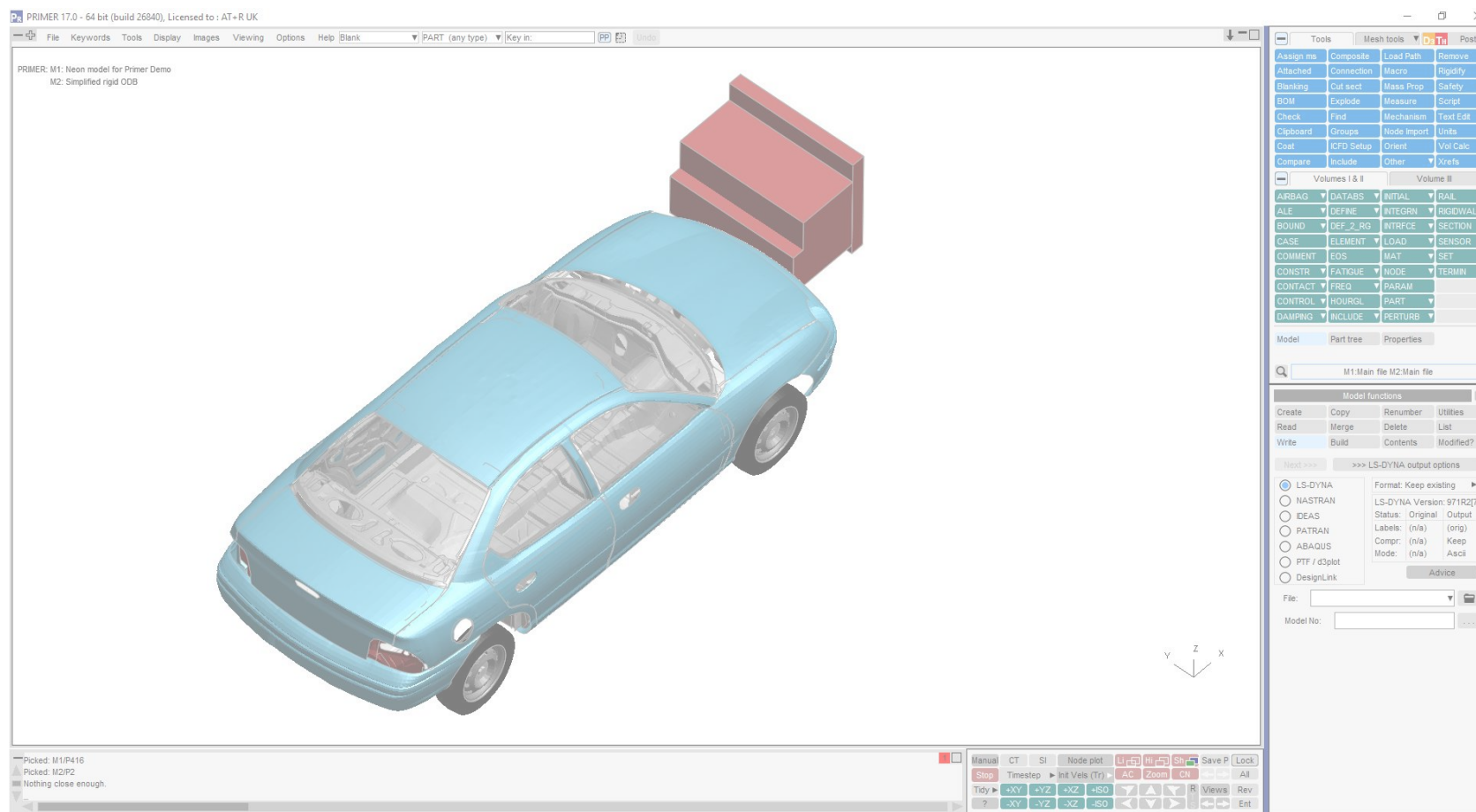


Oasys PRIMER

A training course for new users

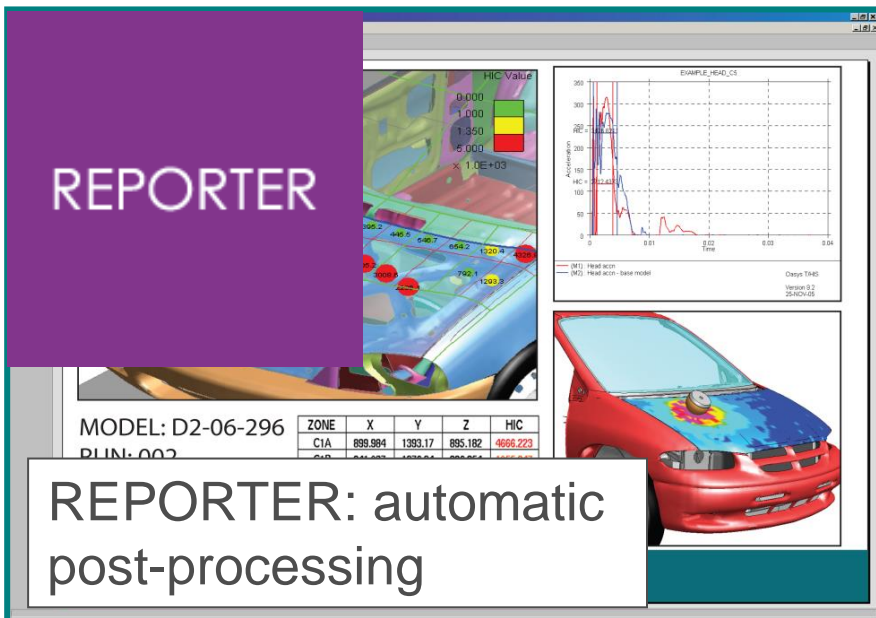
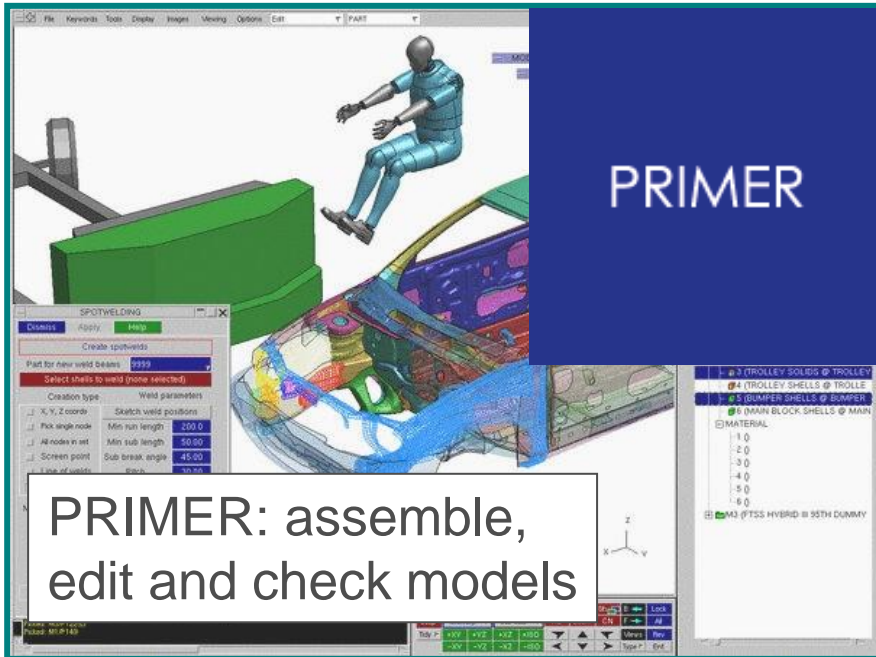


Course Content

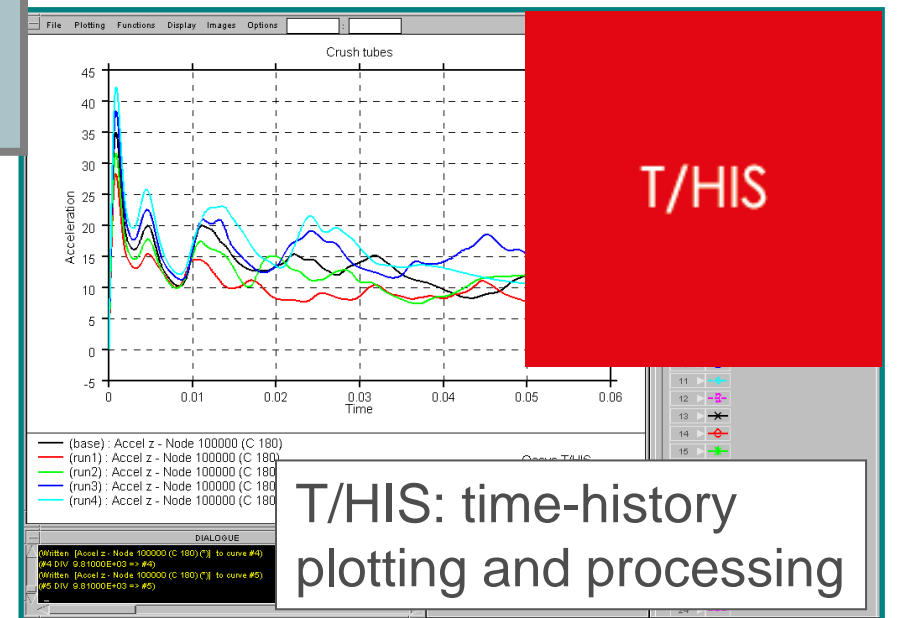
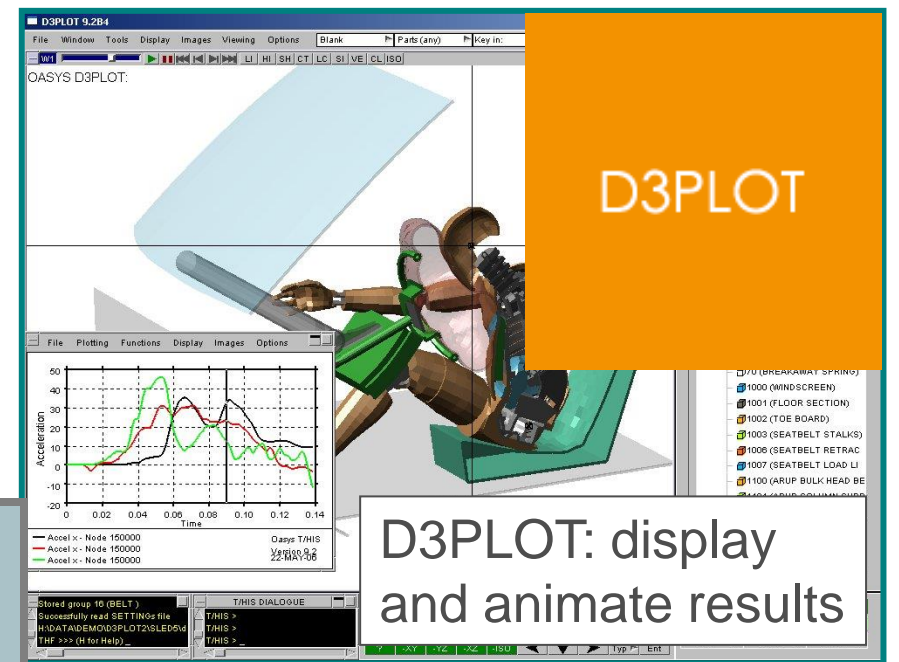
- Introductions
- Purpose of Oasys PRIMER
- User Interface and controlling the display
- Some basic functions – editing, deleting, etc
- Model Checking
- Other methods of checking a model – contouring etc
- Working with multiple models, model merge
- Replace Part
- Working with Include files
- Contact penetration checking and fixing
- Spotwelding/connections
- Functions not covered in this course - demo
- Q&A / Discussion

Before starting this course

- This training course assumes that certain Preferences have been set for PRIMER. To do this either:
 - (a) Put the file named *oa_pref* that is provided with this course in your log-in directory
 - Or (b) use the Preferences editor within PRIMER, checking and changing the primer preferences shown in the *oa_pref* file provided (the preferences can also be edited from within PRIMER).
 - Or (c) hand-edit your existing *oa_pref* file, pasting in the *primer lines from the file provided
- If this course is being taught to others, ensure the “students” have the same *oa_pref* settings as the “teacher”

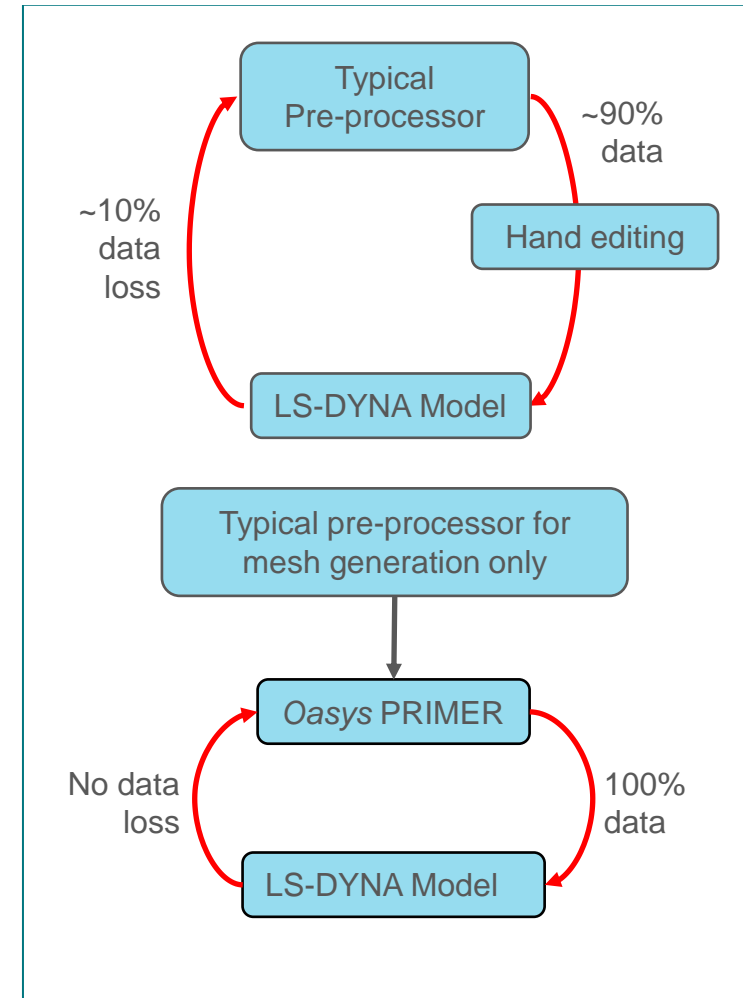


*Oasys software
for LS-DYNA*

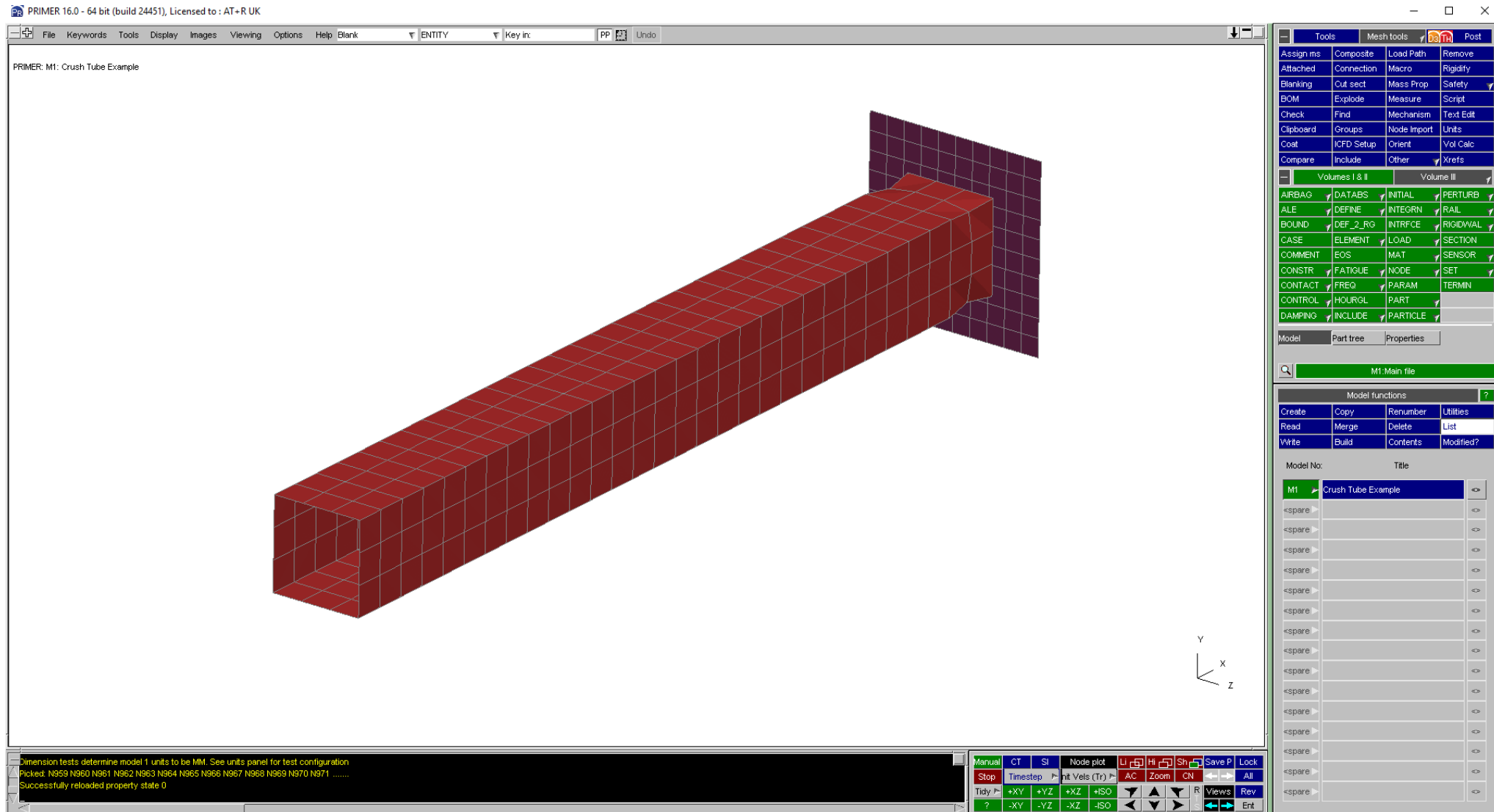


Purpose of Oasys PRIMER

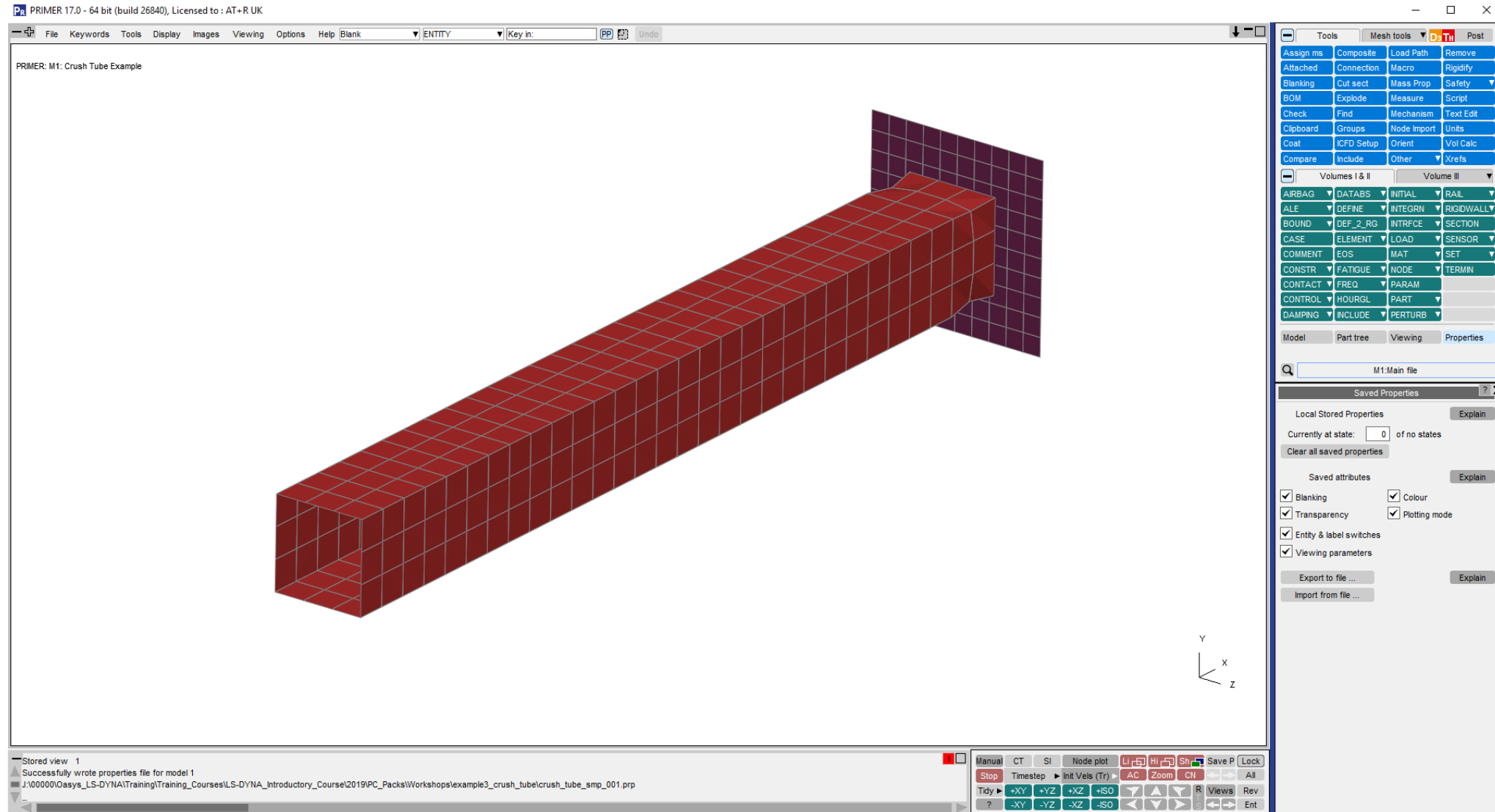
- Oasys PRIMER is a specialized pre-processor used only for LS-DYNA models.
- PRIMER's modeling tools and comprehensive error checking can reduce preparation and debugging time.
- PRIMER is fully compatible with the latest version of LS-DYNA, so no information is lost when reading, or writing out model data.
- PRIMER's model management tools allow the CAE team to work in parallel on different components of the same model.
- Because of PRIMER's robustness and depth of understanding of the LS-DYNA data, users trust PRIMER to work on their complex models without corrupting or losing any data.



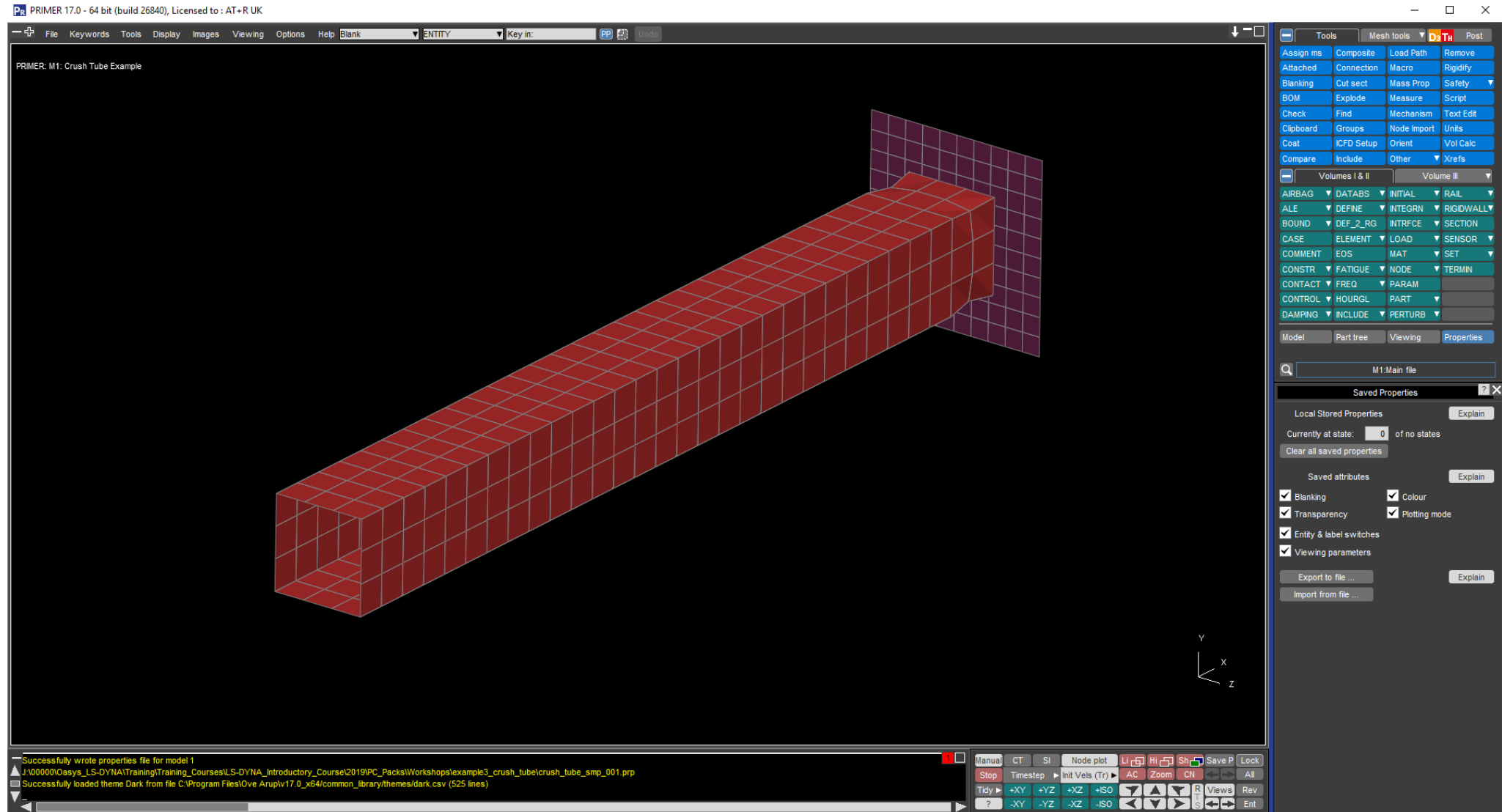
User Interface before...



