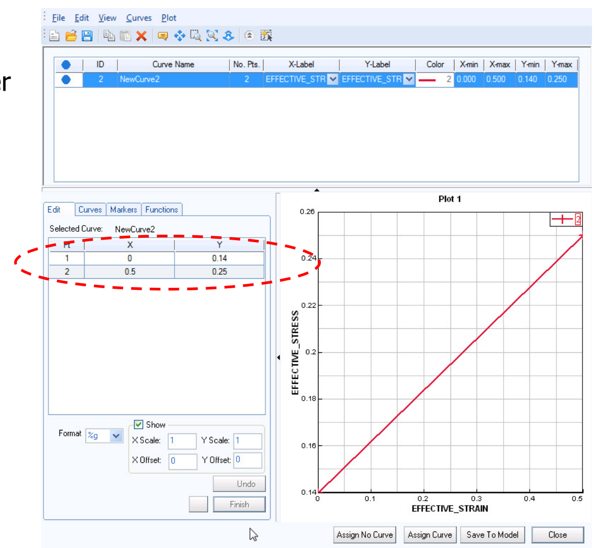
Assigning the Elasto-Plastic Model

1. Material data



In the explorer window the current elastic materials model can be clicked and with the right mouse key opened for editing. In the upper selection Type a new materials model **103 – ELASTIC_PLASTIC** can be selected. The existic data (E=70 kN/mm^2, v=0.3 and density 2.8*10^{-6} kg/mm^3) remain valid, only a new plasticity curve must be added.

The yield stress and plasticity information can be defined in a number of ways (yield stress, Curve, power law, …); the CURVE option is selected by clicking on **Yield Stress** and selecting the type **CURVE**; click the '**0**' just under LC1 and **New** to enter the following curve data.



In this case a simple 2 point curve is defined using:

X – effective strain	Y - effective stress
0.0 (start of plastic strain)	0.14 MPa (= yield stress)
0.5	0.25

Then click **Assign Curve**. Make sure this curve (number 2) is assigned to parameter LC1.

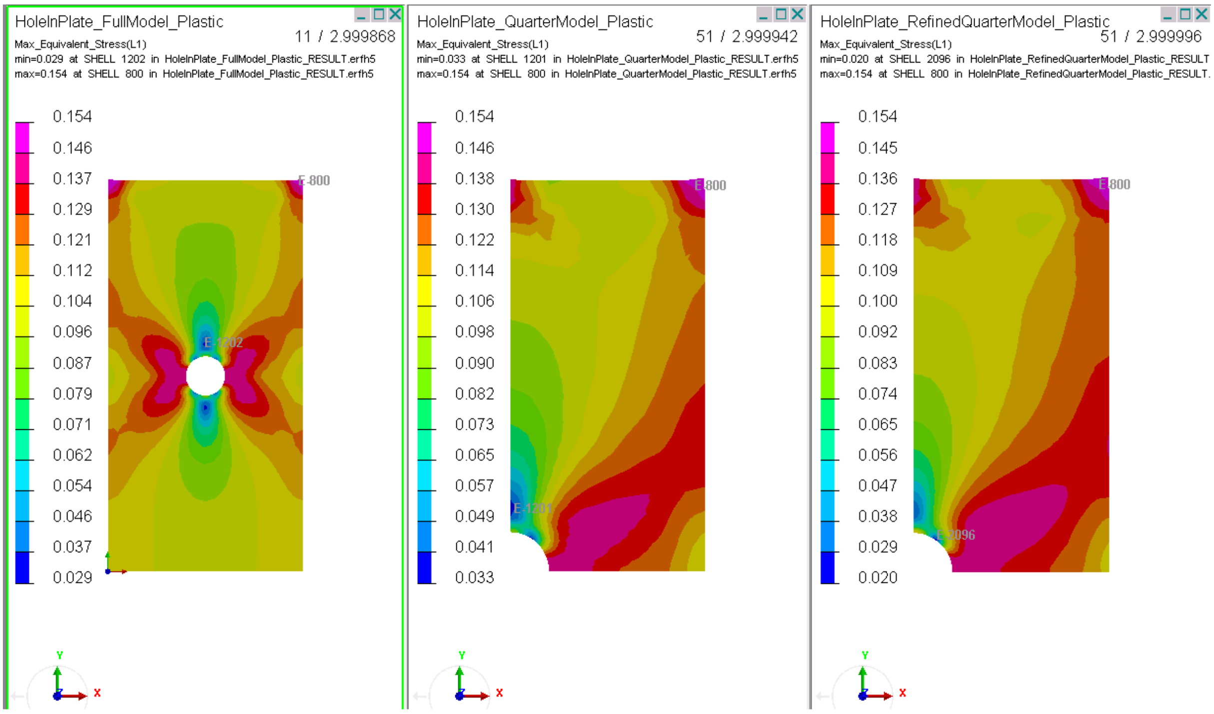
Finish the changes to the material assignments with **Apply** and **Close**.

