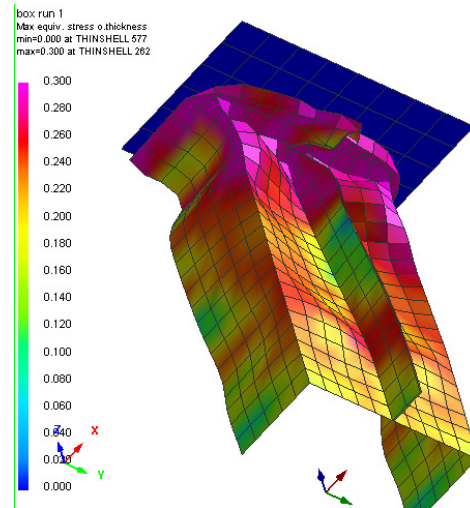


Tutorial 7

Axial Crushing of a Spotwelded Box Beam Structure



Problem description

Outline Axial crushing simulation of a thin walled spotwelded box beam section is undertaken including model setup and post-processing of results.

A convergence study with mesh refinements and changes to spotweld spacing and sheet thickness are made to show the effects on buckling modes, impact force and energy absorption.

Analysis type(s): Explicit

Element type(s): 3D Shell elements, Spotwelds

Materials law(s): Elasto-plastic

Model options: Boundary conditions, Spotwelds, Rigid walls, Contact

Key results: Stress distributions and Impact forces

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

Date: December 2007

Version: V4 (Updated May 2012 for Visual-Crash PAM V8.0)

Background information

Pre-processor, Solver and Post-processor used:

- **Visual-Mesh:** For generation of the refined model meshes (a completed coarse mesh model is provided) and other model changes.
- **Visual Crash PAM:** To apply modifications to the models (boundary conditions, velocities, etc.,).
- **Analysis (PAM-CRASH Explicit):** To perform the explicit Finite Element analysis.
- **Visual Viewer:** Evaluating the results for contour plots, deformations, etc.

Prior knowledge for the exercise

This exercise is not worked through in detail and it is assumed that the reader has gained a good working knowledge of the use of VCP from tutorials 1 through 6.

The completed model of the spotwelded box beam structure is provided and will be examined using **Visual Crash PAM**.

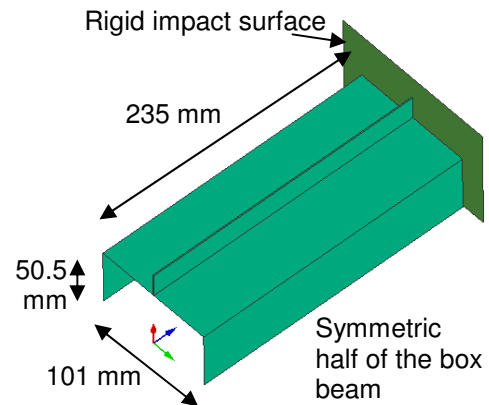
Problem data and description

Units: kN, mm, kg, ms

Description: Mild steel box beam section (symmetric half model), spotwelded on the flanges is impacted against a rigid surface.

Loading: Impact mass (of the wall) is 100kg at a velocity of 7.8 m/sec.

Material: Steel ($E=210 \text{ kN/mm}^2$ (GPa), $\nu=0.3$ and density $7.85 \cdot 10^{-6} \text{ kg/mm}^3$). For plasticity a yield stress of 200 MPa and constant plastic modulus 1.0 GPa to a hardening plastic stress of 300 MPa is assumed.



Supplied datasets

The completed coarse mesh model, including all entities, loadings, contacts and material/part properties is provided and has the name:

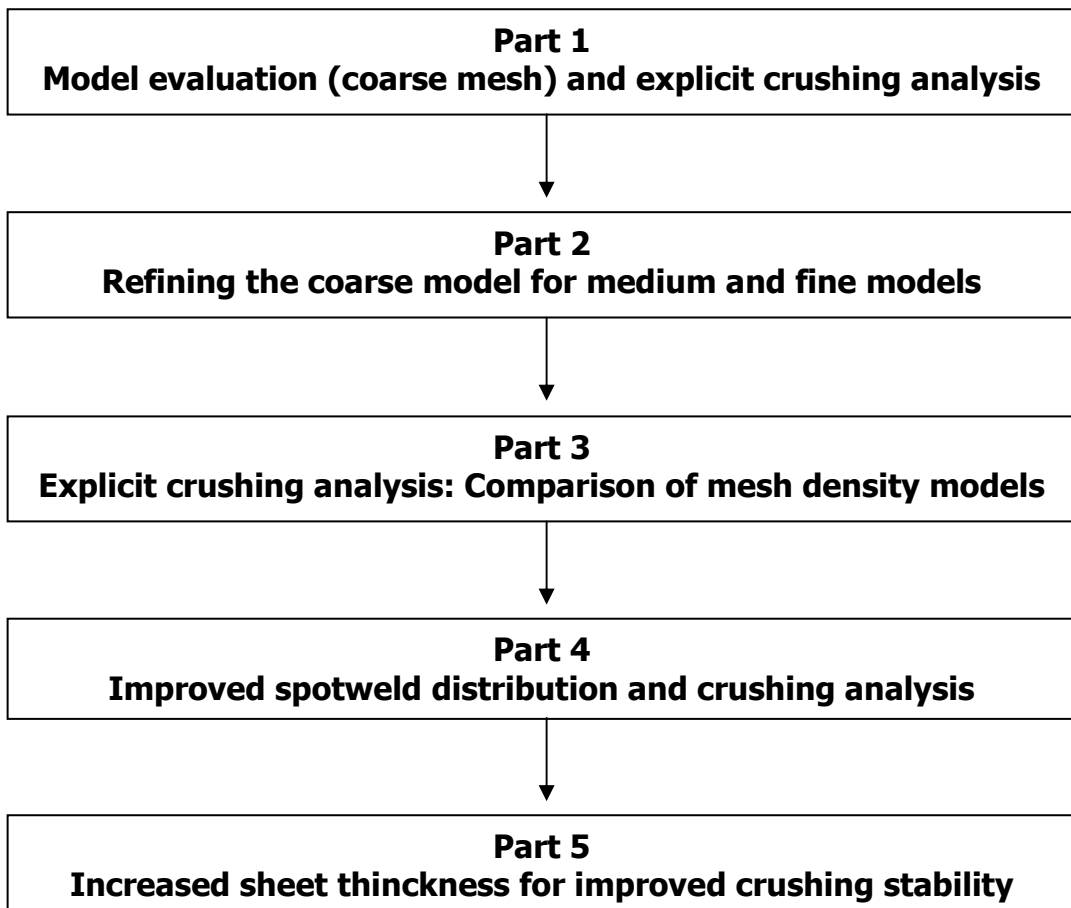
BoxBeam_CoarseModel.pc

During the exercise this model is modified and the mesh refined. It is recommended that copies of the coarse model are made, and these copies used in the exercise; copy:

1. **BoxBeam_CoarseModel.pc** to **BoxBeam_MediumModel.pc** for the first level of refinement
 2. **BoxBeam_CoarseModel.pc** to **BoxBeam_FineModel.pc** for the second level of refinement
-

Contents

Tutorial 7	1
Axial Crushing of a Spotwelded Box Beam Structure	1
Problem description	1
Background information	2
Part 1: Using Visual Crash PAM to examine the box beam	3
Part 2: Preparing the refined box beam models	8
Part 3: Running the models and investigating results	9
Part 4: Analyses with more spotwelds	11
Part 5: Increasing the box beam sheet thickness	12

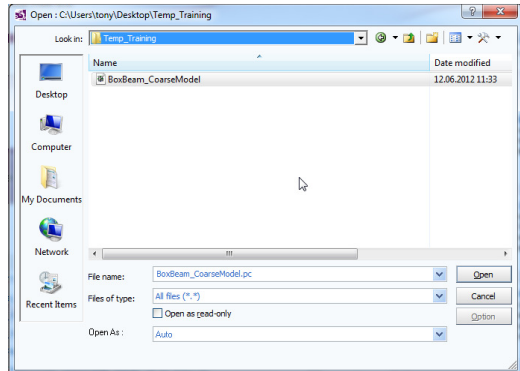


Part 1: Using Visual Crash PAM to examine the box beam

Starting Visual Crash PAM to examine the model

Start **Visual Crash PAM** and read in the file with **Open File**

BoxBeam_CoarseModel.pc

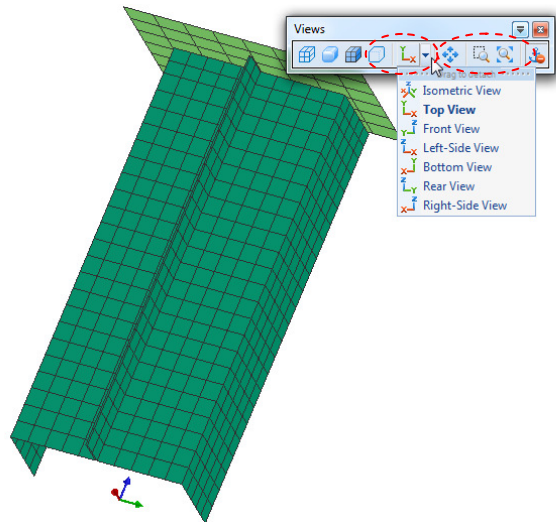


Viewing, positioning and centering of the model

With **View > Window_Background Colour** the background colour can be changed to white (if preferred).

Use options 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.



Examining the assigned entities

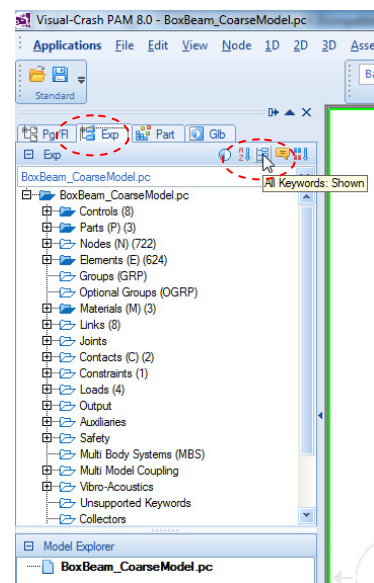
Most entities are most easily examined from the explorer panel (on the left side).