User Interface after...

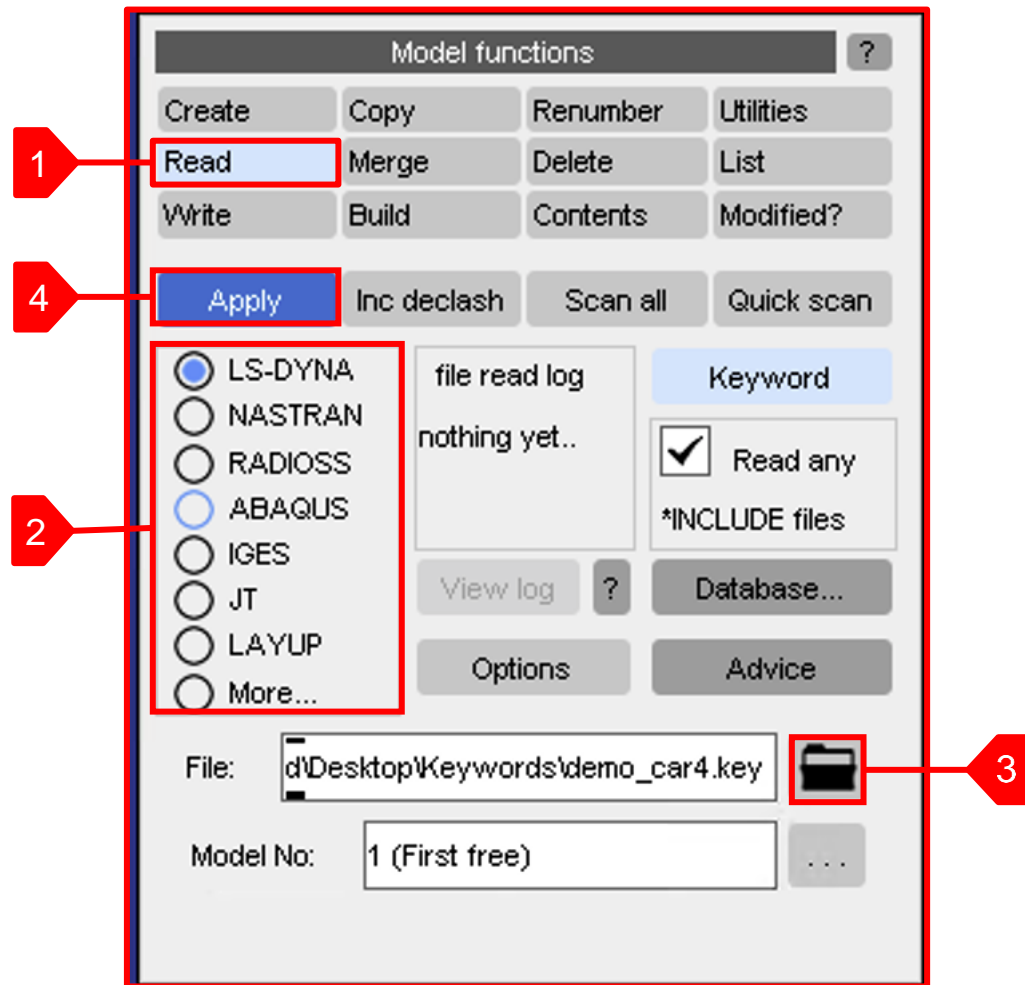


...and now



User Interface and Controlling The Display

Exercise: Read an LS-DYNA Model



Reading in a Model:

1. Start PRIMER. If the menu in the bottom right corner is not already shown as here, click **Model** and **Read**.
2. Leave the file type set to LS-DYNA.
3. Using the file browser, select the file **demo_car4.key**.
4. Click **Apply**.
5. Watch the messages in the dialog box at bottom-left of the screen.

General Layout

Top menus

Allows access to basic options, keywords and tools, in a drop-down menu format.

Quick-Pick Control

Controls the mouse action when applied within the graphics area.

Tools

Provides access to PRIMER specific functions.

Graphics Area

Area within which graphics are drawn.

Keywords

This provides access to the Keywords that are supported and can be edited by PRIMER.

Menu tabs

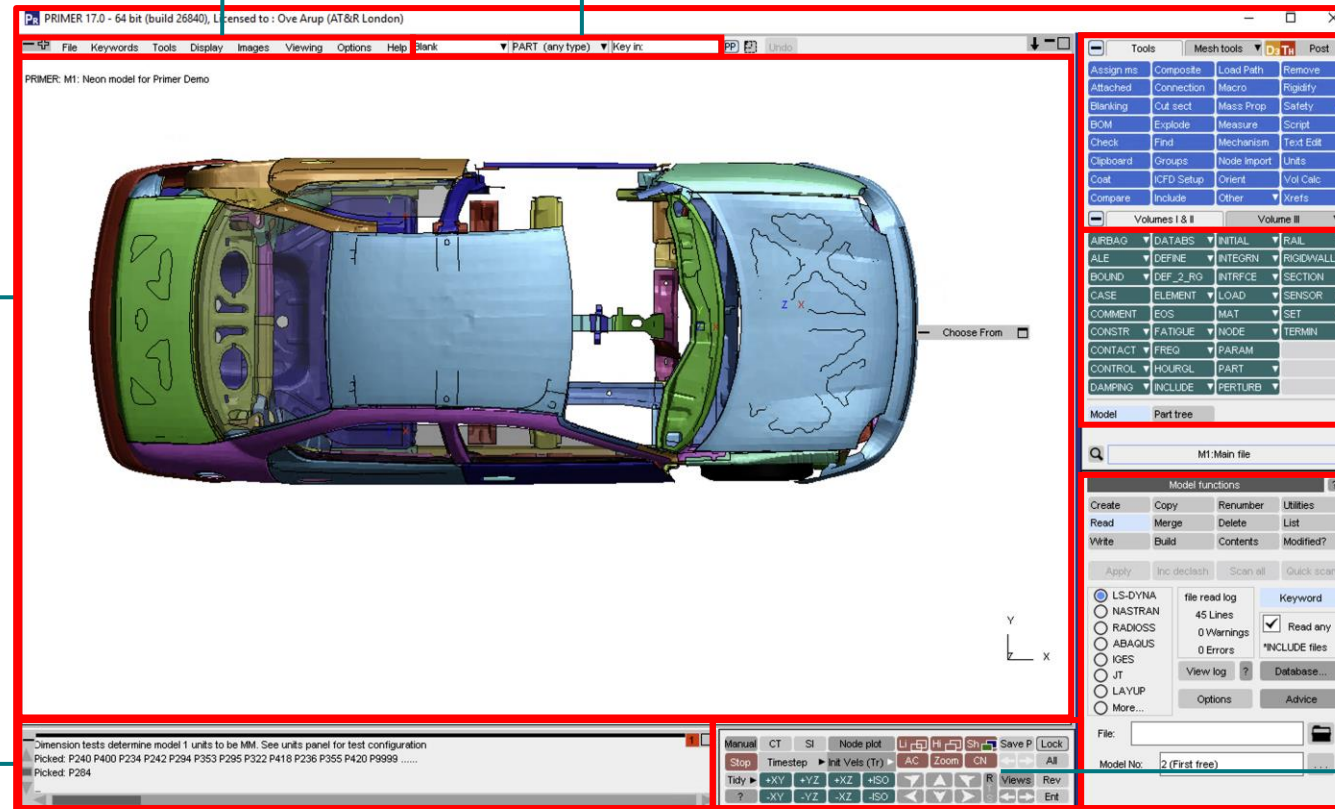
These control which option is displayed in the current menu panel. Model and Part Tree will always be available in addition to selected options.

Viewing & Drawing Commands

Provides all aspects of view control: direction, perspective, scale, etc. Contains the drawing commands and their settings.

Dialogue & List area

Area for command-line input and output, also acts a listing area for messages.

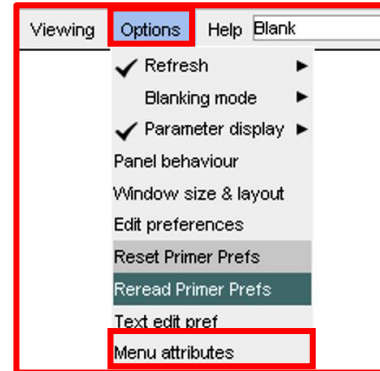


Setting Menu and Text Size

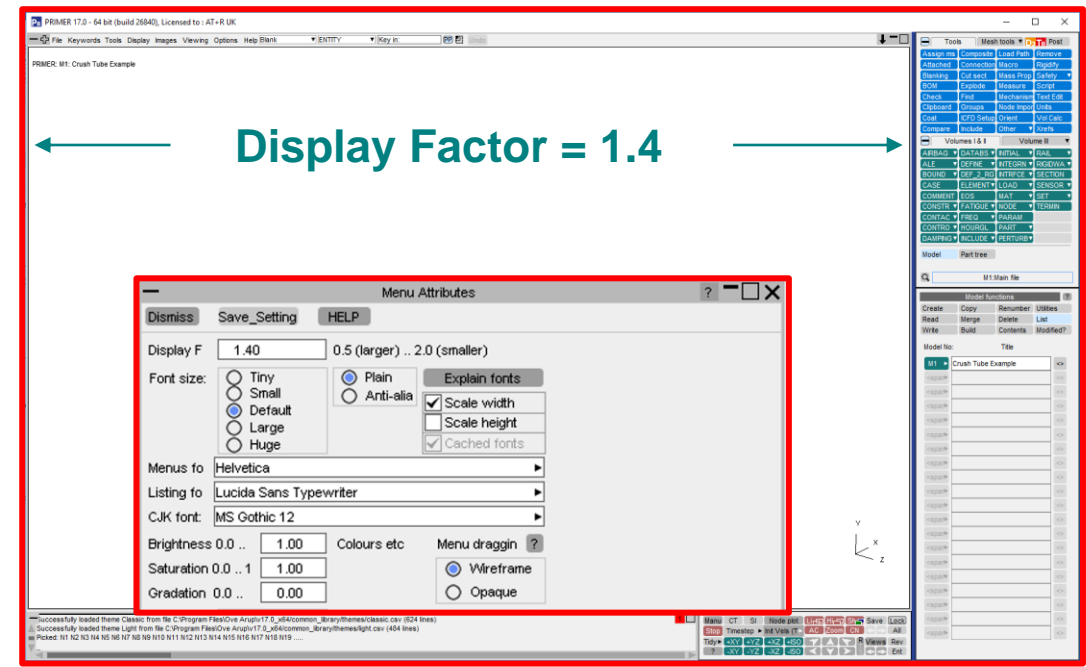
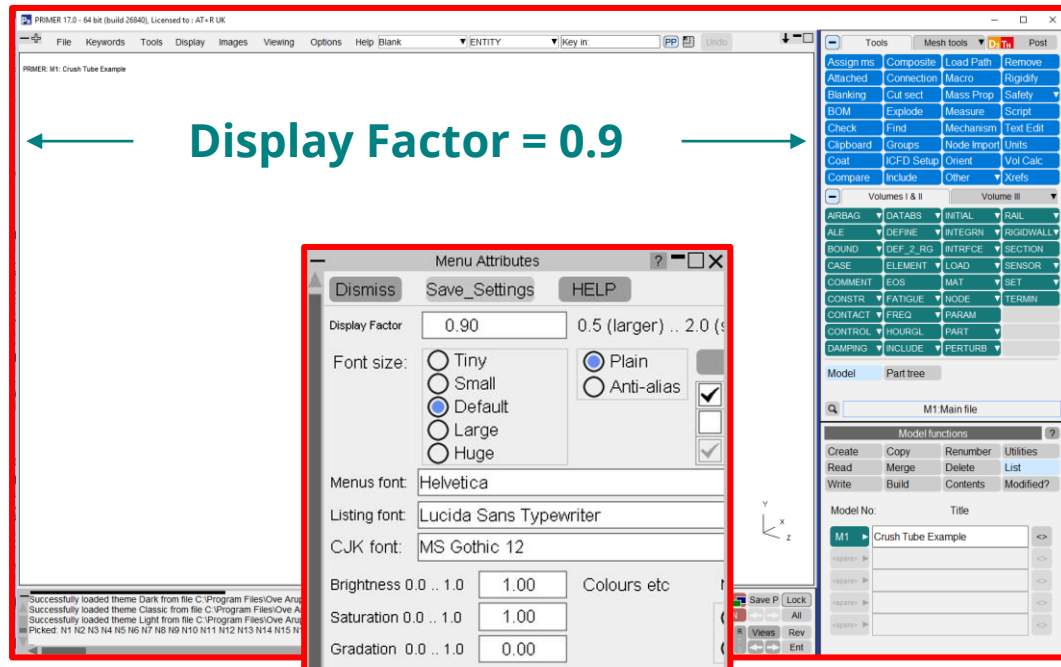
Controlling image/menu/text size:

Depending on your screen size and eyesight, you may prefer the menus to be larger, or to take up less room.

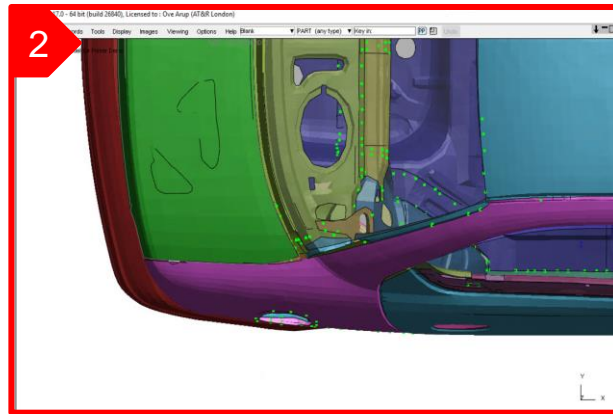
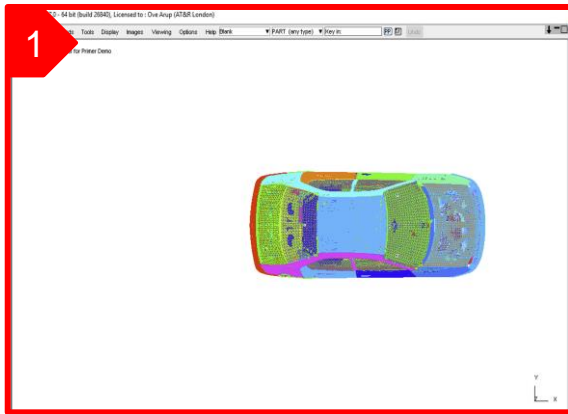
Use Options=>Menu attributes to set the Display Factor.



- Note: if Display Factor is made too large, there will not be room for text – you will see blank buttons.

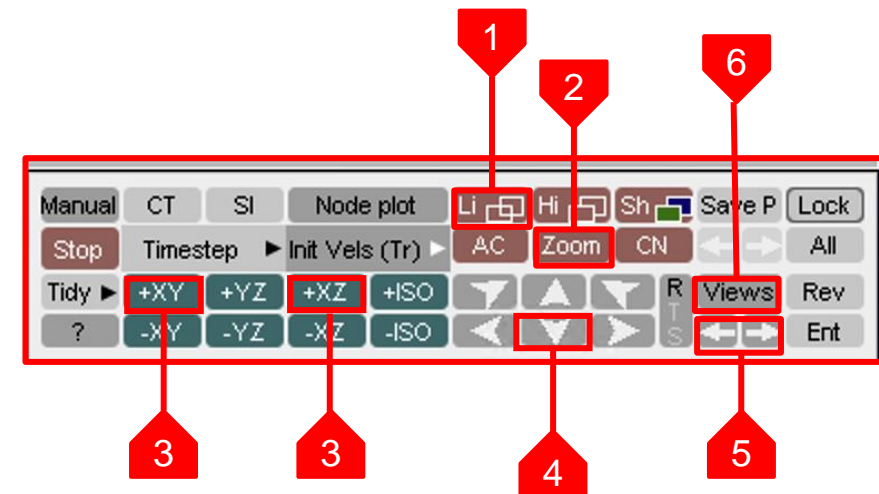
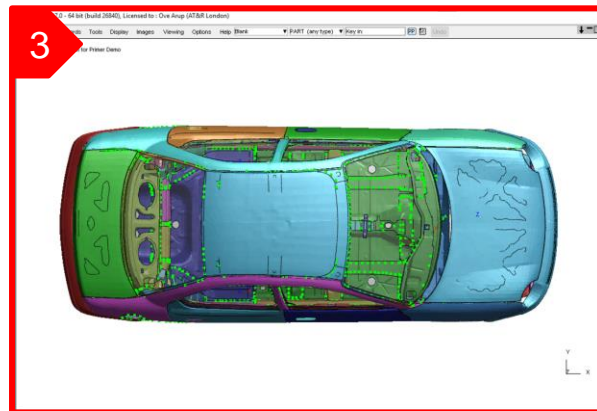
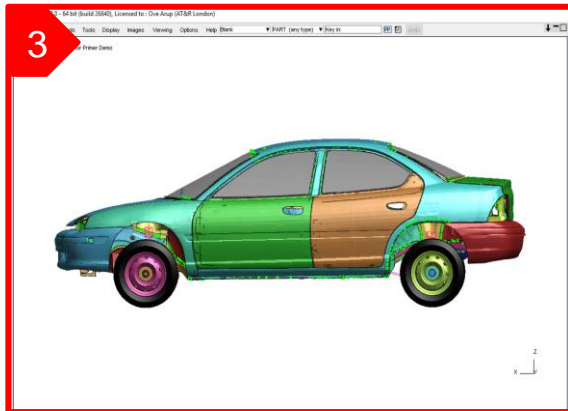


Display Tools



Changing the Display

1. Try the different plotting modes: **Line**, **Hidden** or **Shaded**.
2. Try **Zoom** then drag across an area. Auto scale (**AC**) the view.
3. Try the standard view buttons.
4. Try the Arrow buttons make small changes to the model orientation.
5. Try **Back** and **Forward**, to change to previous views.
6. **Views** allow views to be stored/retrieved

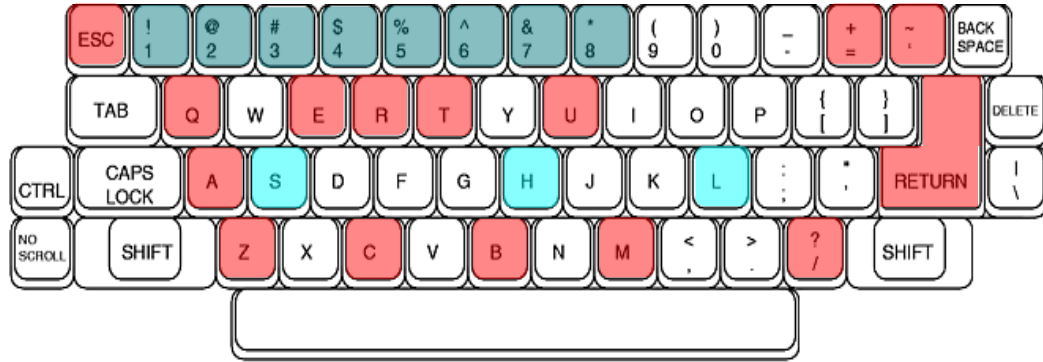


Dynamic Viewing

Dynamic Viewing

1. Hold down the left SHIFT key
2. Position the cursor near the centre of the graphics window, then drag the model using the left button (rotate about screen X/Y). Observe the cursor symbol and the way the model moves.
3. Now position the cursor near the edge of the graphics window, drag with left mouse button (while holding down left-SHIFT) – rotate about screen Z. Observe the cursor symbol and the way the model moves.
4. Try left-SHIFT + drag with middle mouse button (translate)
5. Try left-SHIFT + drag with right button (zoom)

Shortcut Keys



Shortcut Keys

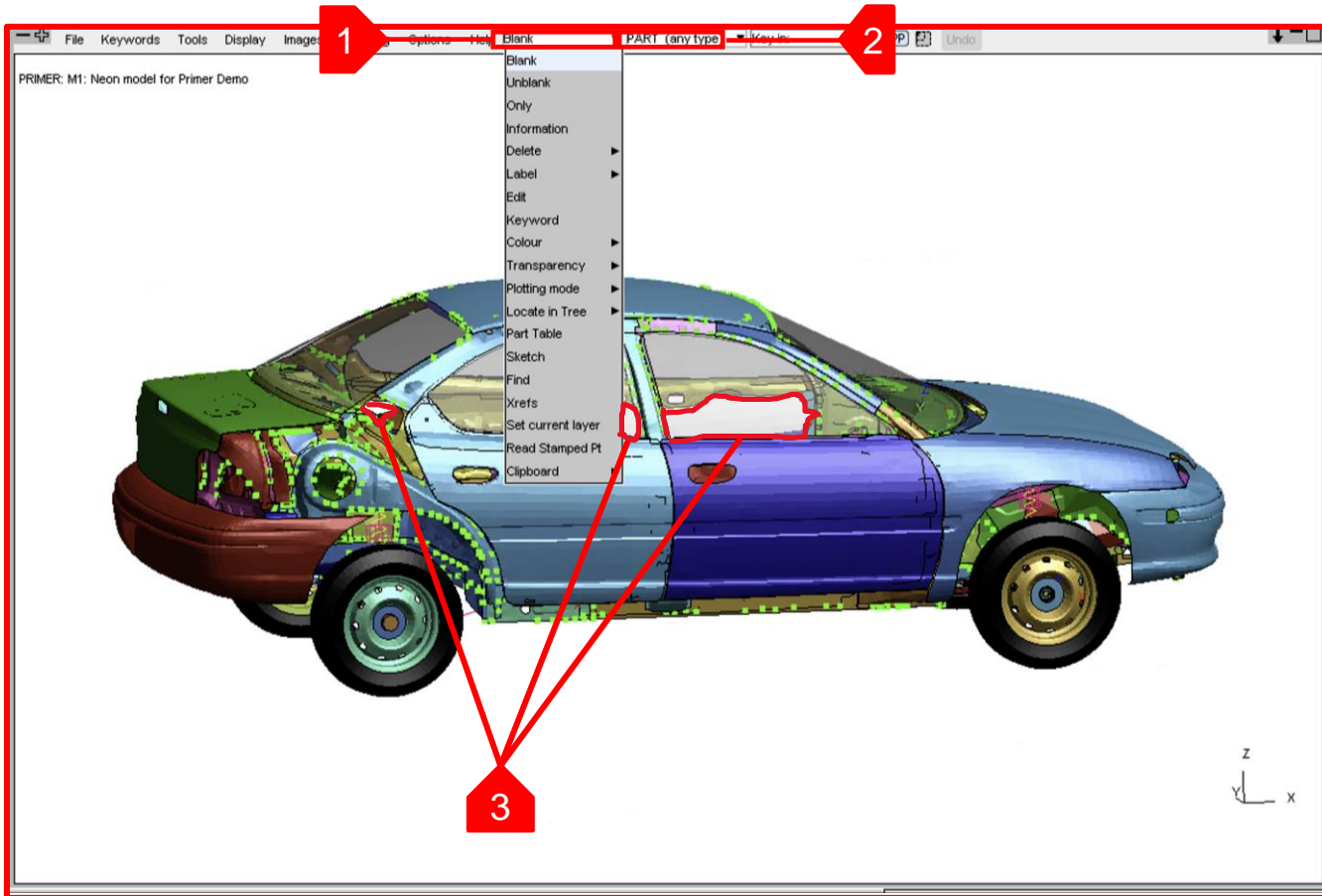
1. Try the shortcut keys:
L H S 1 2 3 4 5 6 7 8 + - Z
after Z, click 2 points across a rectangle to zoom in
2. Now press A to reset



List of Shortcut Keys

? = list of available shortcut keys (also Options=>Shortcuts)
ESC = dismiss the menu that the mouse is over .
RETURN (or middle mouse button) = APPLY.
1,2,3,4,5,6,7,8 = XY, XZ, etc standard views.
A = Autoscale.
B = blanking menu; **R** = reverse all blanking; **U** = unblank all.
E = entity visibility menu.
H, **L** and **S** = perform **H**idden line, **L**ine and **S**haded plots.
M = **m**easure node-to-node.
Q = return to **Q**uick-pick.
Z = **Z**oom (drag across rectangular area).
+ and **-** = zoom in and out.
T = **t**idy all floating menus; **C** = **c**lose all floating menus.
I = **i**conise all floating menus.