<u>Tutorials 3,4: Elastic and elasto-plastic loading of a plate with a central hole</u>

Example results are shown below for Max. equivalent stress (= Von Mises stress). The three plastic models have been loaded into three windows and the previous operation for syncronisation of the three windows has been done.

The results are similar for the three cases and show that significant plasticity has occurred around the notch. Note that plasticity has occurred in regions that have a stress greater than the yield stress (= 0.15MPa).

One problem can also be seen in the corner loaded nodes; the deformations, stresses and strains are high due to high nodal loads being applied (proper uniform loading should have halved these loads for these nodes). It is an example how deformation and countour plots can be usefully used to identify modelling errors.



Part 4: Implicit models: Modifications, analysis and results

Preparation of the elastic and elasto-plastic models

The following list the simple changes needed to convert the explicit datasets to implicit ones:

1. Specify that an **Implicit** analysis will be undertaken.
2. Specify the type of analysis; in this case either a **Static**, **Linear** analysis will be performed for the elastic plate, or a **Static**, **Non Linear** analysis will be performed for the elasto-plastic plate.
3. Specify information concerning the number of loading steps to be undertaken to reach the maximum load (for the nonlinear elasto-plastic analysis only).
4. Specify the element formulation to be used; in this case **shell element type 6**.
5. Finally, for the full plate study only, one modification to the boundary conditions is necessary to prevent Rigid Body motion.

As previously the safest way is to copy the explicit datasets to implicit datasets, and then modify the new (implicit) datasets with **Visual Crash PAM**.

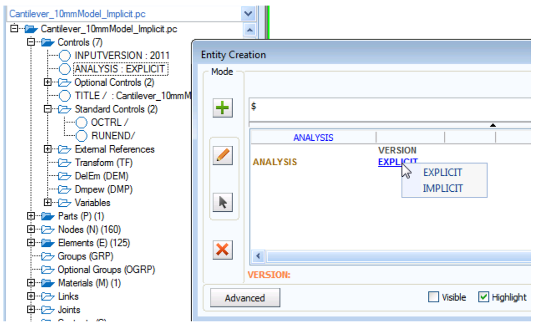
Copy **HoleInPlate_FullModel_Elastic.pc** to **HoleInPlate_FullModel_Elastic_Implicit.pc**
Copy **HoleInPlate_FullModel_Plastic.pc** to **HoleInPlate_FullModel_Plastic_Implicit.pc**

The same is done for the elastic and plastic **QuarterModel** and **RefinedQuarterModel** datasets.

The following give details of the changes needed for implicit analysis.

1. The implicit control card

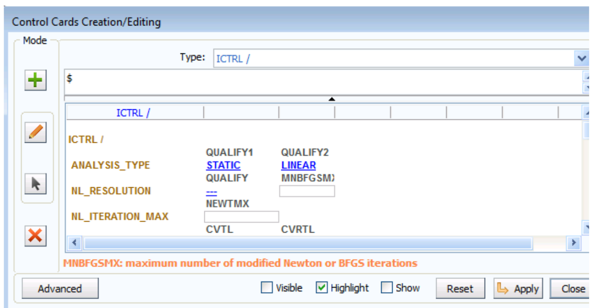
In the explorer panel under **Controls** the type of analysis can be changed from EXPLICIT to IMPLICIT.