The first option indicated in the adjacent figure opens the explorer panel.

The second option indicated toggles between:

1. All specified options in the model are shown.
2. All possible options in PAM-CRASH are shown (this allows an alternative route to define options).

It is more convenient to view only specified options. Then simply click on the entity you want to examine and use the right mouse key to activate the **Edit** option. This will open the entity panel. The following examines all entities assigned to the coarse Box Beam model.



Tutorial 7: Axial crushing of a spotwelded box beam structure

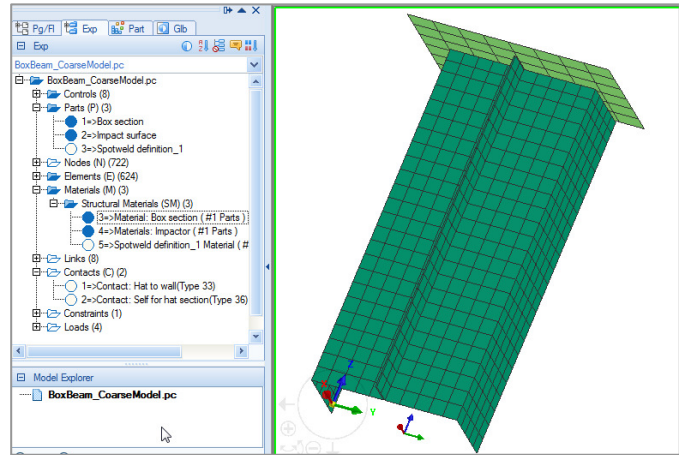
Parts and Materials

First take a look at the Parts and associated Materials.

In the explorer panel under Parts [+] and Materials [+] expand the lists. It will be seen there are 3 parts;

1. The steel box section
2. The impact surface
3. The spotwelds

Each part has its own associated material. Examine the different parts and material properties.

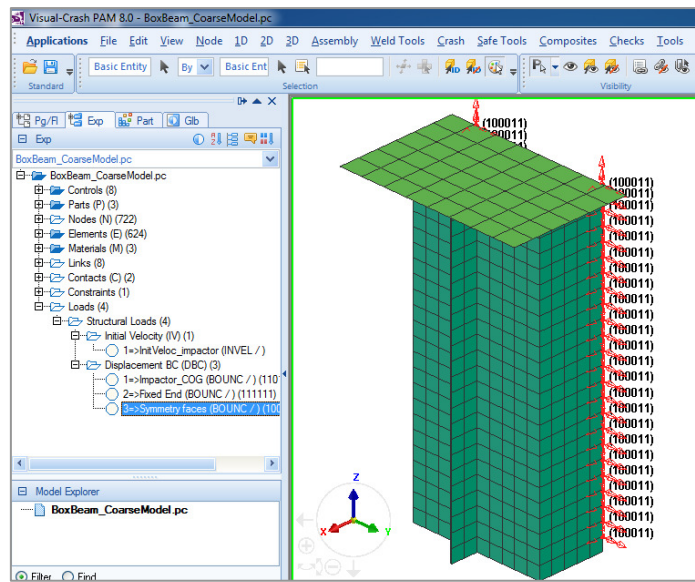


Load(s) (Initial velocities and boundary conditions)

Next, the loads list is expanded to show there is one initial velocity definition for the wall, and 3 displacement boundary conditions. In this study it has been chosen to fix the box section at its base and impact it with the wall (N.B. it is often the other way around!).

The wall has been defined as a rigid body and its COG node (=1000) has been assigned boundary conditions, a mass (see later) and an initial impact velocity.

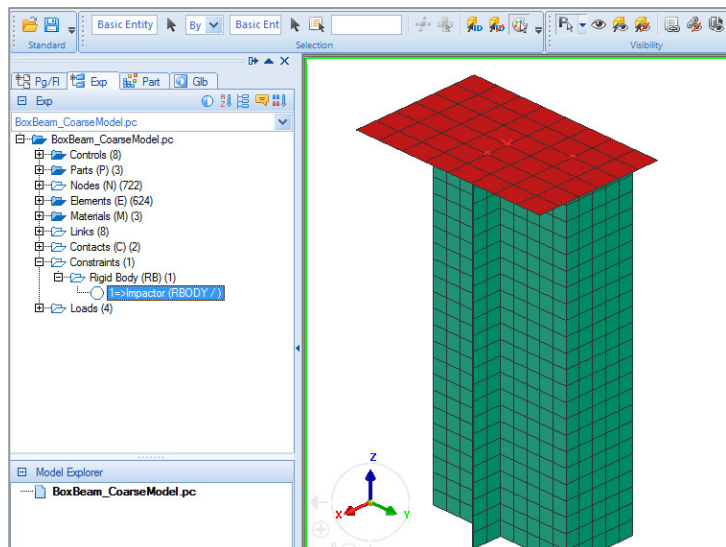
Examine the different boundary conditions. These allow z-direction motion of the wall, fully fixed free end of the box section and impose symmetry conditions (100011) for the box section symmetry plane.