Quick-Pick

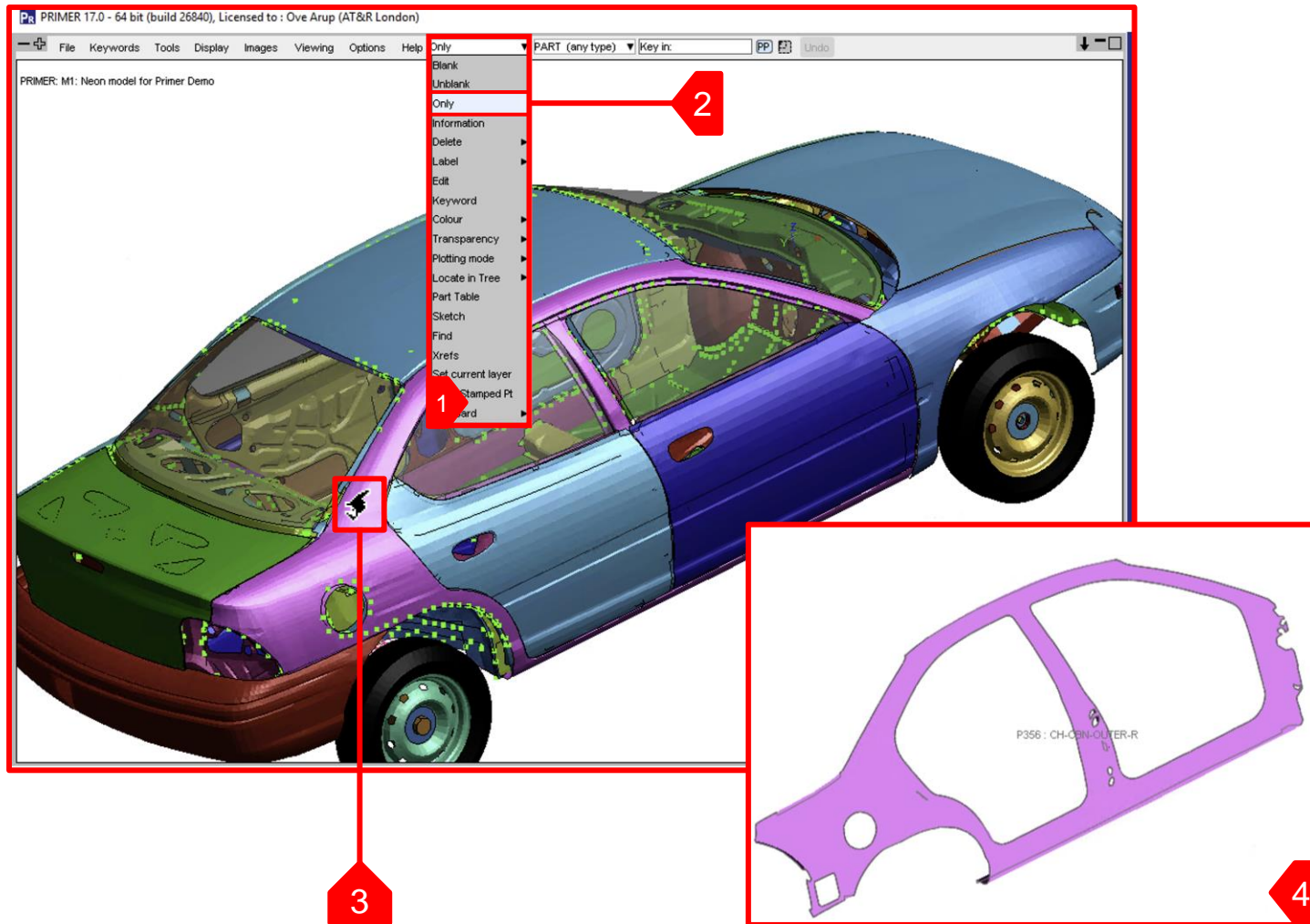


Quick-Pick

Quick-Pick is a powerful time saving tool giving access to commonly used functions.

1. This menu decides what action will happen when you click on an entity. Leave this set to Blank.
2. This menu decides what type of entity you will select – leave it set to Part
3. Click on any part – it disappears (is blanked).
4. Drag across an area – all the parts in that area are blanked.
5. Middle-click to undo previous quick-pick actions.

Quick-Pick

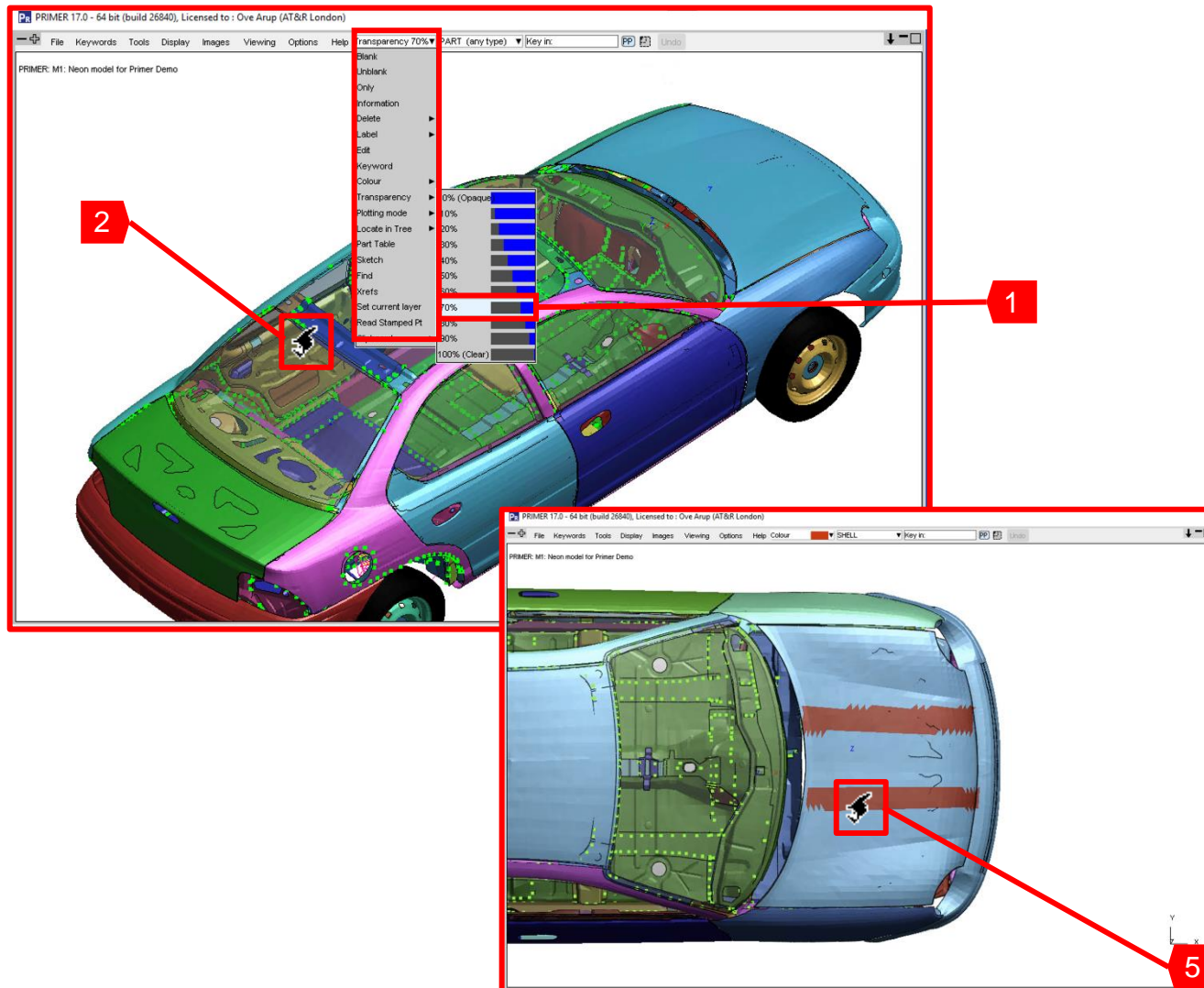


Quick-Pick

1. Right-click to activate menu.
2. Select the display **Only** function.
3. Click the part to be displayed.
4. All other parts are blanked.
5. Try shortcut **R** to **r**everse the blanking
6. Press **U** to **u**nblank all parts again

Other functions include blanking, item information, editing, colouring, transparency, plotting mode, locate in part tree, and create part table. View animation for further examples.

Quick-Pick

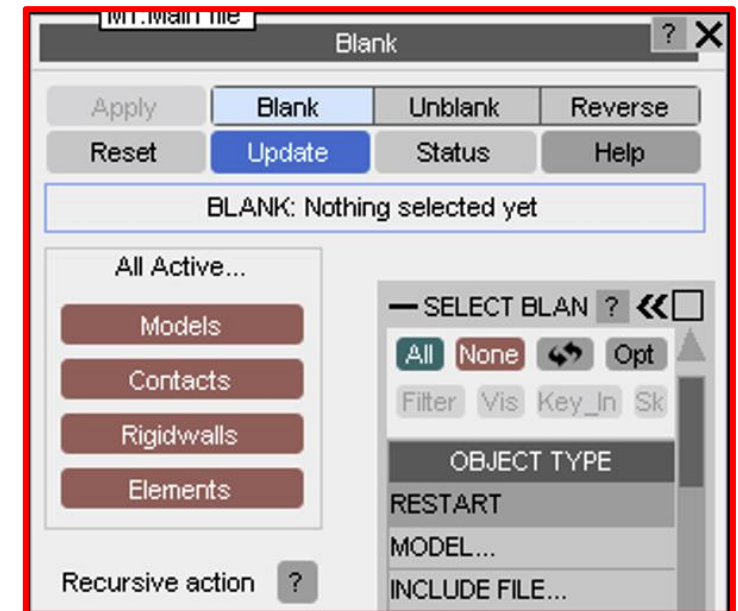
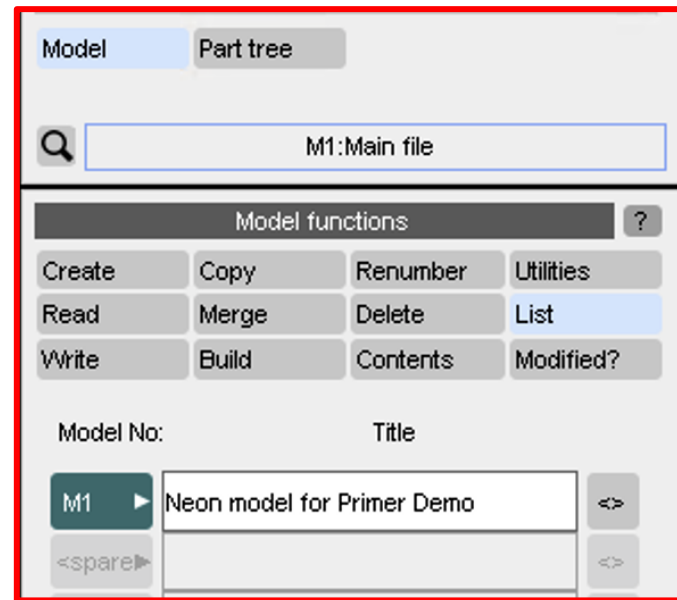


More Quick-Pick

Try other quick-pick combinations:

1. Change the action (left box) to **Transparency 70%**
2. Pick the roof to make it transparent
3. Middle-click to reset
4. Change the action to **colour – red** and entity (right box) to **Element – Shell**
5. Drag across an area to turn the shells red
6. Change the entity back to **Part**
7. Change action to **Information**, click on some parts
8. Change the quickpick control back to Blank Parts

Control of Visibility of Entities

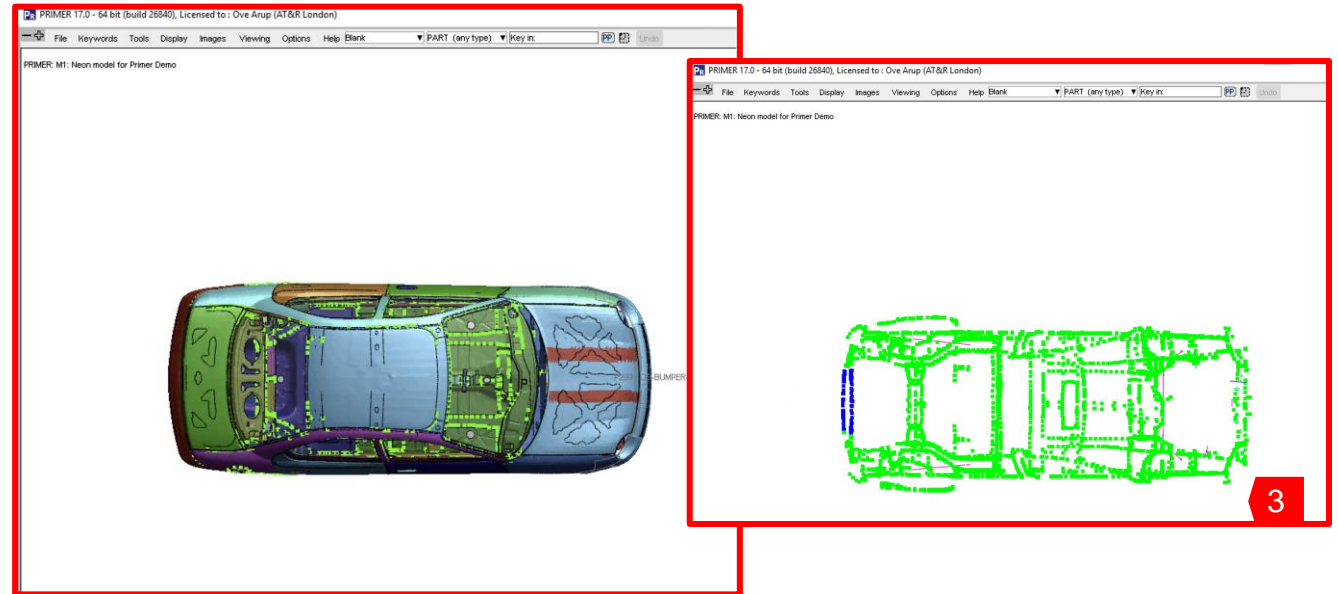
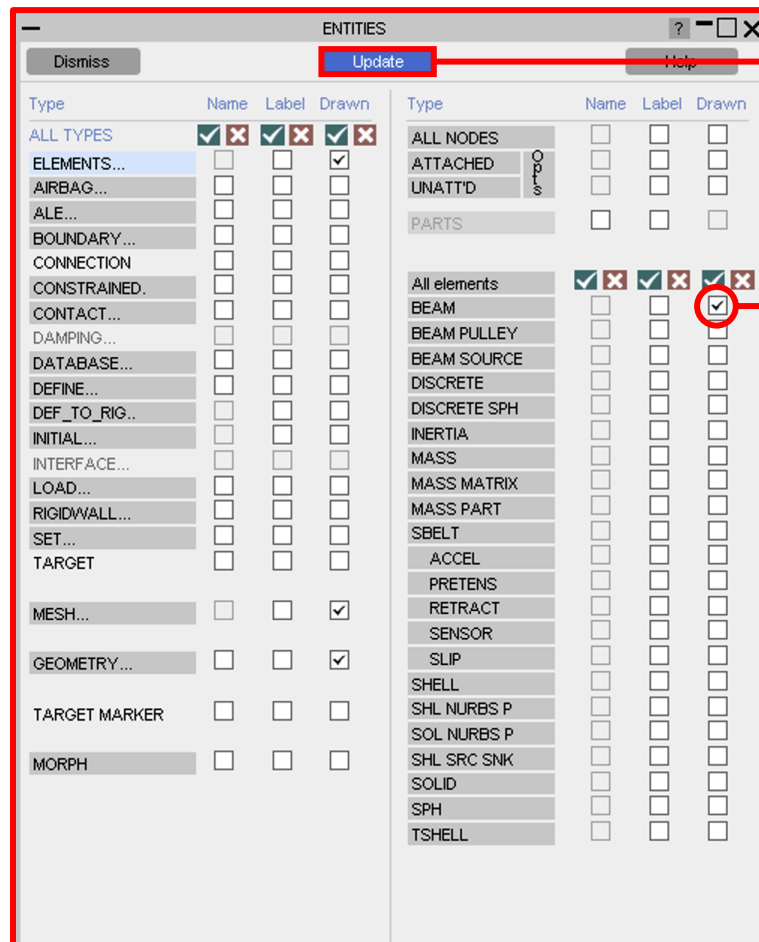


Control of Visibility of Entities

For an entity to be visible the following must all be true.

1. The relevant model must be active (green) in the model **List**.
2. The entity type must be turned on in **Entities** panel (to invoke this panel use shortcut **E**).
3. It must not be blanked.

Entities Displayed



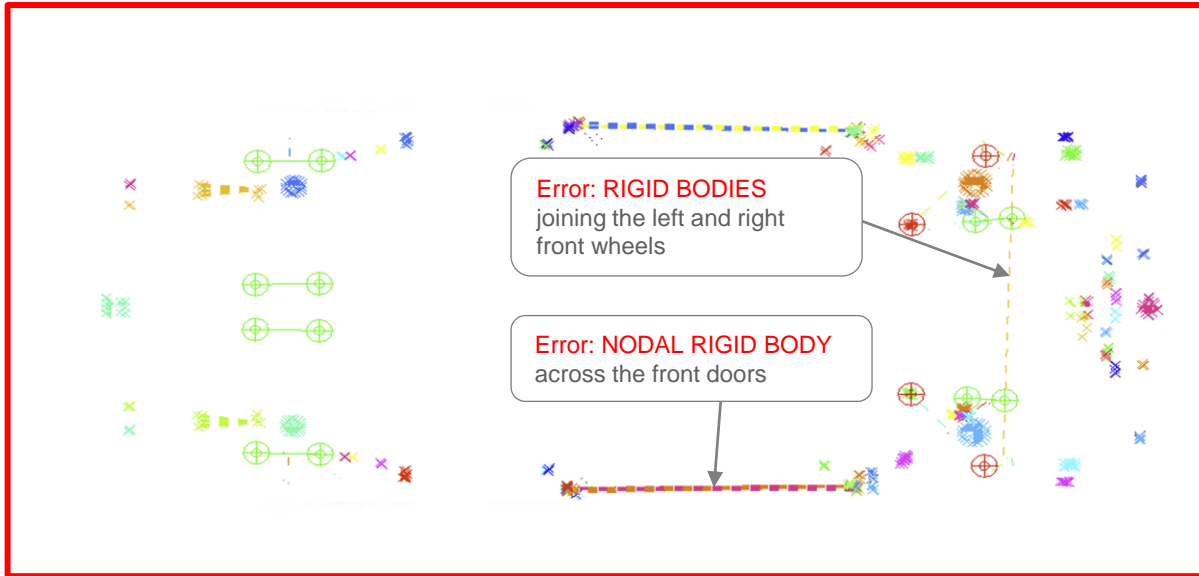
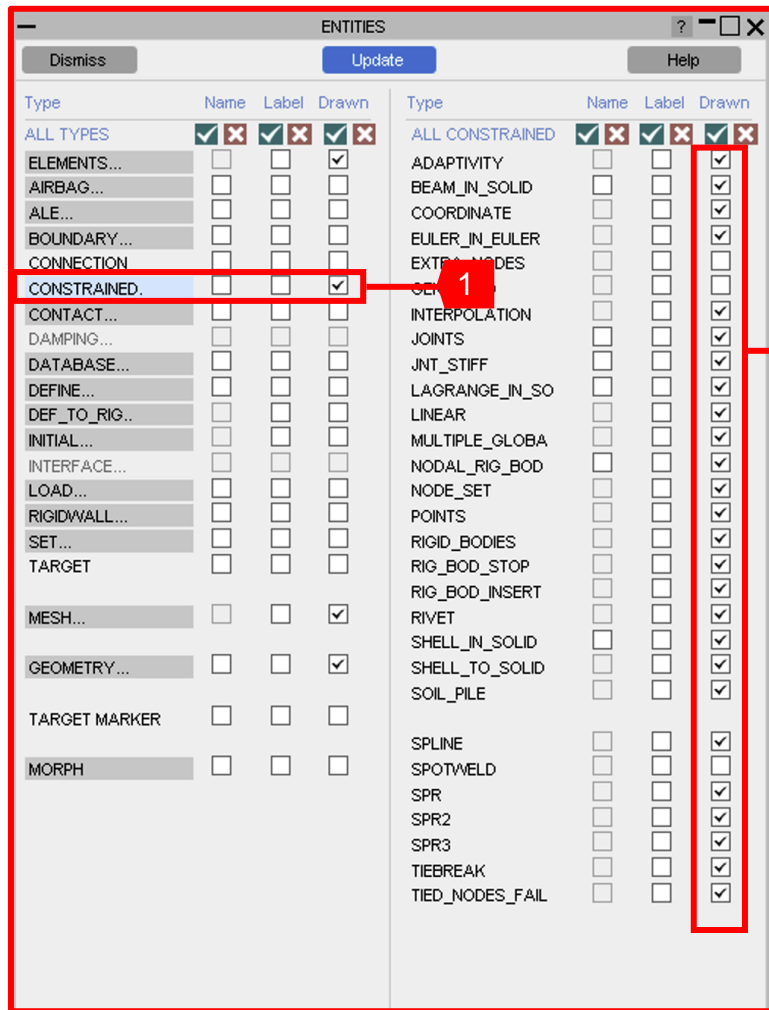
Entities Displayed

The entities panel controls which entities are currently available to be displayed in the graphics window.

1. Use shortcut **E** to bring up this menu.
2. Change the entities displayed by checking and un-checking the appropriate boxes – e.g. only beams ☒
3. Click **Update** or press **S** to display new view.

Entities can also be labelled and labelling options can be chosen.

Entities Displayed



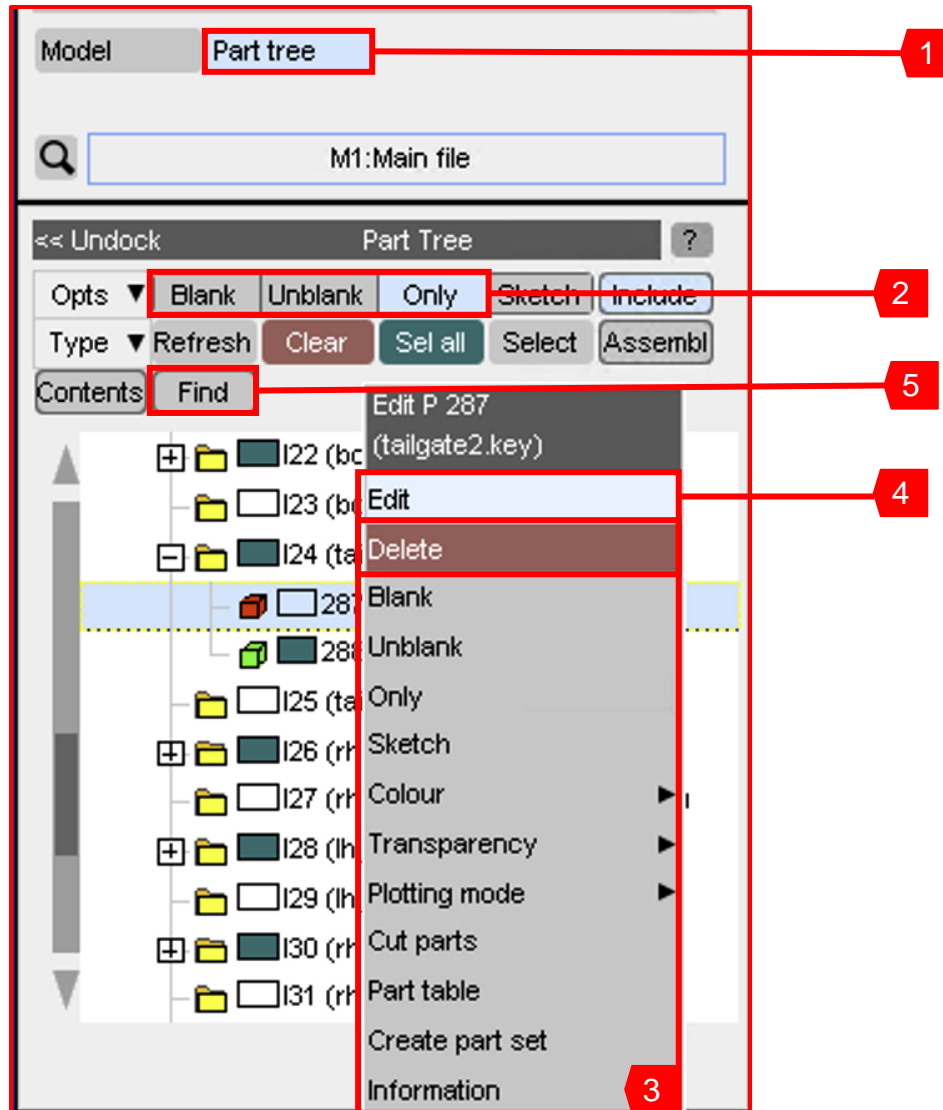
Entities Displayed Contd.