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:

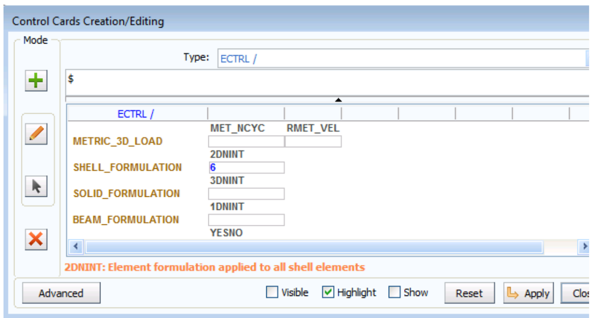
STATIC and **LINEAR** for the elastic cases.
STATIC and **NON LINEAR** for the elasto-plastic cases.



Tutorials 3,4: Elastic and elasto-plastic loading of a plate with a central hole

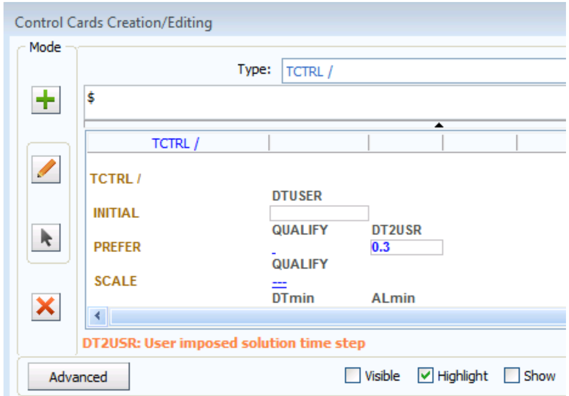
3. The element type

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.**



4. The solution times step (for the nonlinear elasto-plastic analysis only)

In the main window under **Crash > Controls > Standard Controls** look in the window **Type** for TCTRL. For value DT2USR set 0.3. This parameter specifies the time step size, and is used to control the number of load increments made; RUNEND=3.0, DT2USR=0.3 giving 10 increments (between 0 to 1kN in the nodal loads).



5. Changes to boundary conditions (<u>full plate study only</u>)

Boundary conditions for the quarter models do prevent any rigid body motion of the structure in both the translational and rotational directions. This is not the case for the full plate study and rigid body rotation (about the x axis) and rigid body translation (in the x direction) are possible. For an implicit analysis it is necessary that <u>all rigid body motions</u> are suppressed, so that the global stiffness matrix has a unique solution (i.e. the determinant is non-zero).

Consequently, fully constrain <u>any one</u> of the nodes on the lower face. If only one node is constrained the loading in the plate is identical to the previous explicit analyses.

Note in the explicit solution the dynamic equations of motion are solved which do not require suppression of rigid body modes. But clearly sensible constraints are still needed to resist applied loading.

6. Export the dataset

Finally, for each model export (overwrite) the dataset with the correct name and close **Visual Crash PAM**.

Check carefully that in the preparation of these analyses models:

- The elastic models **(filename_Elastic_Implicit.pc)** use the elastic material law 101.
- The plastic models **(filename_Plastic_Implicit.pc)** use the elasto-plastic shell material model 103.

Implicit results for the elastic and elasto-plastic models

Use **PAM-CRASH Implicit** to run the three elastic and elasto-plastic implicit problems,

HoleInPlate_FullModel_Elastic_Implicit.pc
HoleInPlate_QuarterModel_Elastic_Implicit.pc
HoleInPlate_RefinedQuarterModel_Elastic_Implicit.pc

HoleInPlate_FullModel_Plastic_Implicit.pc
HoleInPlate_QuarterModel_Plastic_Implicit.pc
HoleInPlate_RefinedQuarterModel_Plastic_Implicit.pc

Results from the analyses are processed in exactly the same way as the previous explicit analyses using **Visual Viewer** and opening the .ERF results files. For the linear analyses there is only one result state which corresponds to the maximum applied load, whereas for non-linear analyses a deformed state is stored for each load increment in the iterative solution scheme.

The following contour plots from the .ERF files show some example results for the six models. The figure uses a six window plot and syncronisation for the six models. The elastic studies (top row) and elasto-plastic studies (bottom row) show maximum equiv. stress (= Von Mises stress) for the final load. It can be seen that the elastic models have high localised elastic stresses at the root of the notch, whereas the non-linear studies show plasticity and give peak stresses at the yield stress with hardening.