Contact(s)

Two contacts are defined:

1. A type 33 symmetric contact is used between the box section (slave) and wall (master); it is not important which is slave and which is master. This contact type allows contact force between the two parts to be monitored.
2. The hat section undergoes complex folding and self-contact type 36 is most suitable. The wall could also have been included in this definition, but then the section-wall contact time history would not be available.



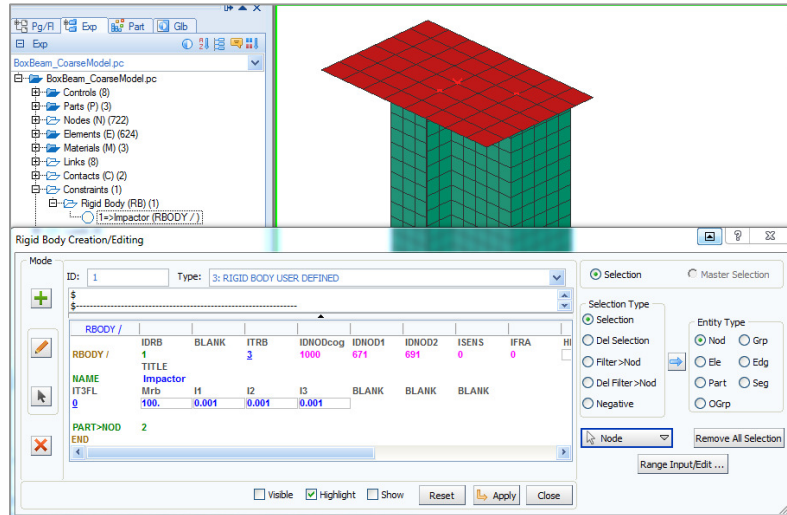
Tutorial 7: Axial crushing of a spotwelded box beam structure

Rigid Body

For convenience the impact wall has been defined as a Type 3 rigid body.

This rigid body type 3 allows mass and inertias to be assigned to the specified COG node. The assigned wall mass is 100kg; the inertias, in this case, have no real meaning due to the imposed boundary conditions.

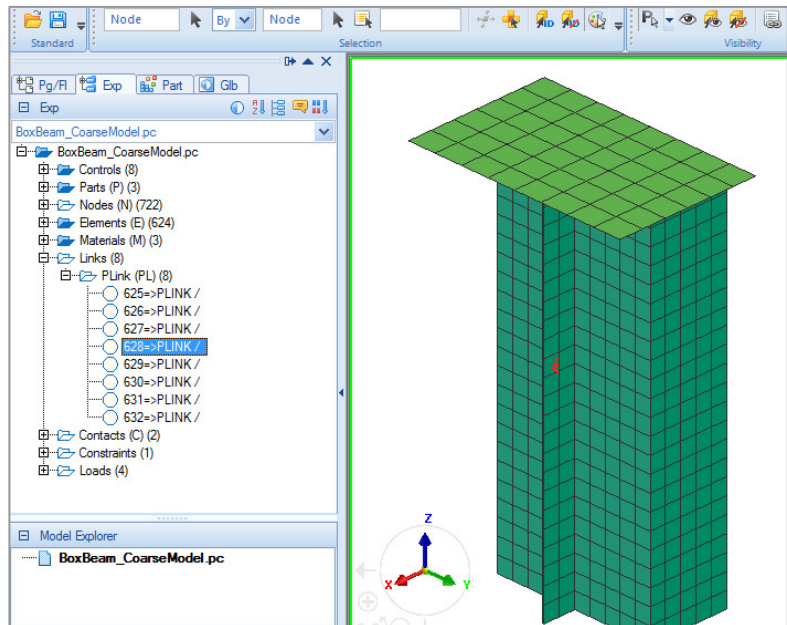
Remark: The initial velocity and constraints on a rigid body must be applied to either all nodes, or to the COG node. In this case they are assigned to the COG node.



Spotwelds (PLINKS)

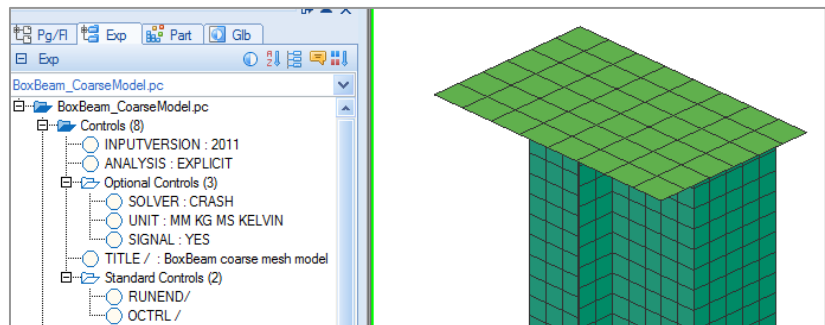
Useful tools are available in **VCP** to define spotwelds. Spotwelds can range from simple rigid bodies that tie nodes together to various mesh independent joints such as PLINKS (which are described in Tutorial 11).