Turn on only solid elements – blank the engine

1. Select **CONSTRAINED** from the left menu list – the list on the right changes to show all CONSTRAINED types, if not, click on 'CONSTRAINED' for the list to appear.
2. Display each type one-by-one, update the plot using **Update**, **S** or **L**
3. Notice the user's errors in this model:
 - RIGID_BODIES joining the left and right front wheels
 - Nodal Rigid Bodies across the front doors

This is a method of checking the model. *Note: this model contains CONSTRAINED_SPOTWELD – we do not recommend this method of joining panels. Please ignore those spotwelds.*

Part Tree

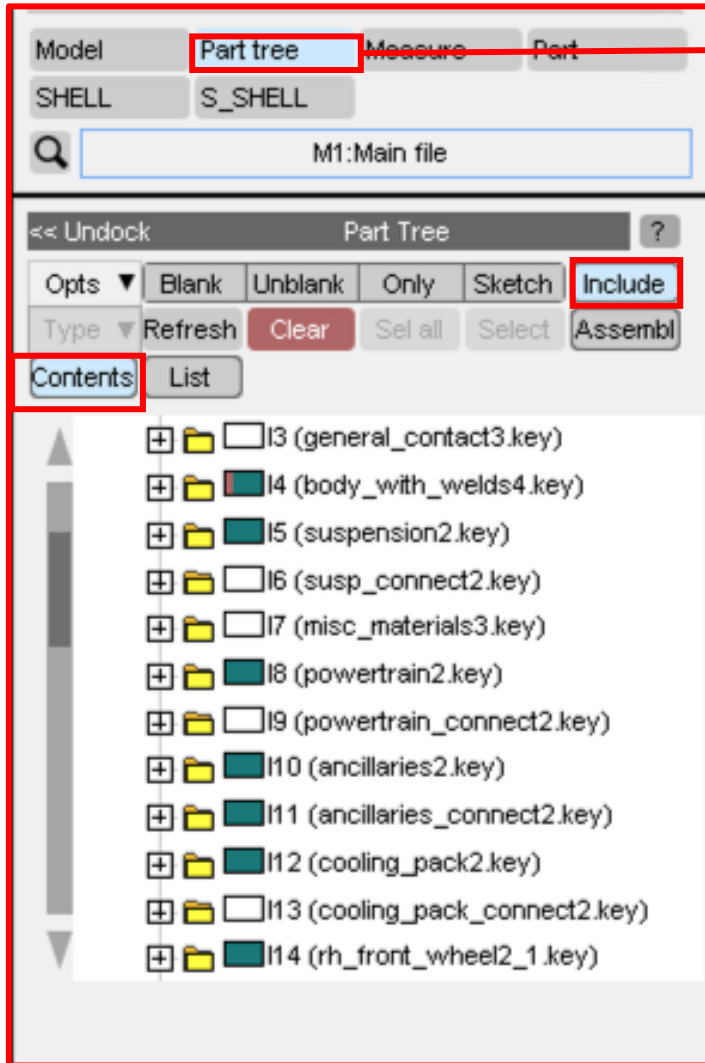


Part Tree

Before proceeding, go back to the Entity panel, turn off Constrained, turn on (Elements) Shells and Solids.

1. Click the **Part Tree** tab to bring up the Part Tree.
2. Click **ONLY**.
3. Click some INCLUDE files.
4. Expand an INCLUDE file (using the +).
5. Try click and shift-click to select multiple parts.
6. Right-click, change the **colour**.
7. Press **FIND**. Type in 194.
8. Click on the part highlighted.
9. Type in *Roof*.
10. Click on the part highlighted
11. Before proceeding, unblank all parts (shortcut U)

Part Tree

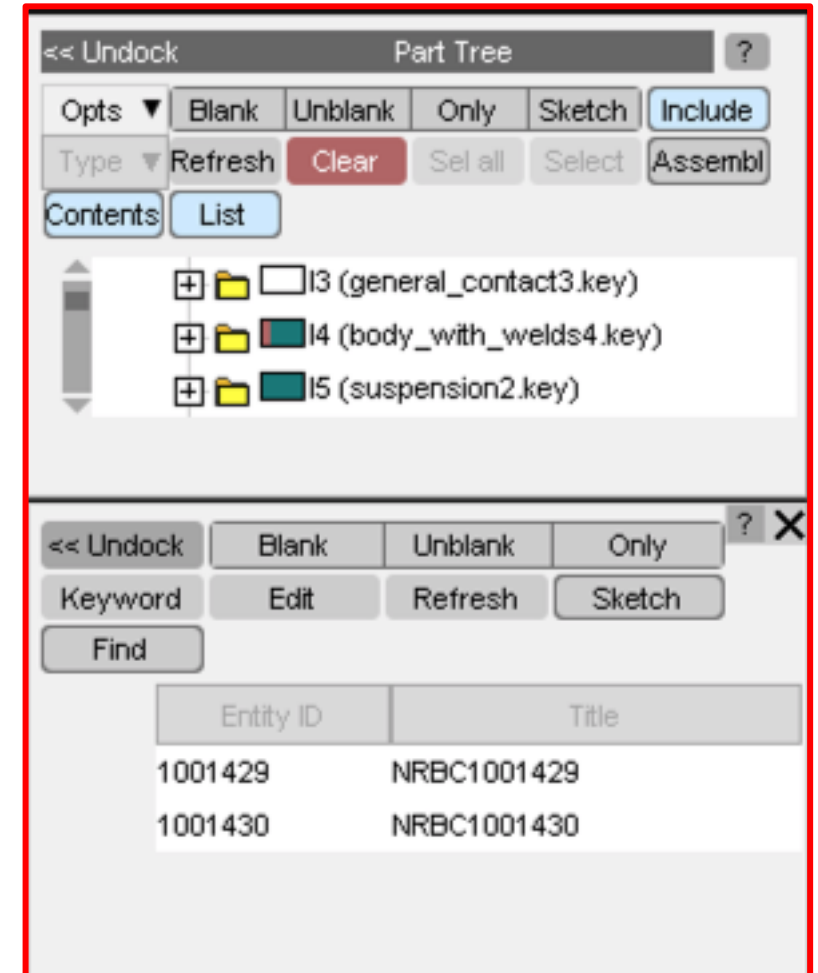


1

Part Tree

“Contents” mode allows you to view all entity types in the part tree, not just parts.

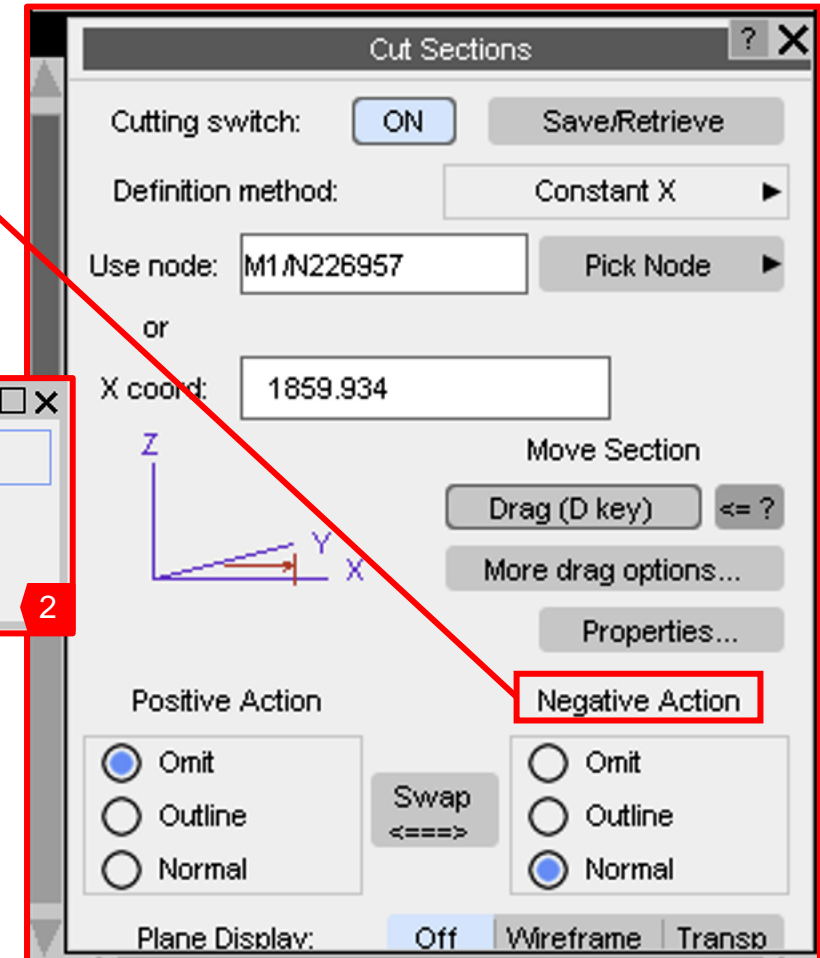
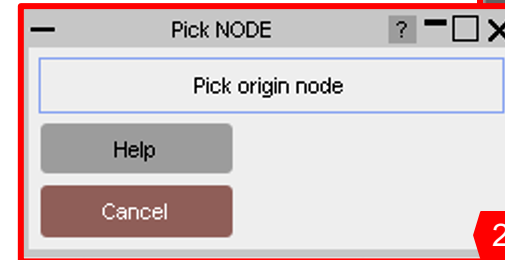
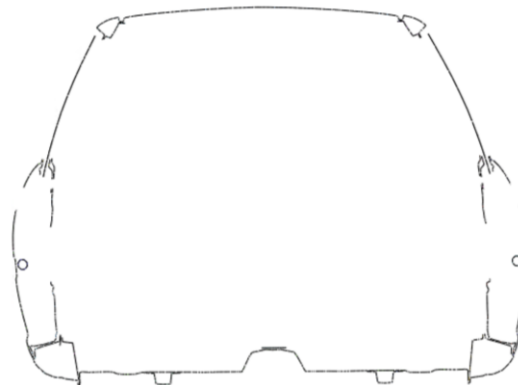
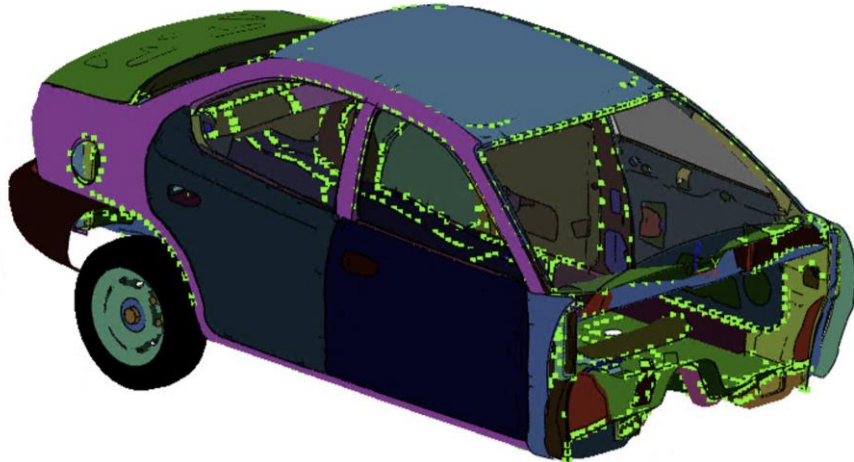
1. Click the **Part Tree** tab to bring up the Part Tree.
2. Click **CONTENTS**.
3. Expand an INCLUDE file (using the +).
4. Right-click, change the **colour**.
5. Press **INCLUDE** to turn off the display of include files.
6. Click **LIST** then select some entities



Cut Section

Cut Section

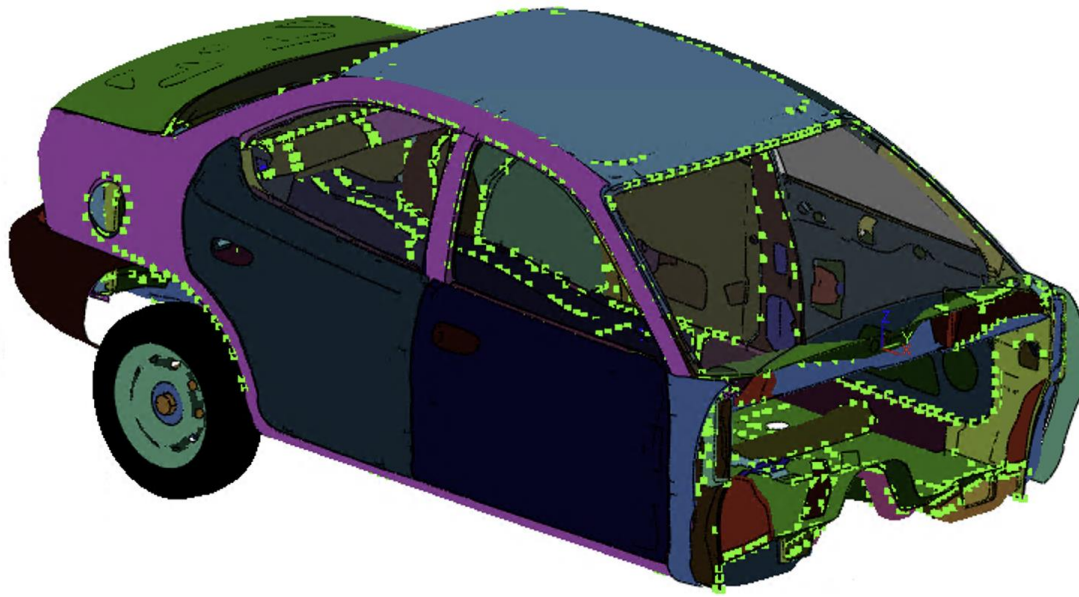
1. Shortcut **X** to bring up the cut-section menu.
2. Shortcut **N** to activate node-picking, then click near the center of the roof. Display changes to a cut-section at constant X through the picked node.
3. In the menu, change Negative Action to **Normal** (i.e elements behind the plane are drawn normally).



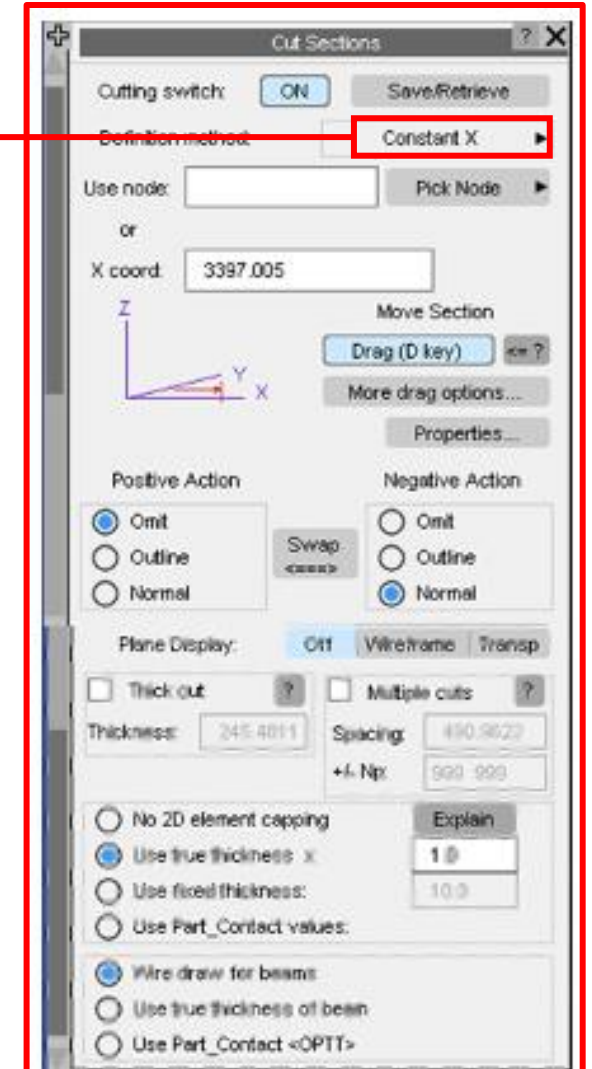
Cut Section

Cut Section

1. Zoom in on the cantrail. The cut edges of shells are drawn with their actual thickness – useful for seeing whether there are gaps (as in this model) or interferences.
2. Shortcut **D**, drag the section through the model.
3. Try switching the section to **Constant Y**.



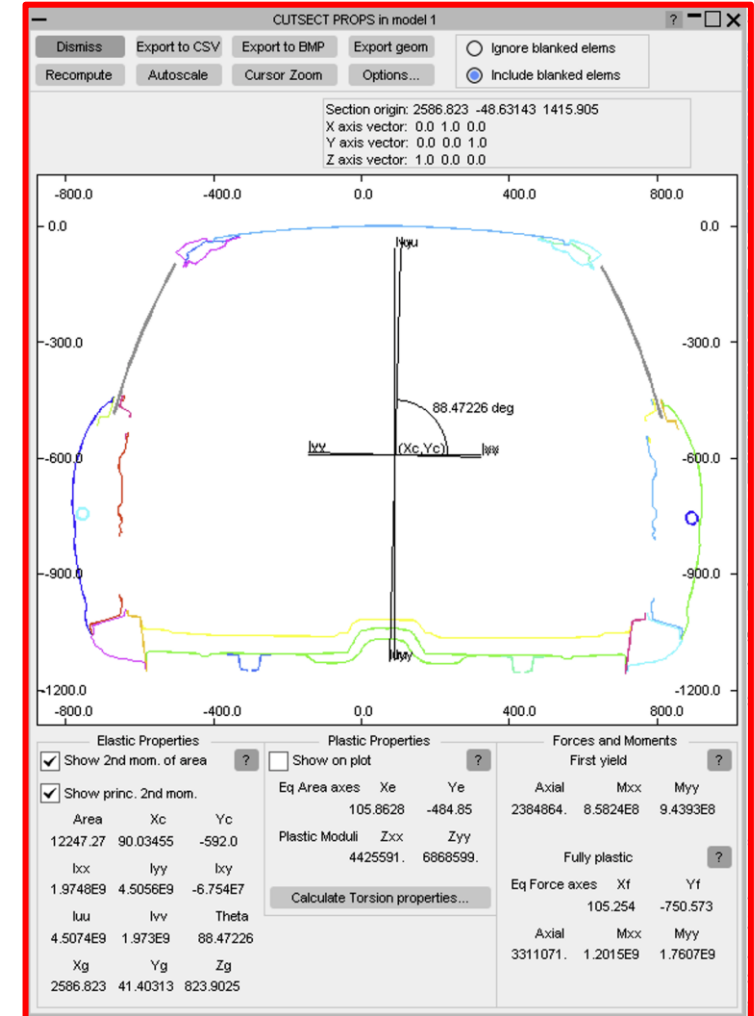
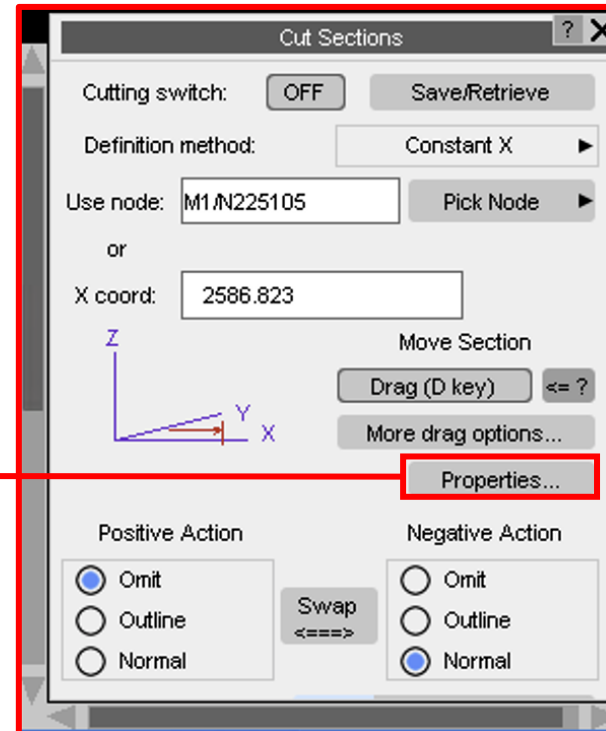
3



Cut Section

Cut Section

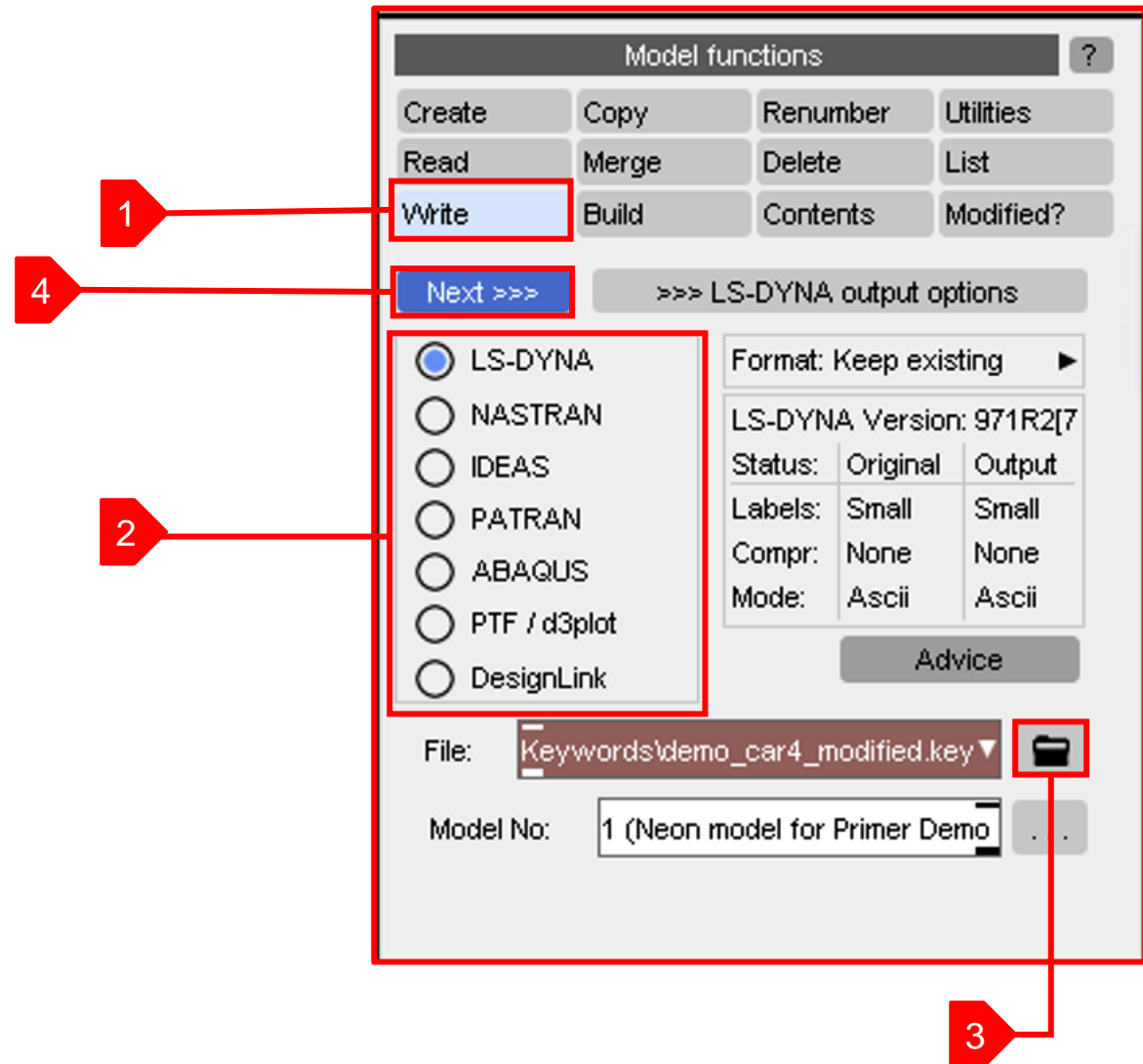
1. Switch back to constant x.
2. ONLY display one of the a-pillars.
3. Click on **PROPERTIES**.
4. Turn off the cut section and dismiss the menu.



Exercise: Write Out an LS-DYNA Model

Writing out a Model:

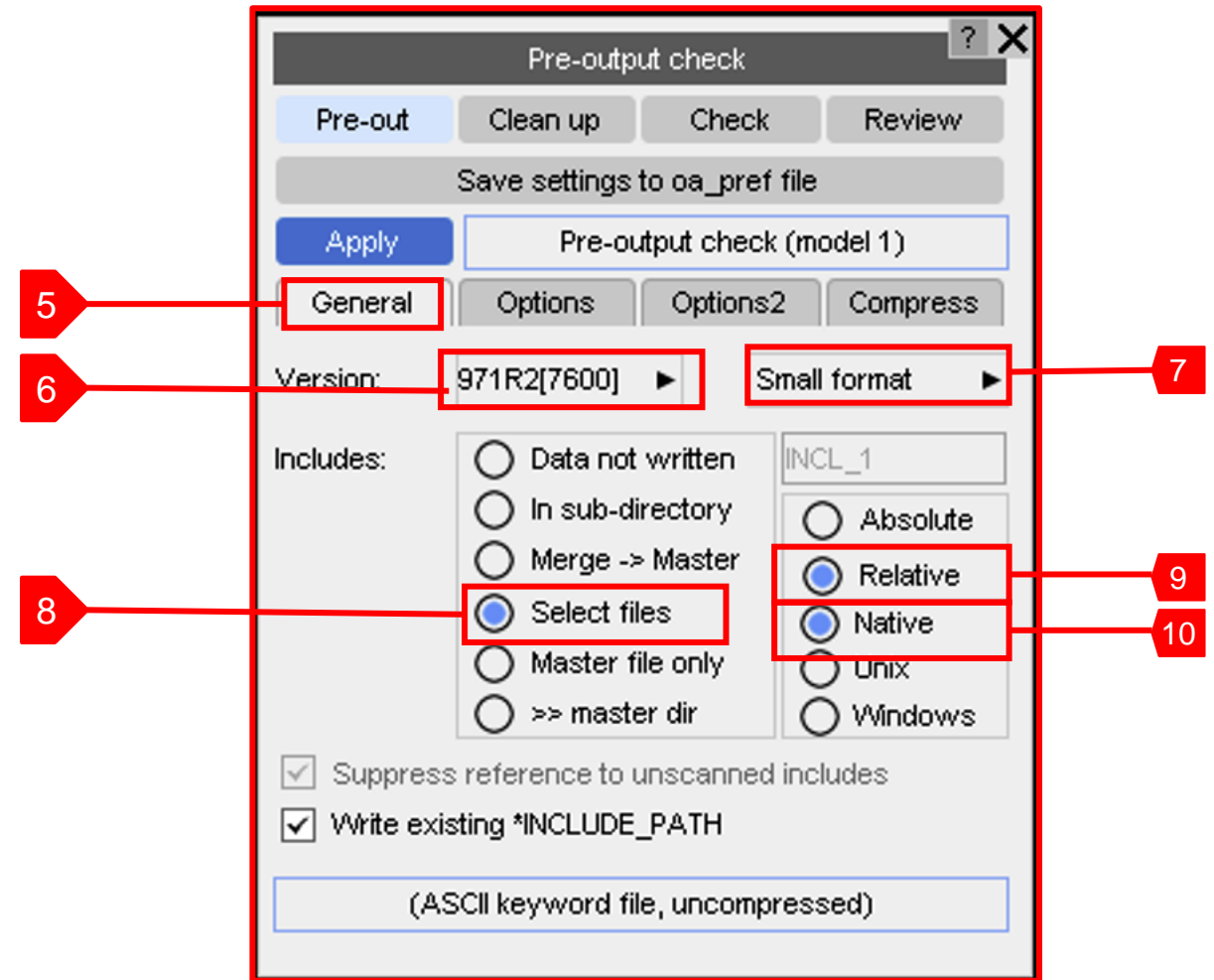
1. Start PRIMER. If the menu in the bottom right corner is not already shown as here, click **Model** and **Write**
2. Leave the file type set to LS-DYNA.
3. Using the file browser, type in the name of the master file that you want to write **demo_car4_modified.key**.
4. Click **Next**.



Exercise: Write Out an LS-DYNA Model

Writing out a Model:

5. You will enter the “Pre-output check” menu. This has 4 tabs. Click on the **General** tab.
6. Select Version. This is the version of LS-DYNA you are running. Some versions of LS-DYNA will have different formatting for some keywords, hence it is important to choose the version that matches your project.
7. Select the keyword format (e.g. Small format)
8. If you model has include files, decide how you want to save the include files. To preserve the existing include file structure, choose **Select files**.
9. Choose how the include path is written out in the keyword file (Absolute or Relative).
10. Choose the keyword file format (Native, Unix or Windows).



Exercise: Write Out an LS-DYNA Model

Pre-output check

Pre-out Clean up Check Review

Save settings to oa_pref file

Apply Pre-output check (model 1)

General **Options** Options2 Compress

Output: ☐ Alphabet ☒ Full matl name ☐ short matl name

☒ Classic

HM comm : aut ☒ Thumbnails ☐ Field headers

☒ Part colours ☐ ANSA comment

☒ Emb'd comments ☒ Xref comments

☐ Timestamp

☒ ZTF output Fast_full

ZTF opts

Pre-output check

Pre-out Clean up Check Review

Save settings to oa_pref file

Apply Pre-output check (model 1)

General Options **Options2** Compress

☐ Write Parameters as values

☐ Write all solids in 2-line format

☒ Auto-promote *ELEMENT cards

☐ Write overflowing character data

☒ Don't write check file

☒ Don't write model mass and CofG

☒ Don't write include file mass

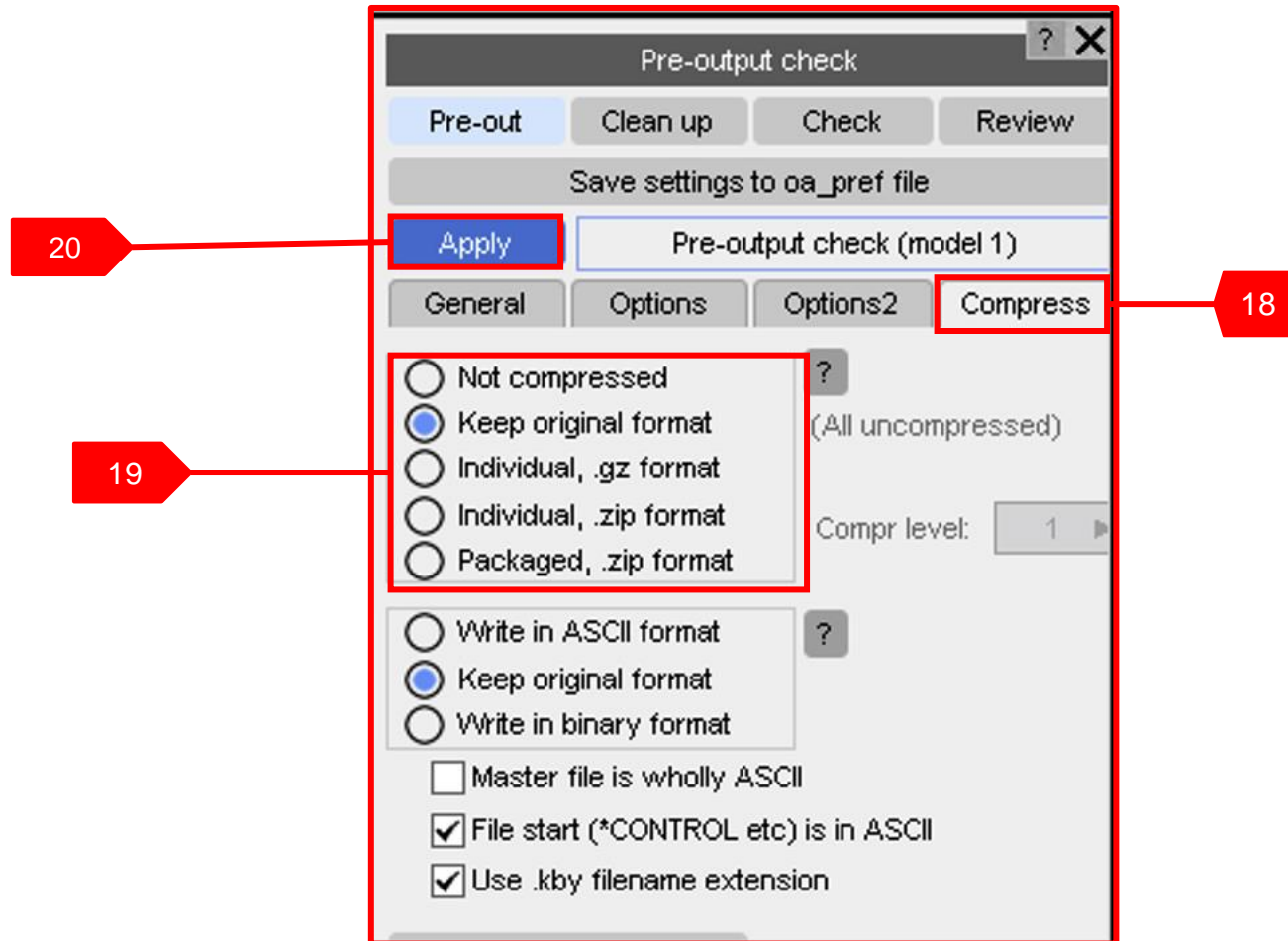
☐ write out all connections

☐ Write assembly data in Primer format

Writing out a Model:

11. Next, click on the **Options** tab.
12. Select output of keywords in either alphabetical order or **classic** order (where the control and database cards are at the top of the keyword file, traditionally called the top deck)
13. Select output naming for materials (typically **Full material name** is chosen).
14. Select check boxes for things written to keyword file as comment. Important to check **ZTF output**, to enable advanced functionality in Oasys POST.
15. Next, click on the **Options2** tab
16. Select whether check file is written to the keyword
17. Select whether additional data is written to the keyword

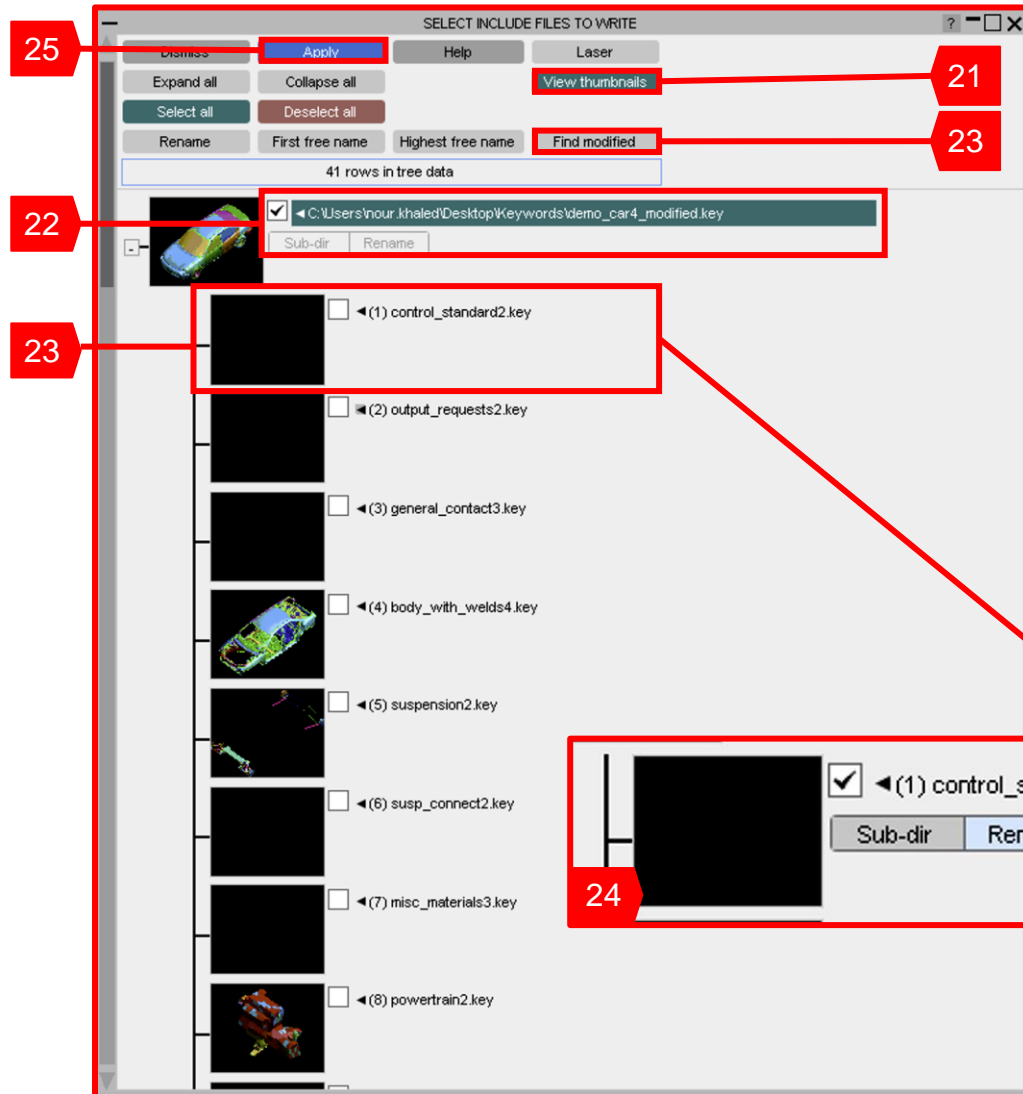
Exercise: Write Out an LS-DYNA Model



Writing out a Model:

18. Next, click on the **Compress** tab.
19. Select options for whether file is compressed or not.
20. Click on **Apply**.

Exercise: Write Out an LS-DYNA Model

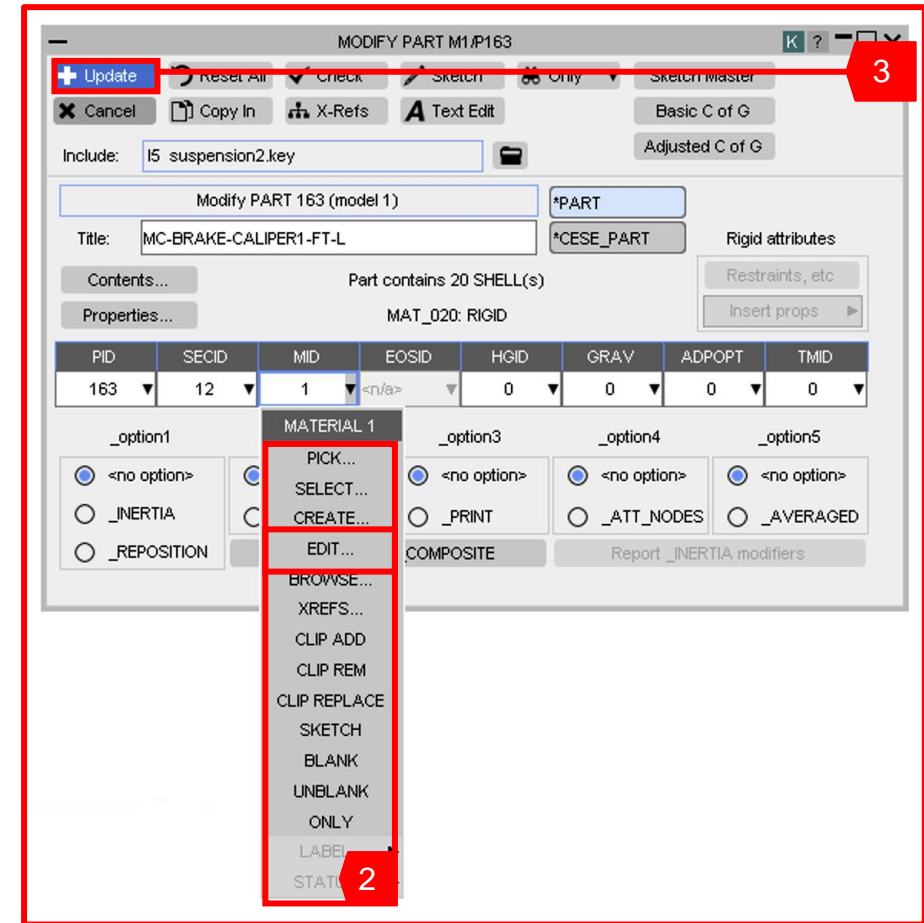
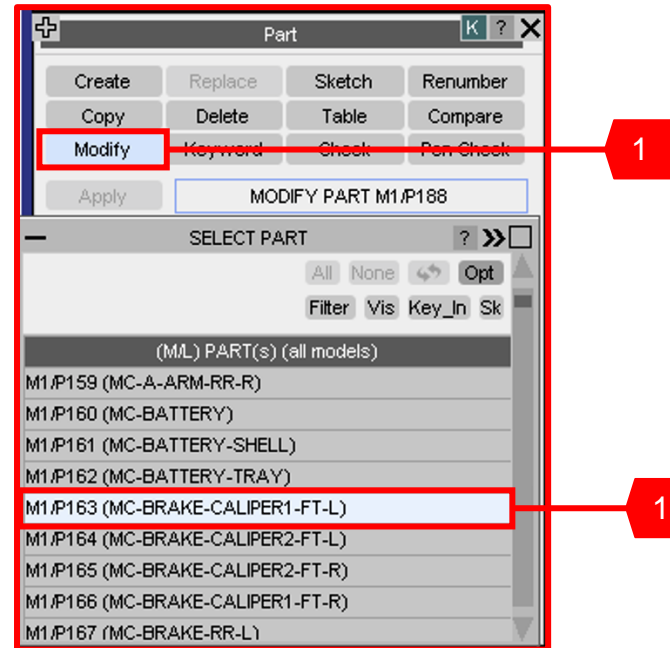
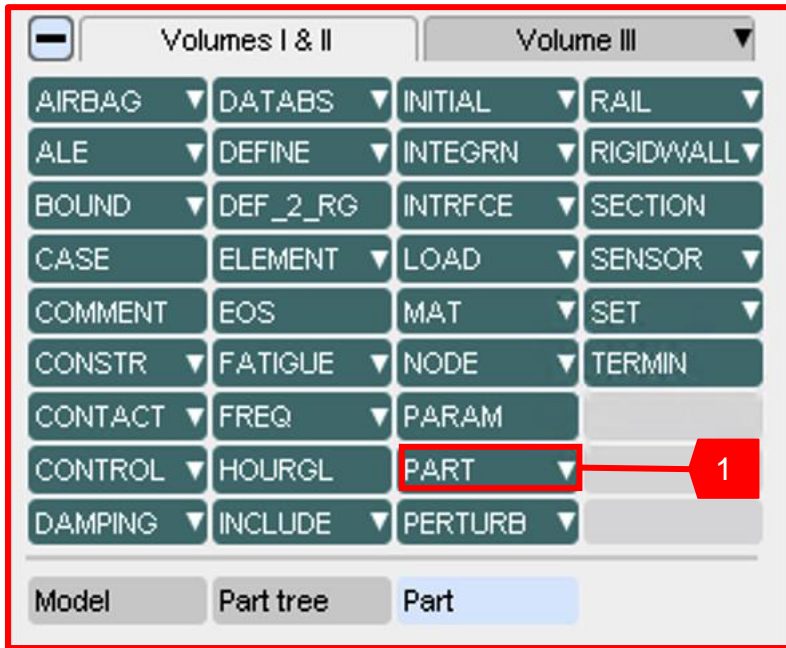


Writing out a Model:

21. The “select include files to write” menu opens up. Clicking the **View thumbnails** toggles include thumbnails on or off.
22. Select the folder and path for the master keyword file (note: you already defined the filename in step 3).
23. Click on Find modified. Any differences between the modified file and the original file are shown in the pop-up window. The include files that have changed are ticked.
24. Alternatively, the include files you want to write out can be manually ticked. Choose whether to write them out in a sub-directory or to Rename them and write them as a new filename.
25. Click on **Apply**.

Some basic functions – using Editing panels, Part Table, Remove, Orient

Using the Editing Panels

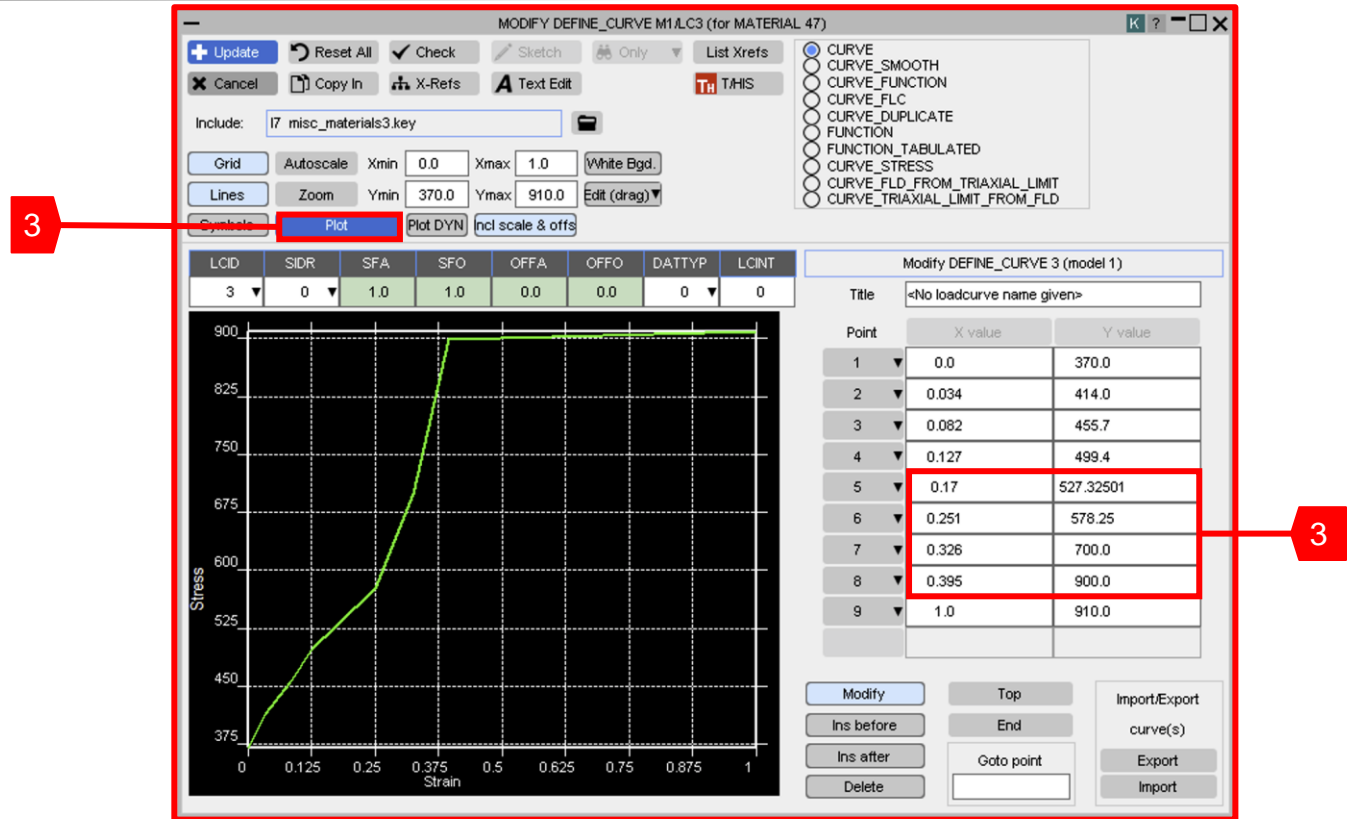
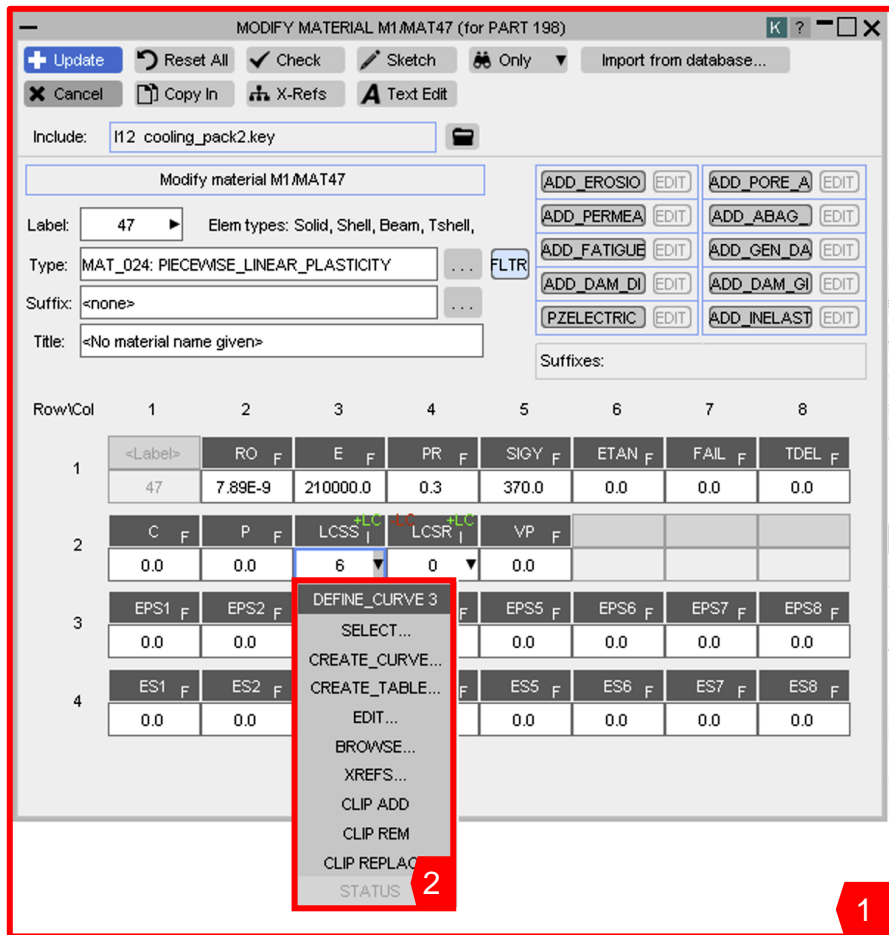


Example – editing a Part and its Material

Parts and all their referenced attributes can easily be edited using PRIMER.