In **Crash > Links > PLINK**, simple individual PLINK's are defined; it is necessary to specify an attachment node (or element) and then associate this to a Part (IPART) which contains information such as the search radius to locate the attachment point(s). This Part is also associated to a Material for mechanical properties and possible failure data via an auxiliary Rupture Model.



Controls

Control information for the analysis, termination time, file outputs and analysis title are available for viewing (or changing) in **Controls > Pam Controls**, or via the explorer panel and editing.

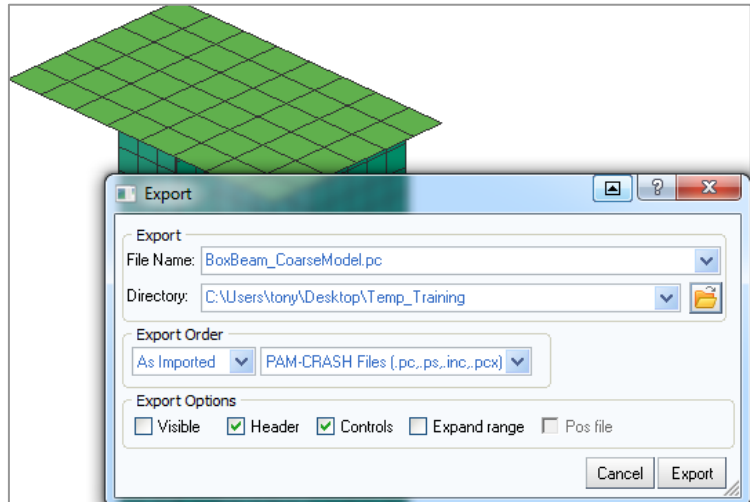


Tutorial 7: Axial crushing of a spotwelded box beam structure

Saving the dataset (.pc file)

Click on **File > Export** and save the model as a PAM-CRASH dataset:

- Save the model in a suitable directory/name; e.g.
BoxBeam_CoarseModel.pc
- OR, if no changes have been made, simply exit the Visual Crash PAM session without saving.



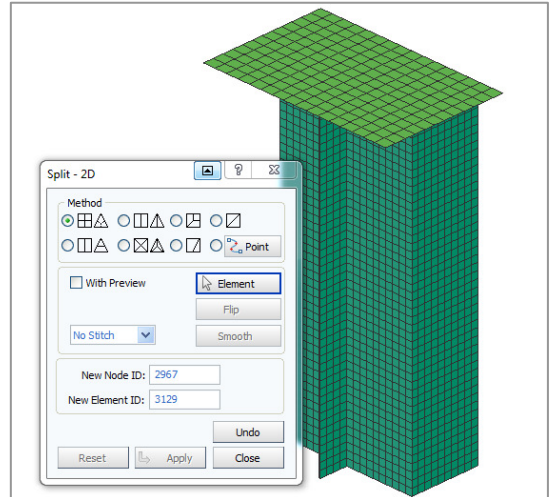
Part 2: Preparing the refined box beam models

Modifications are now made to the **BoxBeam_CoarseModel.pc** model using the copied datasets; that is:

1. Use **BoxBeam_MediumModel.pc** for the first level refined mesh model
2. Use **BoxBeam_FineModel.pc** for the second level refined mesh model

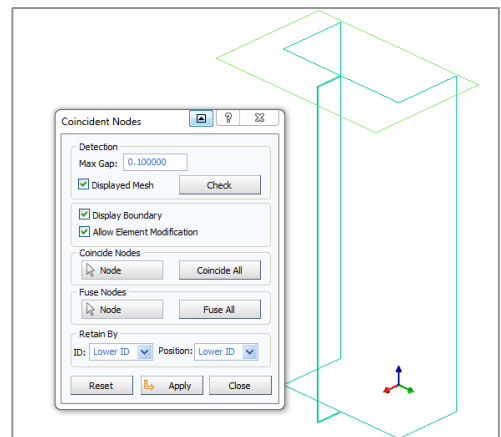
Procedure:

1. Start **Visual Mesh** and open either, **BoxBeam_MediumModel.pc** or **BoxBeam_FineModel.pc**.
2. Use the split option (**2D > Split**) to refine the mesh. For the medium meshed model perform this refinement once; for the fine meshed model do this twice.
3. Change the analysis title in the **Pam Controls**; e.g. to "BoxBeam medium mesh model" for the medium mesh model and a similar title for the fine mesh model.



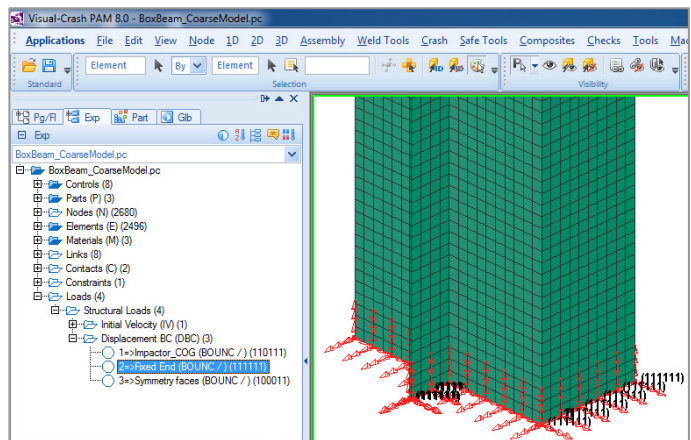
4. Whenever using the mesh splitting option (and as a general check) it is advisable to validate the mesh is properly joined without gaps (due to free or coincident nodes).