1. Go to **Keywords** – **Part** – **Modify**, and select part 163 from the list.
2. Change the material ID by erasing “12” and typing “1”. Now try a different method - right click for further options: this right-mouse menu is available wherever the small arrow appears in a text box. Try **pick** (from screen), **select** (from list). Also try **edit**; change the Youngs Modulus to 10000. After making changes, click **Update** to make the changes permanent. Otherwise, if you press **Abort**, changes are not saved.

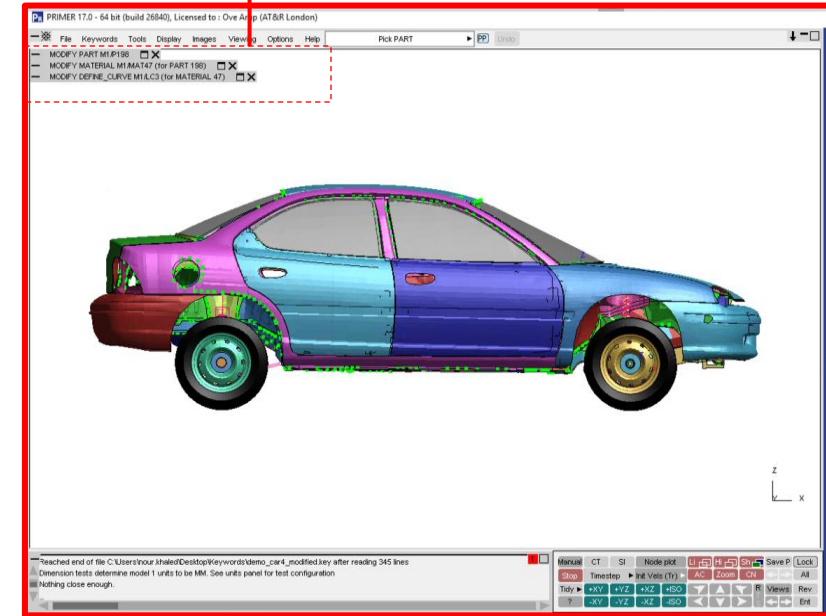
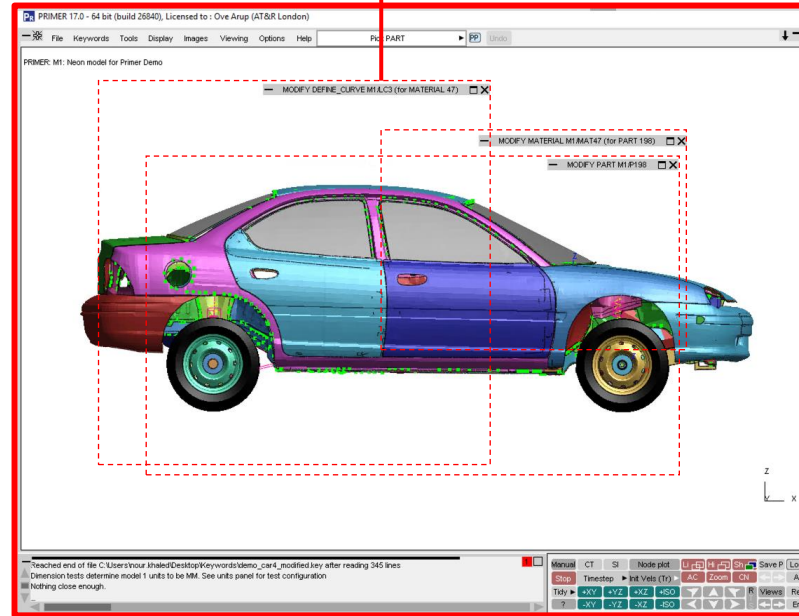
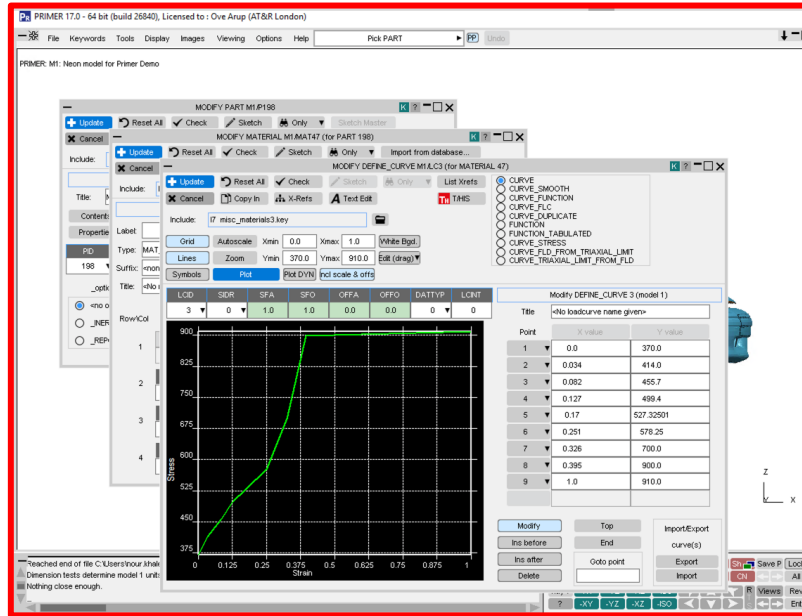
Using the Editing Panels



Part and Material Editing – example 2

1. Return to Quick-pick (shortcut Q). Right-click on the roof, Edit. Right-click on the Material ID, Edit. This material references a stress-strain curve, LCSS.
2. Right-click on the curve ID under LCSS, Edit. The editing menu for this loadcurve appears.
3. Change some data values and press **PLOT**.

Floating Menus



Press **i** to iconise

1

Press **t** to tidy

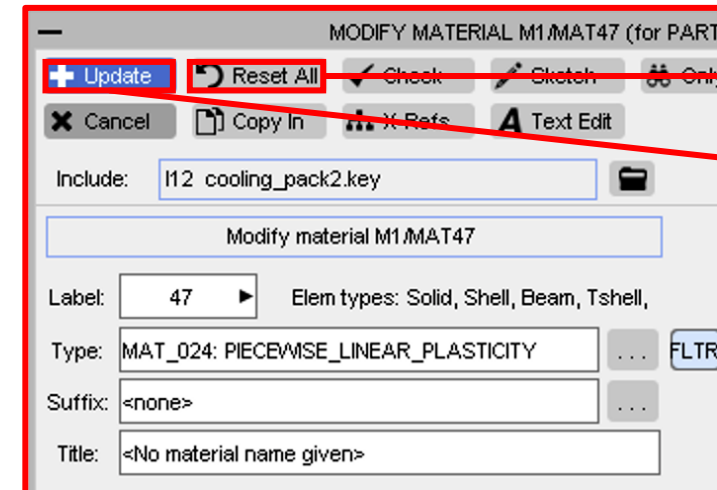
2

Floating Menus

Many menus “float” over the graphics – they can be controlled in several ways:

1. Press **i** to **i**conise all floating menus – they will shrink to their top bar
2. Press **t** to **t**idy all icons – they will be aligned in the top left graphics corner
3. Press **i** again to open up the menus
4. Use **ESC** key to dismiss all the floating menus without saving any changes (= **Cancel**)
5. **Enter** key will dismiss a menu and save any changes (= **UPDATE**)

Note the cursor must be over the menu for ESC & Enter to function



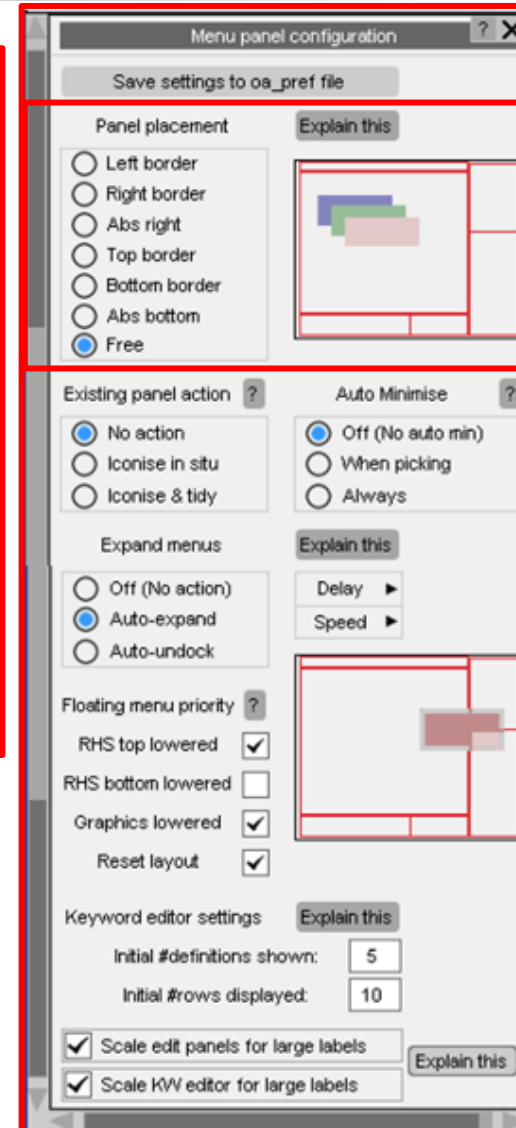
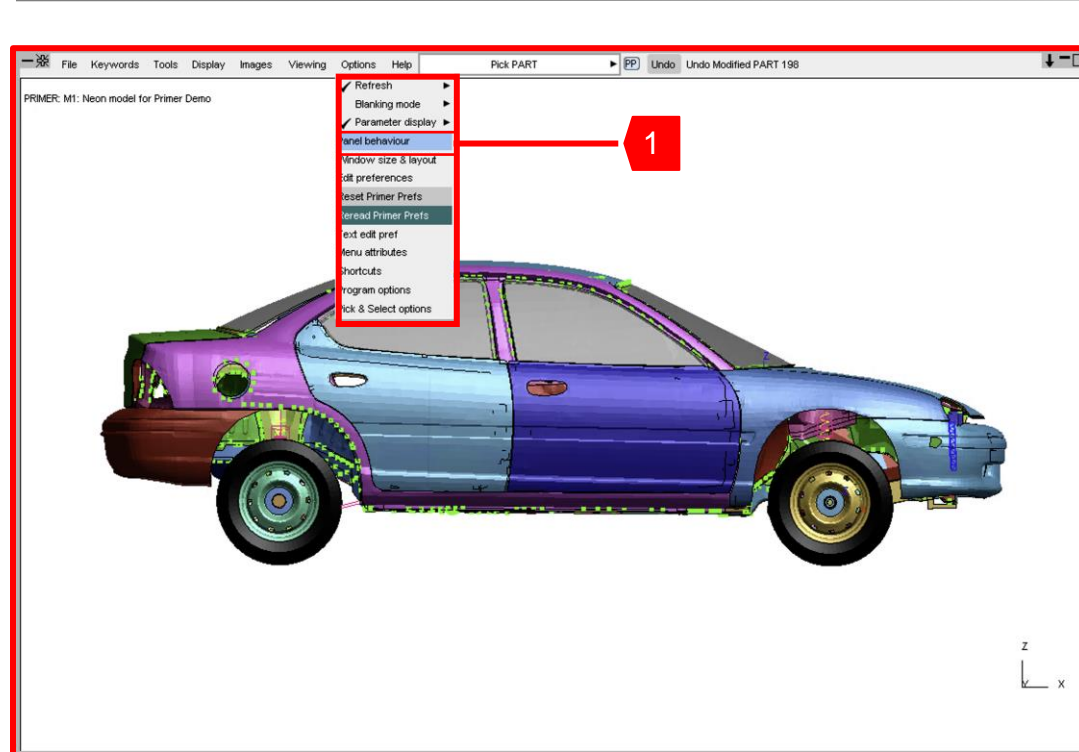
= **ESC**

4

= **Enter**

5

Floating Menus



Floating Menus

You can select where you want floating panels to appear:

1. Click **OPTIONS->PANEL BEHAVIOR**.
2. "Panel placement" setting can be changed here (default **FREE**).

Creating Data Using Edit Menus

Creating data

We will create a new part for shells, 1mm thick, with elasto-plastic material.

1. Keyword=>Part=>Create: the Part editing menu appears. Text boxes coloured red must be filled in before the data can be valid.
2. To choose the Label (ID) for the new part, you can type in 1, or use the drop-down menu (right-click), choose First Free.

CREATE PART in model 1

+ Create Reset All Check Sketch Only Sketch Master

Cancel Copy In X-Refs Text Edit Basic C of G Adjusted C of G

Include: M1 <Master file>

Create PART (model 1) *PART *CESE_PART

Title: <No part name given>

Contents... Part has no elements defined

Properties... Material type: <undefined>

Rigid attributes

Restrains, etc

Insert props

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
1	0	0	0	0	0	0	0

PART label

Highest + 1 (10001)

First free (1)

Highest + 1 in layer (10001)

First free in layer (274)

Highest free in layer (10001)

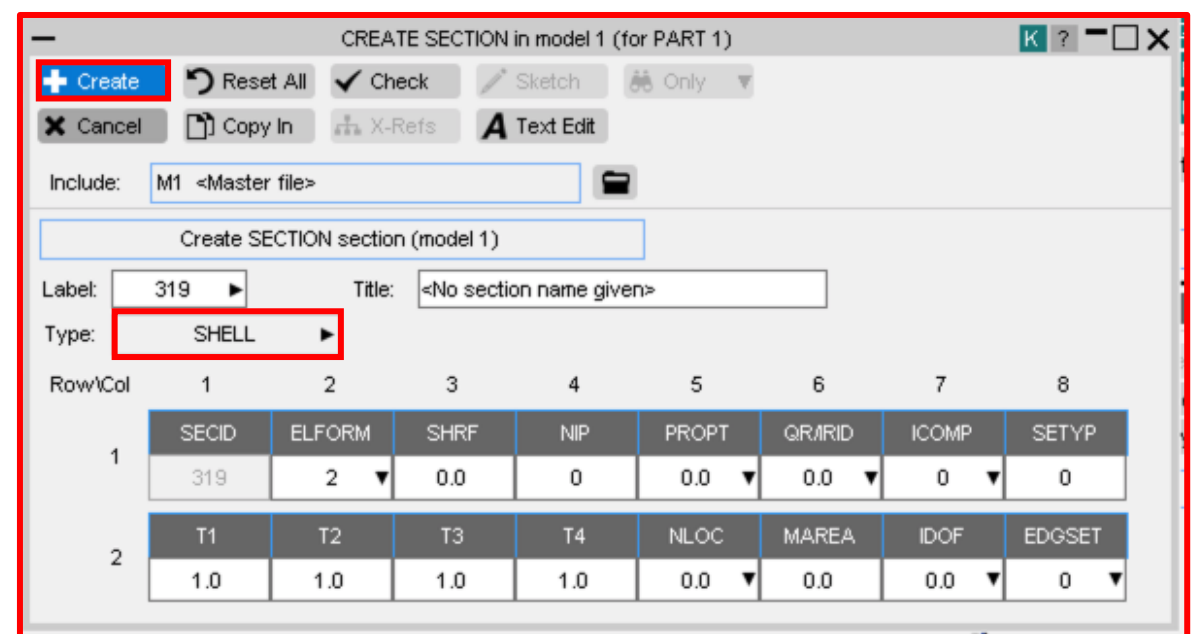
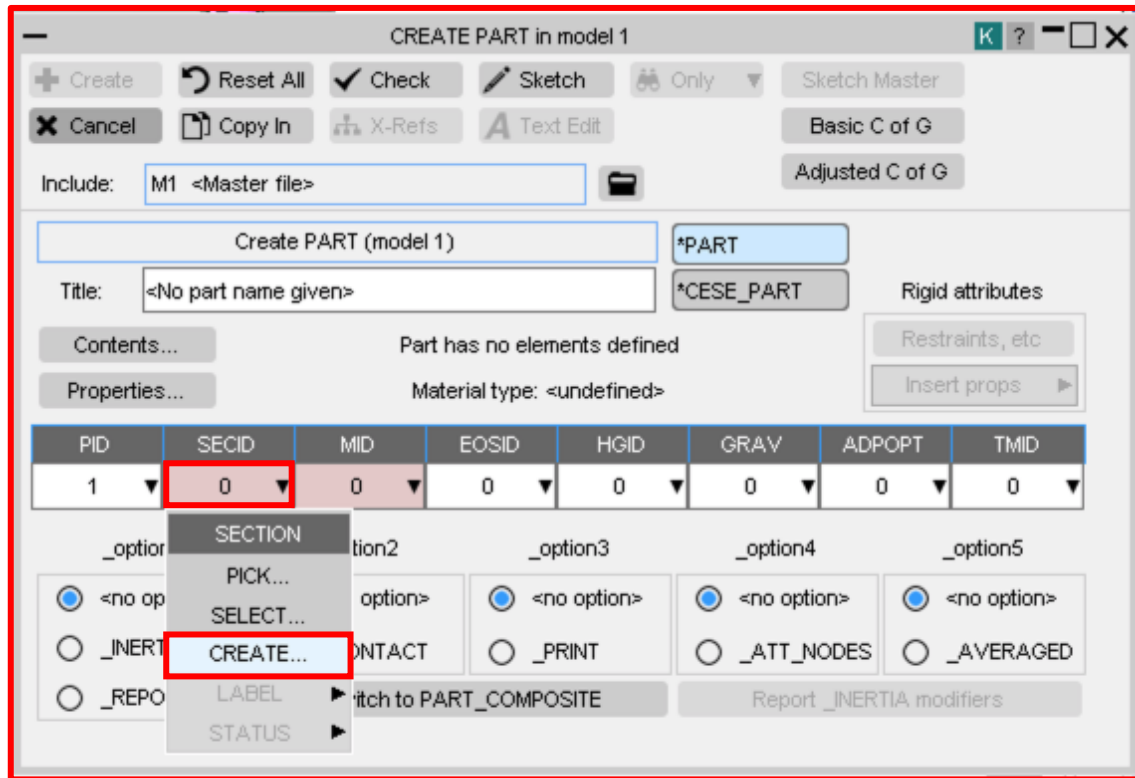
Lock label

_option2 no option> _option3 <no option> _option4 <no option> _option5 <no option>

CONTACT _PRINT _ATT_NODES _AVERAGED

switch to PART_COMPOSITE Report _INERTIA modifiers

Creating Data Using Edit Menus



Creating data

We will create a new part for shells, 1mm thick, with elasto-plastic material.

1. From Section ID, drop-down to Create.
2. Create a new section with ID 3001, element type Shell, thickness 1.0, and leave all other entries as the default. Press CREATE_SECTION.

Creating Data Using Edit Menus

Creating data

We will create a new part for shells, 1mm thick, with elasto-plastic material.

1. From Material ID, drop-down to Create. Create a material with ID 3001, material type 3, RO=7.8e-9, E=200e3, PR=0.3, SIGY=300, ETAN=1000.
2. Press CREATE_MATERIAL.

CREATE MATERIAL in model 1 (for PART 1)

+ Create **↺** Reset All **✓** Check **✗** Cancel **📄** Copy In **🔗** X-Refs **📁** Import from database...

Include: M1 <Master file>

Create material in model

Label: 235 ▶ Elem types: Solid, Shell, Beam, Tshell,

Type: MAT_003: PLASTIC_KINEMATIC ... FLTR

Suffix: <none> ...

Title: <No material name given>

Either browse for the material type, or enter the type number (in this case, 3)

Suffixes:

Row\Col	1	2	3	4	5	6	7	8
1	<Label>	RO F	E F	PR F	SIGY F	ETAN F	BETA F	
	235	7.8E-9	200000.0	0.3	300.0	1000.0	0.0	
2	SRC F	SRP F	FS F	VP F				
	0.0	0.0	0.0	0.0				

Creating Data Using Edit Menus

Creating data

We will create a new part for shells, 1mm thick, with elasto-plastic material.

1. Give the part a title e.g. "My new Part"
2. In the Part menu, all the data input is now complete – press Create.

CREATE PART in model 1

Create Reset All Check Sketch Only Sketch Master
Cancel Copy In X-Refs Text Edit Basic C of G Adjusted C of G

Include: M1 <Master file>

Create PART (model 1) *PART *CESE_PART Rigid attributes
Title: My New Part Restraints, etc
Contents... Part has no elements defined Insert props
Properties... MAT_020: RIGID

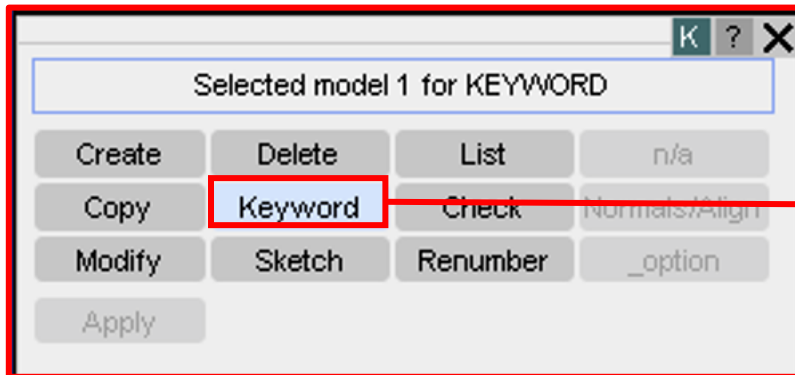
PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
1	1	1	0	0	0	0	0

_option1 _option2 _option3 _option4 _option5

☒ <no option> ☒ <no option> ☒ <no option> ☒ <no option> ☒ <no option>
☐ _INERTIA ☐ _CONTACT ☐ _PRINT ☐ _ATT_NODES ☐ _AVERAGED
☐ _REPOSITION

Switch to PART_COMPOSITE Report _INERTIA modifiers

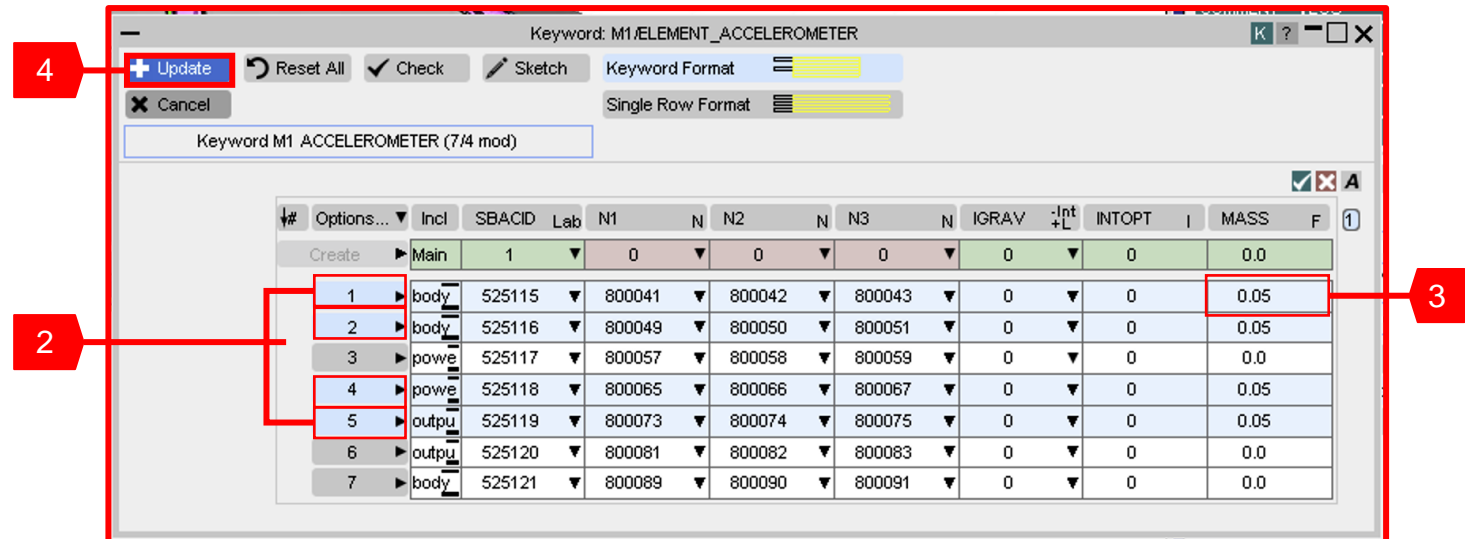
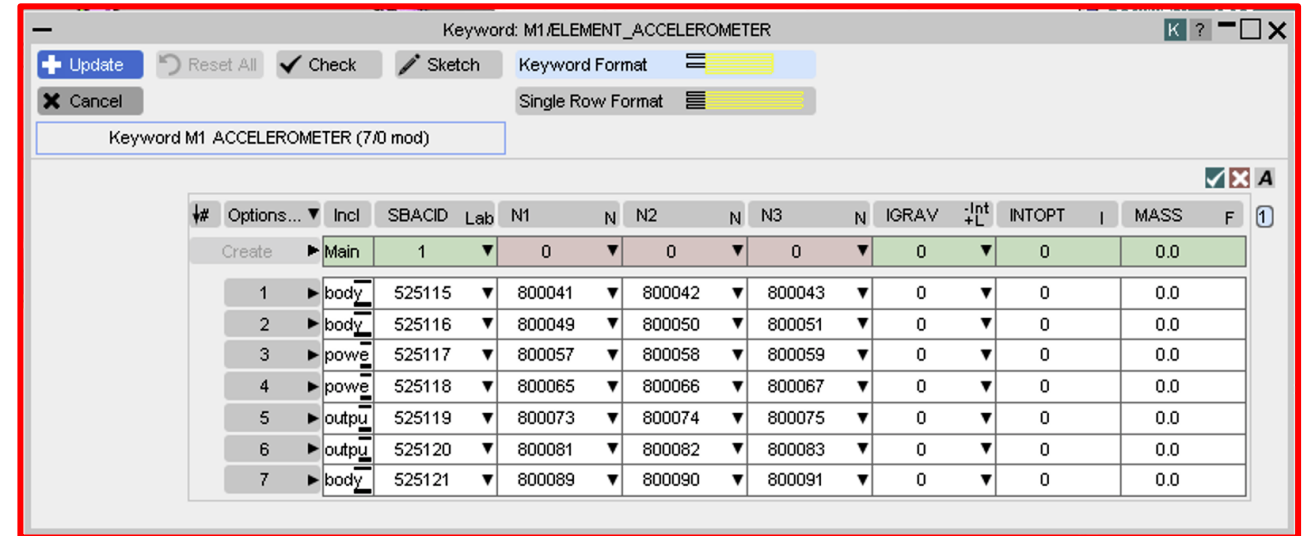
Keyword Editor – Editing Existing Data



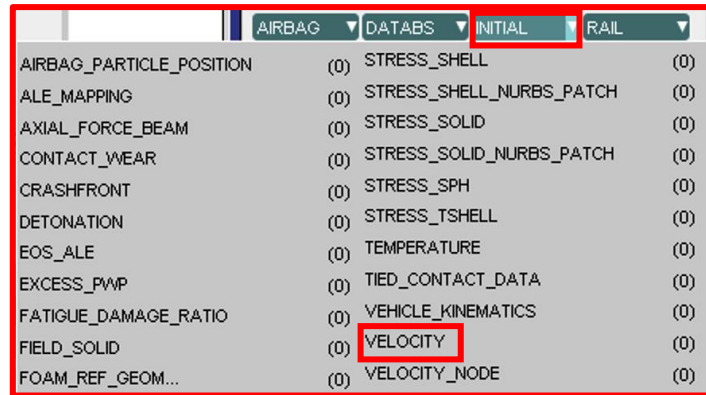
Keyword Editor

Many KEYWORDS can be edited using a generic menu format called the Keyword Editor – the same field of many keywords can be updated in one action

1. For example open the Keyword editor for *ELEMENT_SEATBELT_ACCELEROMETER
2. Select some accelerometers from the list
3. Enter 5e-2 in to one of the selected rows. The value will be automatically entered into all the selected rows
4. Press **UPDATE**



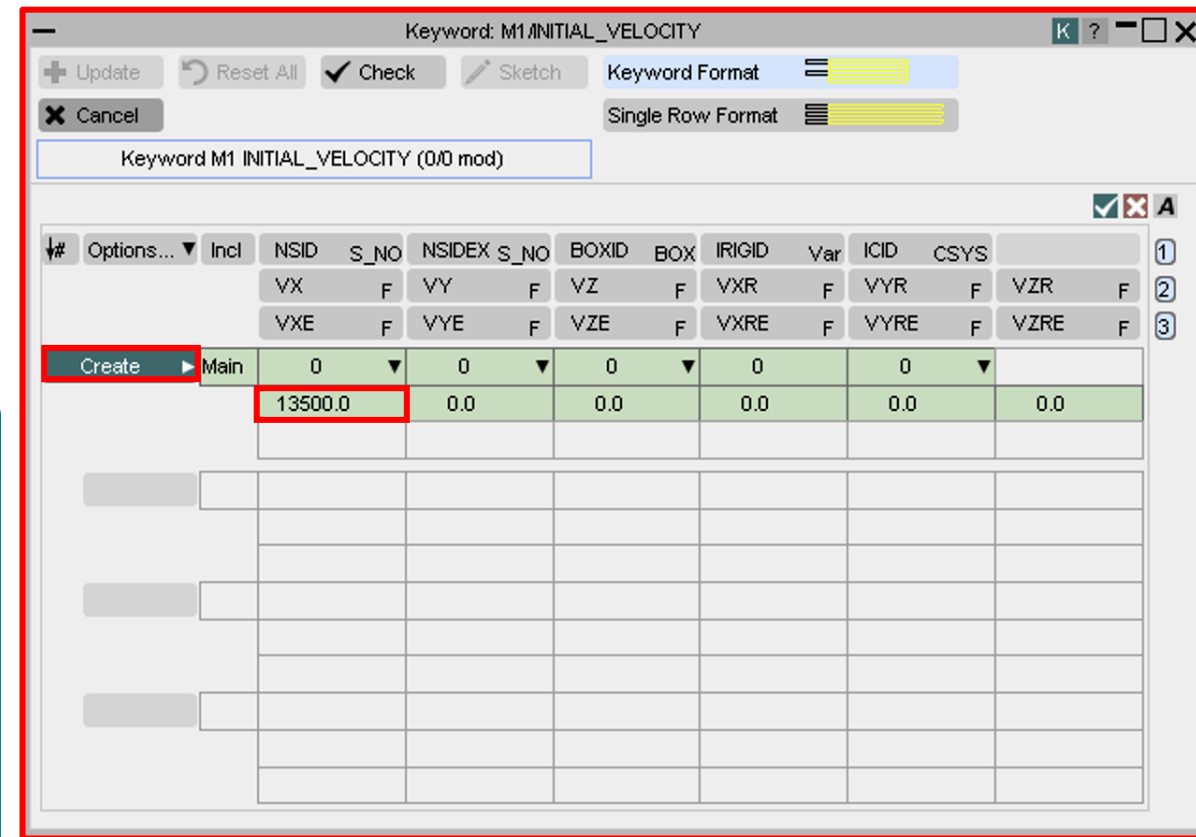
Keyword Editor – Creating New Data



Keyword Editor

Some entity types have both Keyword editor and single-entity editor (Modify menu). Others have only a Modify menu, while others again have only a Keyword editor.

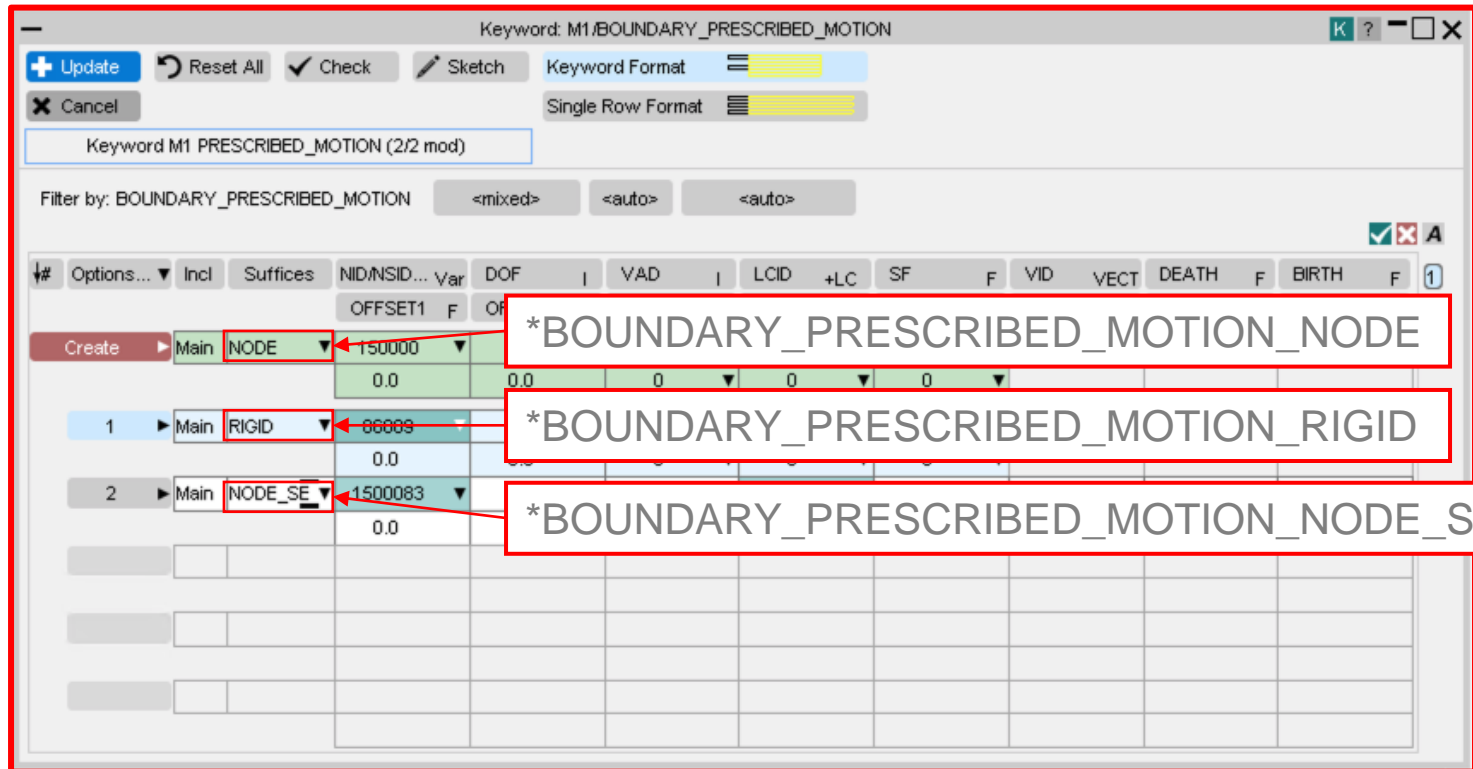
1. Go to Keyword=>INITIAL_VELOCITY.
2. The Create and Modify buttons are greyed out – only a Keyword editor is available. Start the keyword editor. There are no Initial Velocity definitions currently in the model.
3. Enter the data shown: VX=13500.
4. Press **CREATE** – the data appears in Green.
5. The data is still only temporary – to save it in the model, press **UPDATE**.
6. When you redraw the model, PRIMER will also draw the newly-created initial velocity data. To avoid this, visit the Entity panel and turn off INITIAL. Entity visibility is switched on whenever new data is created.



Keyword Editor – Displaying Data

Keyword Editor

The keyword editor displays all the _OPTIONS for this keyword together.
Go to BOUND keyword then PRESCRIBED_MOTION.



Keyword: M1/BOUNDARY_PRESCRIBED_MOTION

Buttons: + Update, Reset All, Check, Sketch, Keyword Format, Single Row Format, X Cancel

Keyword M1 PRESCRIBED_MOTION (2/2 mod)

Filter by: BOUNDARY_PRESCRIBED_MOTION <mixed> <auto> <auto>

#	Options...	Incl	Surfaces	NID/NSID...	Var	DOF	VAD	LCD	+LC	SF	F	VID	VECT	DEATH	F	BIRTH	F
	Create	Main	NODE	150000													
				0.0		0.0	0	0	0								
1	Main	RIGID	86689														
				0.0													
2	Main	NODE_SET	1500083														
				0.0													

*BOUNDARY_PRESCRIBED_MOTION_NODE

*BOUNDARY_PRESCRIBED_MOTION_RIGID

*BOUNDARY_PRESCRIBED_MOTION_NODE_SET

Keyword Editor – Displaying Data

Keyword Editor

Each line of data may be blanked. In this example (*ELEMENT_BEAM), the second lines have been blanked.

Keyword: M1/ELEMENT_BEAM

Filter by: ELEMENT_BEAM

#	Options...	Incl	Suffixes	EID	Lab	PID	P	N1	N	N2	N	N3	N	RT1	I	RR1	I	RT2	I	RR2	I	LOCAL	I
				PID1	P	PID2	P																
Create	▶	Main	<none>	460053	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
1	▶	body	PID	1	▼	9999	▼	9999000	▼	9999001	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
2	▶	body	PID	2	▼	9999	▼	9999001	▼	9999002	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
3	▶	body	PID	3	▼	9999	▼	9999003	▼	9999004	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
4	▶	body	PID	4	▼	9999	▼	9999005	▼	9999006	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
5	▶	body	PID	5	▼	9999	▼	9999006	▼	9999007	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
6	▶	body	PID	6	▼	9999	▼	9999008	▼	9999009	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
7	▶	body	PID	7	▼	9999	▼	9999009	▼	9999010	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
8	▶	body	PID	8	▼	9999	▼	9999011	▼	9999012	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
9	▶	body	PID	9	▼	9999	▼	9999012	▼	9999013	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
10	▶	body	PID	10	▼	9999	▼	9999014	▼	9999015	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼
11	▶	body	PID	11	▼	9999	▼	9999015	▼	9999016	▼	0	▼	0	▼	0	▼	0	▼	0	▼	0	▼

Line 1

Line 2

Keyword Editor – Displaying Data

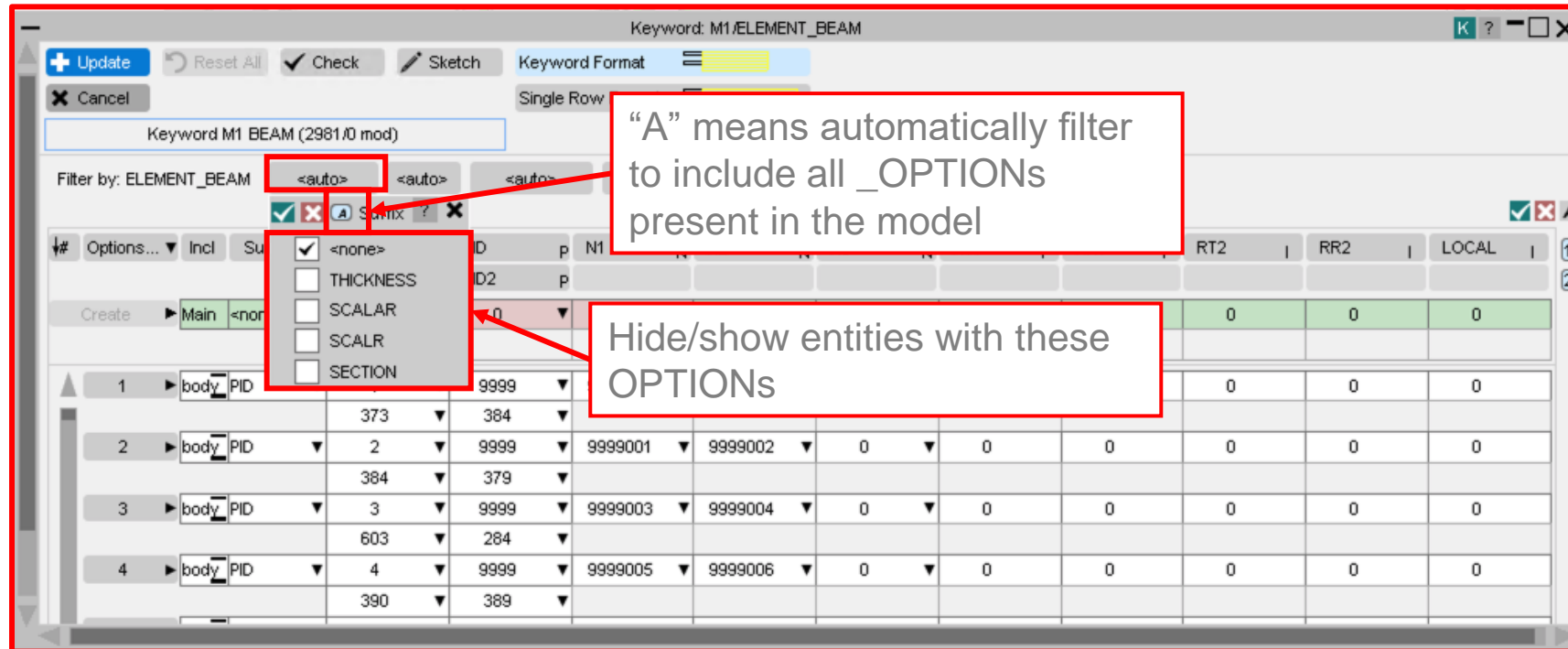
Keyword Editor

Where the LS-DYNA data has multiple lines, “Single-row layout” displays the data on a single line.

The screenshot shows the Keyword Editor window for 'Keyword: M1/ELEMENT_BEAM'. The 'Single Row Format' option is selected under the 'Keyword Format' dropdown. The table below displays the data for the 'M1 BEAM (2981/0 mod)' keyword.

#	Options...	Incl	Suffices	EID	Lab	PID	P	N1	N	N2	N	N3	N	RT1	I	RR1	I	RT2	I	RR2	I	LOCAL	I	PID1	P	PID2	P
Create	▶	Main	<none>	460053		0		0		0		0		0		0		0		0		0					
1	▶	body	PID	1		9999		9999000		9999001		0		0		0		0		0		0		373		384	
2	▶	body	PID	2		9999		9999001		9999002		0		0		0		0		0		0		384		379	
3	▶	body	PID	3		9999		9999003		9999004		0		0		0		0		0		0		603		284	
4	▶	body	PID	4		9999		9999005		9999006		0		0		0		0		0		0		390		389	
5	▶	body	PID	5		9999		9999006		9999007		0		0		0		0		0		0		389		377	
6	▶	body	PID	6		9999		9999008		9999009		0		0		0		0		0		0		390		389	
7	▶	body	PID	7		9999		9999009		9999010		0		0		0		0		0		0		389		377	
8	▶	body	PID	8		9999		9999011		9999012		0		0		0		0		0		0		390		389	
9	▶	body	PID	9		9999		9999012		9999013		0		0		0		0		0		0		389		377	
10	▶	body	PID	10		9999		9999014		9999015		0		0		0		0		0		0		390		389	
11	▶	body	PID	11		9999		9999015		9999016		0		0		0		0		0		0		389		377	

Keyword Editor – Displaying Data



Keyword Editor

It is possible to display only those entities having certain _OPTIONS, using the "Filter by" buttons:

Keyword Editor – Displaying Data

Keyword Editor

The data may be sorted by clicking on the header of any column.

Filter by: ELEMENT_BEAM <auto> <auto> <auto> <auto> <auto> <auto>

#	Options...	Incl	Suffices	EID	Lab	PID	P	N1	N2	N	N3	N	RT1	RR1	RT2	RR2	LOCAL
				PID1	P	PID2	P										
Create	▶	Main	<none>	2917		0		0	0	0	0	0	0	0	0	0	0
1	▶	cool	<none>	307753		184		314210	155006	308880		0	0	0	0	0	0
2	▶	cool	<none>	307752		184		314188	155031	308880		0	0	0	0	0	0
3	▶	ancil	<none>	300729		289		300419	157516	300413		0	0	0	0	0	0
4	▶	susp	<none>	460040		306		460035	207004	310567		0	0	0	0	0	0

Keyword Editor – Displaying Data

Keyword Editor

The “Options” button allows you to select entities in the keyword editor, and then carry out various operations on those selected.

The screenshot shows the Keyword Editor window with the title bar 'Keyword: M1/ELEMENT_BEAM'. The window contains a toolbar with buttons: '+ Update', 'Reset All', 'Check', 'Sketch', 'Keyword Format', and 'Single Row Format'. Below the toolbar is a text field containing 'Keyword M1 BEAM (2981/0 mod)'. A filter bar shows 'Filter by: ELEMENT_BEAM' followed by several '<auto>' buttons. The main area is a table with columns: #, Options..., Incl, Suffices, EID, Lab, PID, p, N1, N, N2, N, N3, N, RT1, RR1, RT2, RR2, and LOCAL. The 'Options...' menu is open, showing options: Options..., Sel_All, Unsel_All, Select..., Show_All, Only_Sel, Hide_Sel, Sketch_Sel, Reset_Sel, Delete_Sel, and Help. The table data is as follows:

#	Options...	Incl	Suffices	EID	Lab	PID	p	N1	N	N2	N	N3	N	RT1	RR1	RT2	RR2	LOCAL
				PID1	p	PID2	p											
			<none>	2917		0		0		0		0		0	0	0	0	0
			<none>	307753		184		314210		155006		308880		0	0	0	0	0
			<none>	307752		184		314188		155031		308880		0	0	0	0	0
			<none>	300729		289		300419		157516		300413		0	0	0	0	0
			<none>	460040		306		460035		207004		310567		0	0	0	0	0

Keyword Editor – Displaying Data

Keyword Editor

Further options are available from right-click on entity (in left column) – Sketch, Blank, Delete, etc

The screenshot shows the Keyword Editor window for the keyword M1/ELEMENT_BEAM. The window includes a toolbar with buttons for Update, Reset All, Check, Sketch, Keyword Format, and Single Row Format. Below the toolbar is a text field containing 'Keyword M1 BEAM (2981/0 mod)' and a filter section labeled 'Filter by: ELEMENT_BEAM' with several '<auto>' buttons. The main area is a table with columns for #, Options..., Incl, Suffices, EID, Lab, PID, p, N1, N, N2, N, N3, N, RT1, RR1, RT2, RR2, and LOCAL. The table contains several rows of data. A context menu is open over the first row of the table, showing options: Edit..., Check, Reset, Xrefs, Sketch, Blank, Unblank, Only, Text edit, Export file, Delete, and Help. The 'Sketch' and 'Blank' options are highlighted with a red box.