Use the **Checks > Coincident Nodes** option with a suitable gap size. The adjacent view shows that only proper edges are present and there are no internal gaps/edges. This option can fuse (merge) free nodes if required.



5. The splitting operation has introduced a new mesh with new nodes. It will be seen in the model that any existing nodal conditions, such as boundary conditions, only apply to the 'previous' coarse mesh nodes (not to the new ones).

The new nodes should also have appropriate conditions. This is easily done by updating the selection (e.g. by removing the old nodal list and making a new list by dragging a box over the required edge).



6. Export the dataset with the same name; either, **BoxBeam_MediumModel.pc** or **BoxBeam_FineModel.pc**.

Part 3: Running the models and investigating results

The Pam-Crash datasets are run. Then open the time history results file; e.g. file **BoxBeam_CoarseMesh_RESULT.erfh5** in a new **Visual Viewer** session.

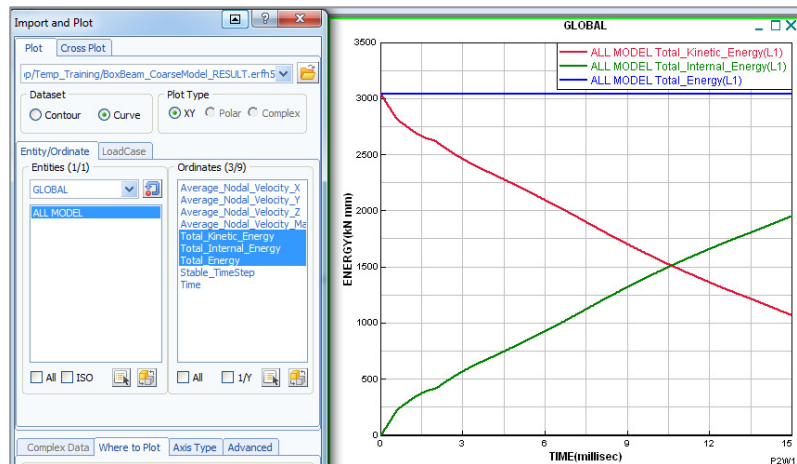
For this type of problem which involves a free body (the wall) hitting a deformable elasto-plastic structure it is possible to check for solution stability. In this case the Initial (Kinetic) energy of the impactor is converted to Internal (Plastic) energy in the structure. The Total energy of the system (=Kinetic + Internal) during the crash should be constant since no energy is added or removed from the system. Any numerical instabilities will add spurious energy to the system which will show as an increase in Total energy.

Global energy plots and numerical stability

Plot the three global time histories for External, Internal and Total Energy. The tab for curve superposition should be activated so that all three curves appear on one plot.

Notice the Total energy is constant indicating a stable solution.

Remark: A constant total energy does not necessarily mean the solution is good! Other factors, especially the deformation mechanisms must be checked carefully. Also, as demonstrated below, effects such as mesh density can greatly influence the folding behaviour.

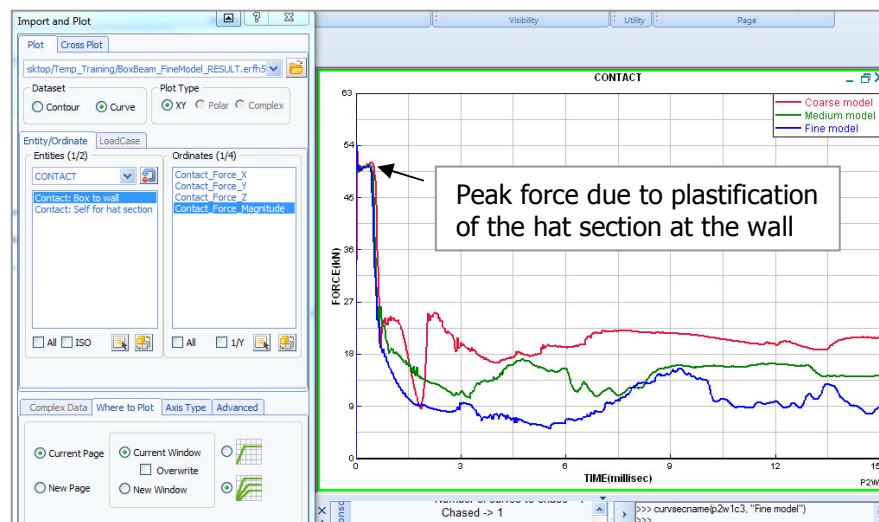


Impact force time histories

The adjacent plot shows contact force (Contact_Force_Magnitude) at the wall for the three simulations.

Plot the first time history (e.g. for the coarse model) using the options shown. Then use the **Chase Curve(s)** option introduced in Tutorial 4 to get the other curves.

There is clearly a large difference in the impact force time histories. This is due to inability of the coarse mesh models to



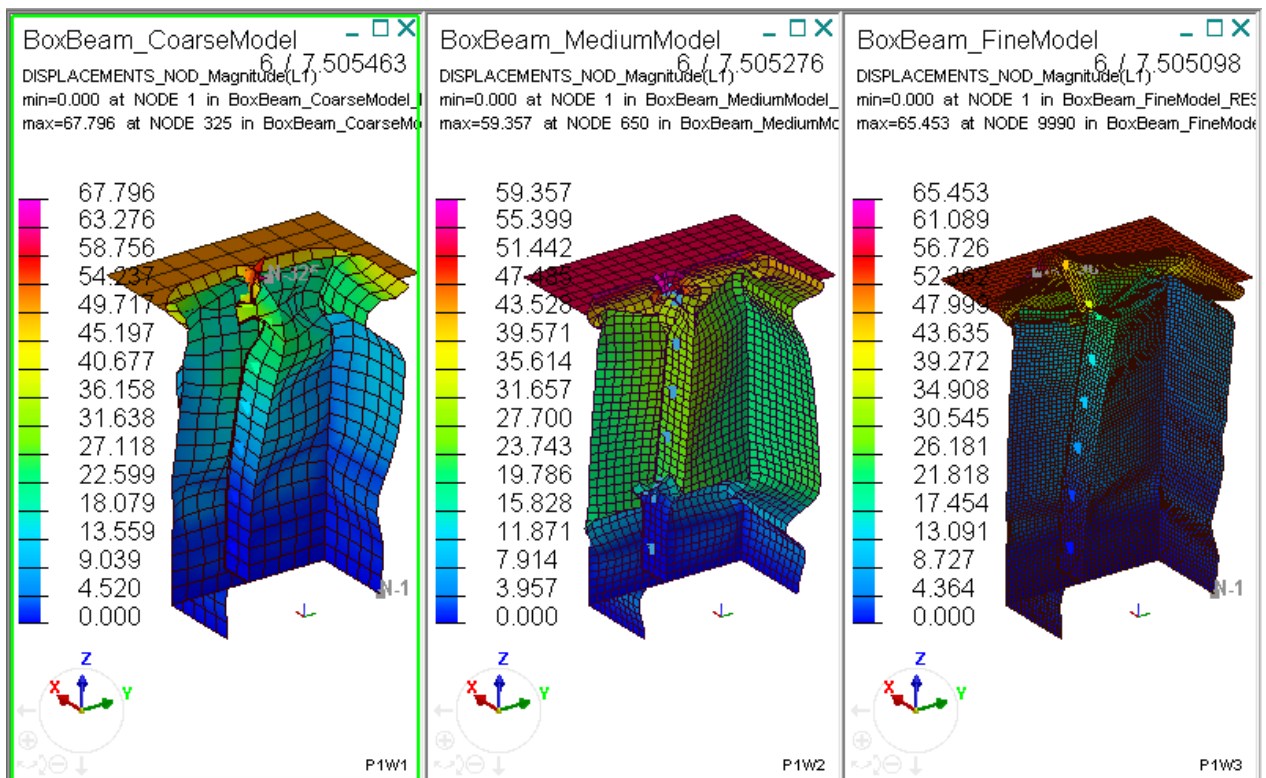
Tutorial 7: Axial crushing of a spotwelded box beam structure

capture the true folding mechanisms and will lead to higher (unrealistic) energy absorption. Sufficiently fine meshes will converge to the true folding. An inspection of the deformation plots, shown below, is helpful to understand these trends.

Deformation plots

The .erfh5 files for the three models are loaded into Visual Viewer with three windows opened (use option **Window>Page_Layout**). The three plots shown below compare deformations at 7.5msec. The coarse mesh model has difficulty to capture the folding; however, the medium and fine mesh results are probably realistic, but show the dangers that poor design of the spotwelds (insufficient number or inappropriate location) can have on folding. The spacing of the spotwelds is not in harmony with the natural folding (wavelength) of the section and has led to blocking and secondary folding at the rear of the section.

There are analytical models to determine folding wavelength of the section, or trial and error with FE simulation models could be used. An improved distribution of spotwelds would give stable progressive crushing from the front and help prevent erratic crushing processes. Furthermore, poorly placed spotwelds are likely to be highly loaded and could fail which would have serious effect on energy absorption.



The following studies try to improve stability of the crushing behaviour:

Part 4 uses more spotwelds.

Part 5 increases thickness of the section.

In both cases only the medium meshed model is considered to restrict CPU time for teaching purposes.

Part 4: Analysis with more spotwelds

Copy

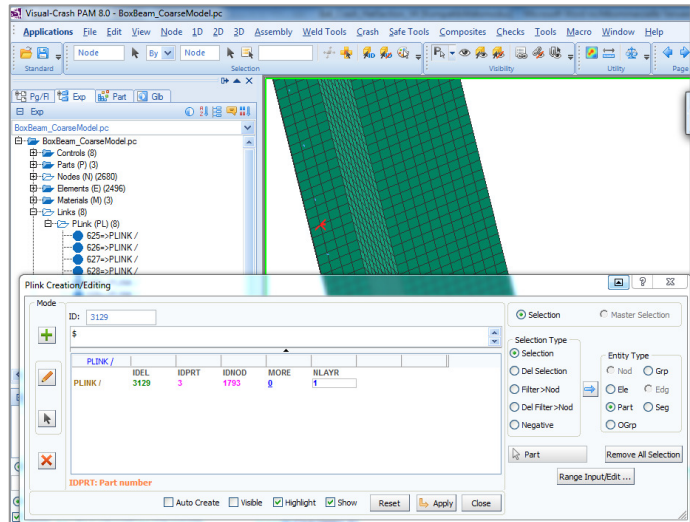
BoxBeam_MediumModel.pc

to

**BoxBeam_MediumModel_
MoreSpotWelds.pc**

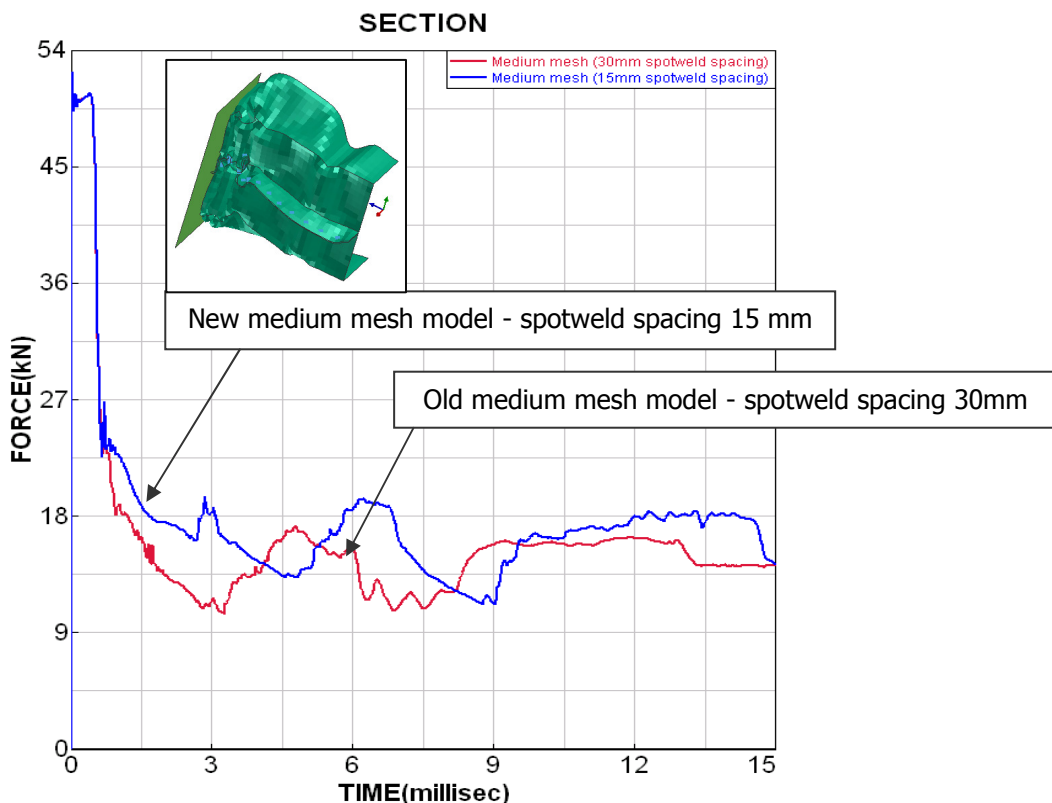
Start **Visual Crash PAM** and open the new dataset. More spotweld (PLINK) elements are added as follows.

Use the **Crash > Links > Plink** option and identify new connection nodes; these are then associated to the existing Part (=3) for Plink elements. Add one new Plink elements between each pair of existing ones, change the analysis title and export (overwrite) the modified dataset:



BoxBeam_MediumModel_MoreSpotWelds.pc

Run the new model with PAM-CRASH. The contact force time histories for the new medium meshed model (spotweld spacing 15mm) is compared below with the previous model (spotweld spacing 30mm). The impact force time histories and deformation plots do show some improvement; however, the collapse is still slightly unstable. The following (last) exercise increases the box beam sheet thickness to 1.75 and 2.5mm to try and improve section stability.



Part 5: Increasing the box beam sheet thickness

Use the medium model with more spotwelds model and copy these to two new datasets,

e.g. **BoxBeam_MediumModel_MoreSpotWeld_Thick175.pc**

BoxBeam_MediumModel_MoreSpotWeld_Thick250.pc

Start Visual Crash PAM and the sheet thickness from 1mm to 1.75mm or 2.5mm in these datasets; this is defined in the Parts cards. The thicker sheet will also absorb more energy so increase the impactor mass (impactor rigid body from 100kg to 200 kg) and also increase the run time (control parameter RUNEND) from 15msec to 25msec. Export the modified datasets.

Run the new models and open the results file (.erfh5) with **Visual Viewer**. With increasing sheet thickness the deformations now shows a stable collapse and the wall impact force time history shows a typical cyclic form as the section collapses, lobes on the folds meet and the force increases again until the next cyclic collapse starts. The section plot shown below is produced via **Sections > Section Cut**.