#	Options...	Incl	Suffices	EID	Lab	PID	p	N1	N	N2	N	N3	N	RT1	RR1	RT2	RR2	LOCAL
1	B307753			307753	184	314210	155006	308880	0	0	0	0	0	0	0	0	0	
2				307752	184	314188	155031	308880	0	0	0	0	0	0	0	0	0	
3				300729	289	300419	157516	300413	0	0	0	0	0	0	0	0	0	
4				460040	306	460035	207004	310567	0	0	0	0	0	0	0	0	0	

Using Option Selection Menu

1. PRIMER 17.0 - 64 bit (build 26840), Licensed to : Ove Arup (AT&R London)

File Keywords Tools Display Images Viewing Options Help Pick PART

2. SCREEN PI ? [X]

Dismiss Help

Picked NO PART(s)

Scr_Area All_Vis

Scr_Circ Scr_Poly

Path_L Path_A

Free_Edge Hole

Feat_Line An 20

Feat_Abs Explain

<undefined>

<undefined>

<undefined>

3. SELECT PART ? [X]

All None [X] Opt

Filter Vis Key_In Sk

(M/L) PART(s) (all models)

M1/P152 (MC-A-ARM-BUSH1-L)

M1/P153 (MC-A-ARM-BUSH1-R)

M1/P154 (MC-A-ARM-BUSH2-L)

M1/P155 (MC-A-ARM-BUSH2-R)

M1/P156 (MC-A-ARM-FT-L)

M1/P157 (MC-A-ARM-FT-R)

M1/P158 (MC-A-ARM-RR-L)

4. FILTER ITEMS [X]

Dismiss Vis Help

<No text filter>

Filter PART defns

By Model ... (all models)

By Include file ... (ANY)

By PART TREE ASSEMBLY ... (ANY)

By SET_PART ... (ANY)

By PART type ... (ANY)

By MATERIAL ... (ANY)

By MATERIAL type ... (ANY)

By ELEMENT type ... (ANY)

5. SELECT PART ? [X]

All None [X] Opt

Filter Vis Key_In Sk

(M/L) PART(s) (all models)

M1/P152 (MC-A-ARM-BUSH1-L)

M1/P153 (MC-A-ARM-BUSH1-R)

M1/P154 (MC-A-ARM-BUSH2-L)

M1/P155 (MC-A-ARM-BUSH2-R)

M1/P156 (MC-A-ARM-FT-L)

M1/P157 (MC-A-ARM-FT-R)

M1/P158 (MC-A-ARM-RR-L)

Select ELEM TYPE

Any Elem type

SOLID

BEAM

SHELL

DISCRETE

MASS

ACCELEROMETER

Using Object Selection Menu

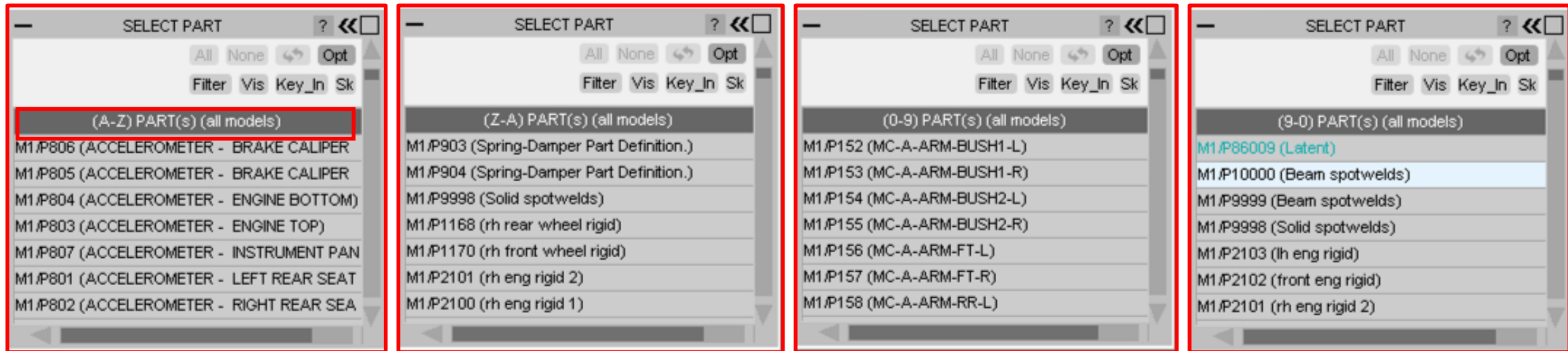
Objects can be selected in several ways in PRIMER. E.g. from PART > MODIFY...

1. Parts can be picked straight away from the graphics window – click on a part, or drag across an area. *(The quick-pick selection box turns blue to tell you the cursor function is controlled by a menu)*
2. The **VIS** option gives further options for selecting visible entities: “ALL” or by screen polygon.
3. Alternatively, click on items in the list. Use shift or control to select multiple items.
4. To refine the listing, click **Filter**. For example, to show only parts that are used for solid elements, select **element type** then **shell**.
5. A further alternative is to choose **KEY_IN** & type in the entity ID.

Using Option Selection Menu

Object menu order

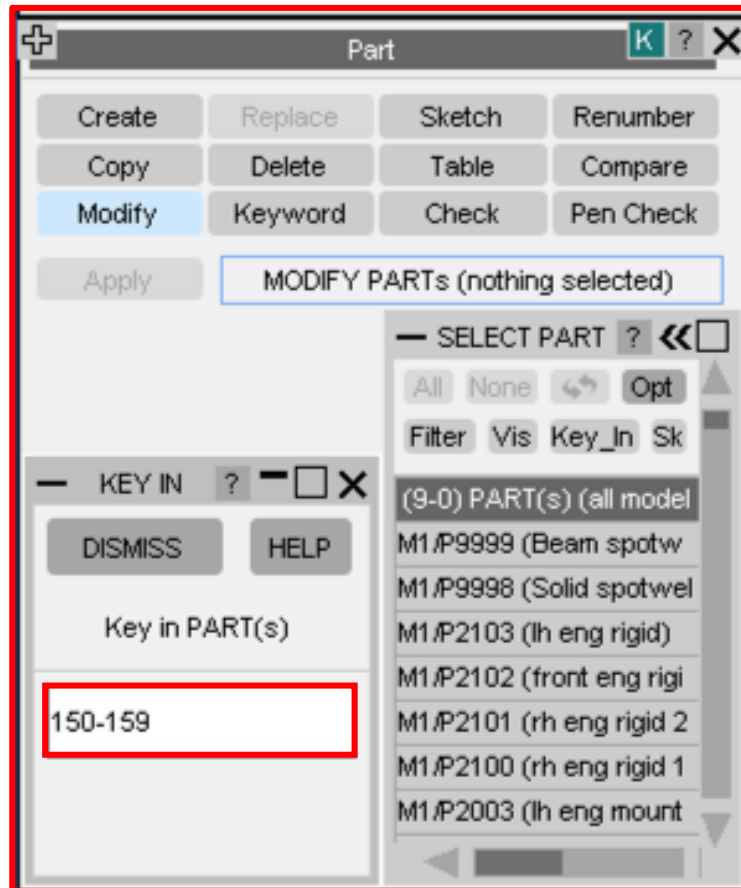
Clicking on the dark grey bar toggles between original order => A-Z => Z-A => 1-9 => 9-1 => original order



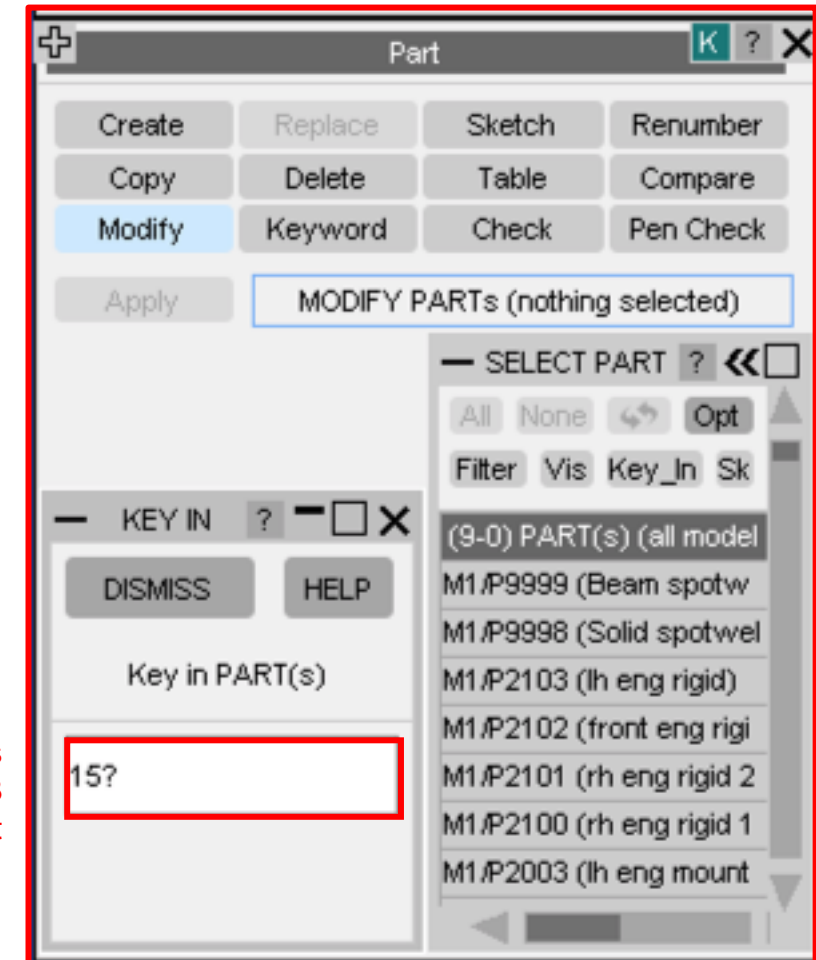
Using Option Selection Menu

Keyword Editor – KEY_IN

“KEY_IN” can be used to type in selections. Ranges and wildcards can be used



Select all parts with ID's between 150 and 159 inclusive

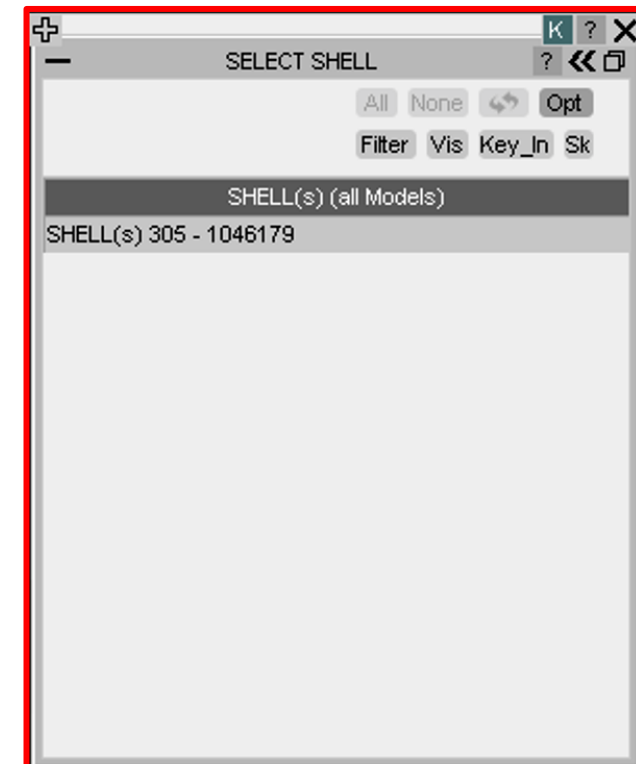
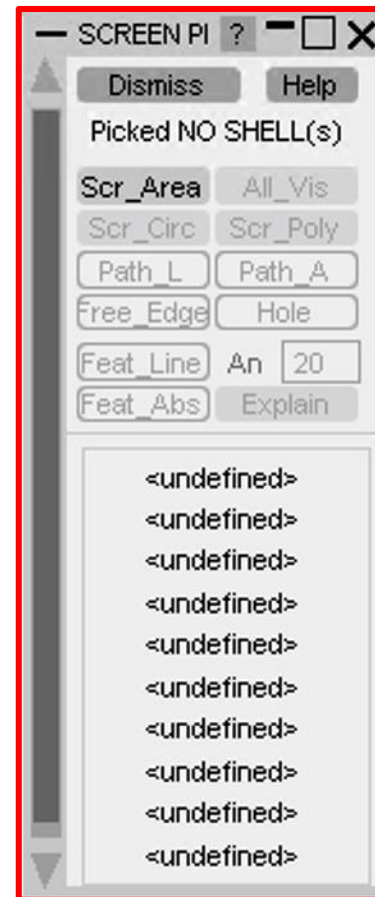
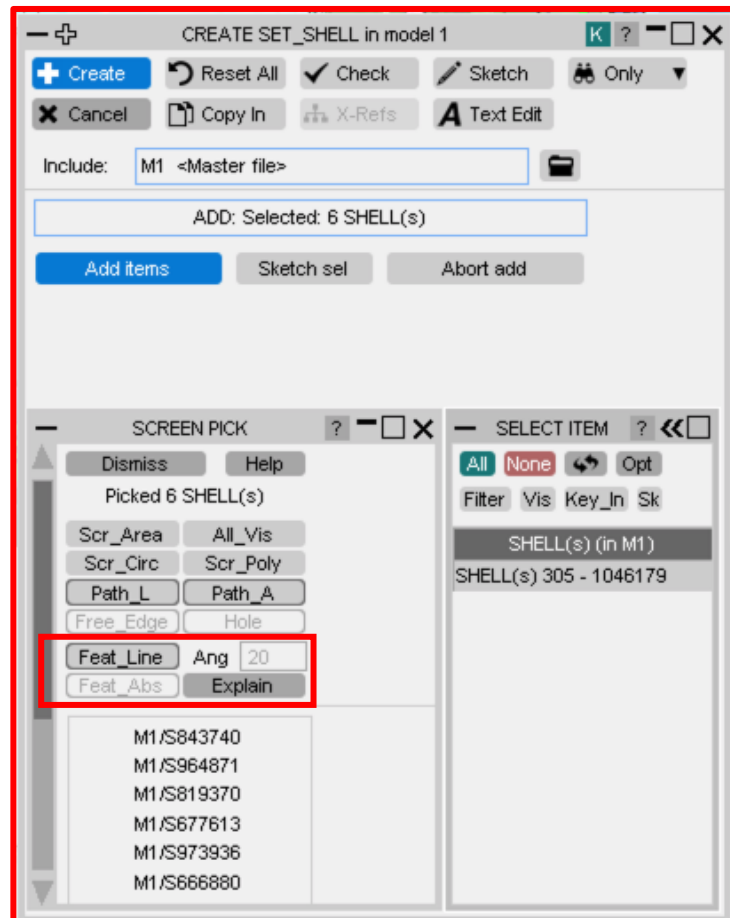


Select all parts with ID's 3 digits long that begin with "15"

Using Option Selection Menu

Keyword Editor – VIS – FEATURE LINE

Useful for selecting shells from a seed shell up to a break angle – e.g. all shells on a flange



Cursor Action Control



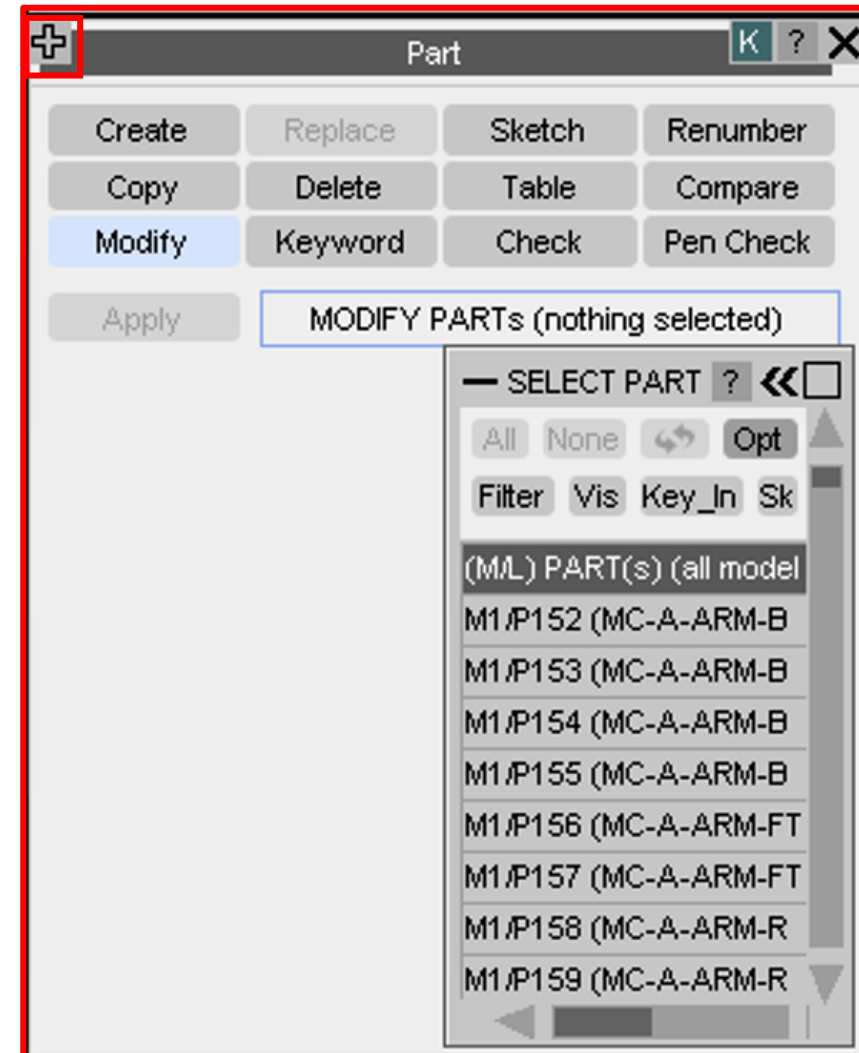
Cursor action = Quick-pick

- Return to Quickpick mode using shortcut Q, or dismiss the menu that is currently controlling cursor action

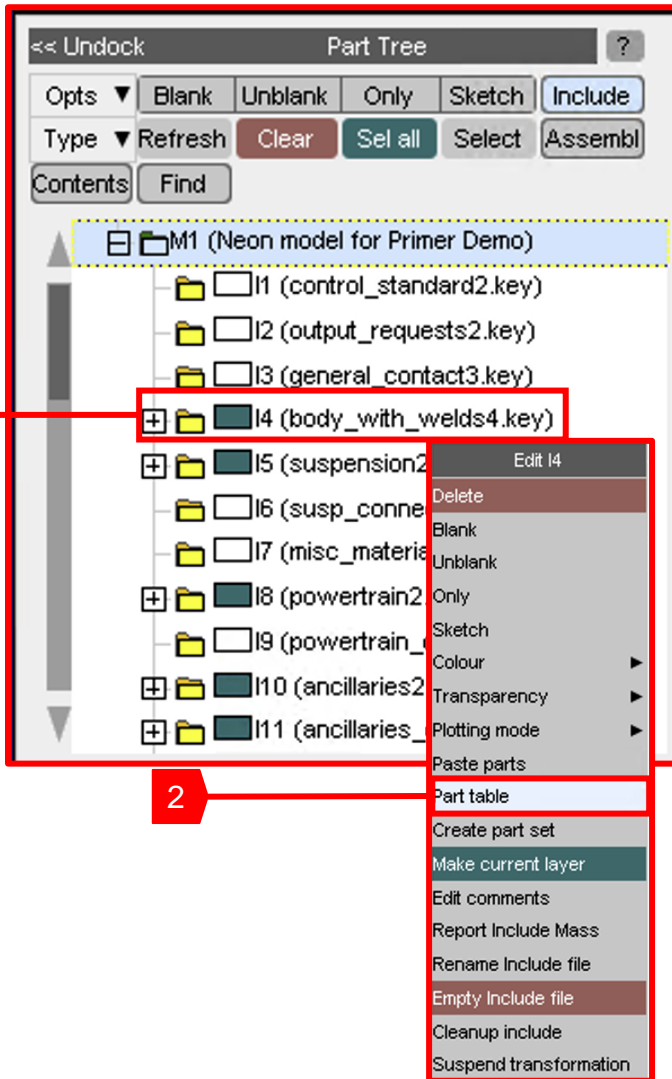


Cursor action = menu pick

- E.g. PRIMER is expecting you to pick a Part for the Modify menu
- To return to picking a part for the Modify menu (e.g. after temporarily using Quick-pick), click on the cross at top-left of the menu



Part Table



- ## Part Table
1. In the Part Tree, right-click the include file **body_with_welds4.key**
 2. Choose "Part Table" option
 3. In the **View** menu, turn on *Mass (Dyna Part)*, *Mass (Dyna Added)*, *Sect NIP* and *Sect Elform*.
 4. Turn off *Part Type* and *Section ID*.

The screenshot shows the 'PART TABLE' window. A red box labeled '3' points to the 'View...' button. The 'View' menu is open, showing options: ☒ Part ID, ☒ Mass (Dyna Part), ☒ Mass (Dyna added), ☒ Sect Elform, ☒ Sect Gauge, ☒ Sect NIP, ☐ Part Type, and ☐ Section ID. The table below shows the resulting data.

Table Ch	Model	Mass (Assign lum	Mass Iner (XX YY	Mat Yield	Part cont CPARM8	Smallest TS
<input checked="" type="checkbox"/> Part ID	Blanking	Mass (Component	Mass Iner (XY XZ	Merge Status	Part Inertia	Stamped part
P353	Colour	Mass (Lumped def	Mat CON1	Numel	<input checked="" type="checkbox"/> Part title	Style
P354	Composite	Mass (NRB)	Mat CON2	Part Contact	Part type	Transparency
P355	Comp. num. layers	Mass (NS)	Mat Density	Part cont FS	Sect ID	Save Settings
P416	EQOS ID	Mass (struct)	Mat Encrypted	Part cont FD	<input checked="" type="checkbox"/> Sect Elform	Dismiss
P362	HGLS ID	Mass (transferred	Mat Fail strain	Part cont DC	<input checked="" type="checkbox"/> Sect Gauge	Unset all
P414	HGLS Type	Mass (Dyna added	Mat Modulus	Part cont VC	<input checked="" type="checkbox"/> Sect NIP	Reset
P410	HGLS Coeff	Mass (%Added)	Mat Title	Part cont OPTT	Sect Title	
P391	Include	Mass C of G	Mat Type	Part cont SFT	Sect SHRF	
P389	CH-RAILS-FT-L-	223	2	3	0.000903256	0.003809
P311	CH-A-PILLAR-B	153	2	3	5.43483e-05	0.00347552
P352	CH-CBN-FIREW	194	2	3	0.000228272	0.00344807

Part Table

Dismiss	View...	Refresh	Write...	Clear	Sel all	Show all	Total Parts: 171 (171 selected)
Table Changes:		Undo	Apply	Select	Show sel	M1 Mass: 1.2203 CofG: 2760.1, 34.392, 636.31	
Part ID	Part title	Mat ID	Sect Elform	Sect NIP	Mass (Dyna add) [0.0142823]	Mass (Dyna Pa) [0.205303]	Sect Gauge
P353	CH-CBN-FLOOR	195	2	3	0.000394783	0.0131394	0.705000
P354	CH-CBN-FLOOR	196	2	3	9.26798e-05	0.0122001	0.706000
P355	CH-CBN-OUTER	197	2	3	0.00028267	0.0113474	0.829000
P416	CH-ROOF	250	2	3	3.09549e-05	0.00684239	0.702000
P362	CH-CBN-SILL-B	204	2	3	2.13682e-05	0.00637916	1.701000
P414	CH-RAILS-U-RR	248	2	3	4.8488e-05	0.00516152	1.916000
P410	CH-RAILS-U-RR	244	2	3	4.84855e-05	0.00515994	1.916000
P391	CH-RAILS-FT-R-	225	2	3	0.00245228	0.00386516	1.895000
P389	CH-RAILS-FT-L-	223	2	3	0.000903256	0.003809	1.895000
P311	CH-A-PILLAR-B	153	2	3	5.43483e-05	0.00347552	1.611000
P352	CH-CBN-FIREW	194	2	3	0.000228272	0.00344807	0.735000

Part Table

The table now shows the requested data.

1. Click on the column heading "Part Mass".
The parts are now in order of mass, lightest at the top.
2. Click again. Now the heaviest parts are at the top.
3. The total mass is shown on the column heading.

Part Table

Part Table

1. Next, click twice on the column heading “Sect Gauge”, to bring the thickest parts to the top.
2. Click the top part, shift-click down to Part 316 – to select the 6 thickest parts.
3. Right-click on the column “Sect NIP” (number of integration points).
4. Type 5 in the text box. *Now the number of integration points is set to 5 for the selected parts.*
5. To change the data in the model, press **Apply**.
6. Dismiss the table, unblank all parts.

PART TABLE

Dismiss View... Refresh Write... Clear Sel all Show all Total Parts: 171 (6 selected)

Table Changes: Undo Apply Select Show sel M1 Mass: 1.2203 CofG: 2760.1, 34.392 636.31

Part ID	Part title	Mat ID	Sect Elform	Sect NIP	Mass (Dyna add [0.0142823])	Mass (Dyna Par [0.205303])	Sect Gauge
P616	CH-RAILS-U-RR	301	2	5	<undefined>	0.000199242	2.800000
P615	CH-RAILS-U-RR	300	2	5	<undefined>	0.000208763	2.800000
P612	CH-RAILS-U-RR	297	2	5	2.68619e-05	0.00017136	2.800000
P611	CH-RAILS-U-RR	296	2	5	2.68554e-05	0.000171366	2.800000
P317	CH-A-PILLAR-B	159	2	5		0.000100282	2.800000
P316	CH-A-PILLAR-B	158	2	5		0.000100277	2.800000
P614	CH-FENDER-SIL	299	2	3		0.000143157	2.520000
P613	CH-FENDER-SIL	298	2	3		0.00014316	2.520000
P610	CH-RAILS-U-RR	295	2	3	7.5753e-05	0.00277413	2.520000
P609	CH-RAILS-U-RR	294	2	3	7.80346e-05	0.0027737	2.520000
P603	CH-FENDER-SIL	293	2	3	0	0.000396354	2.500000

Change Sect NIP

Sketch

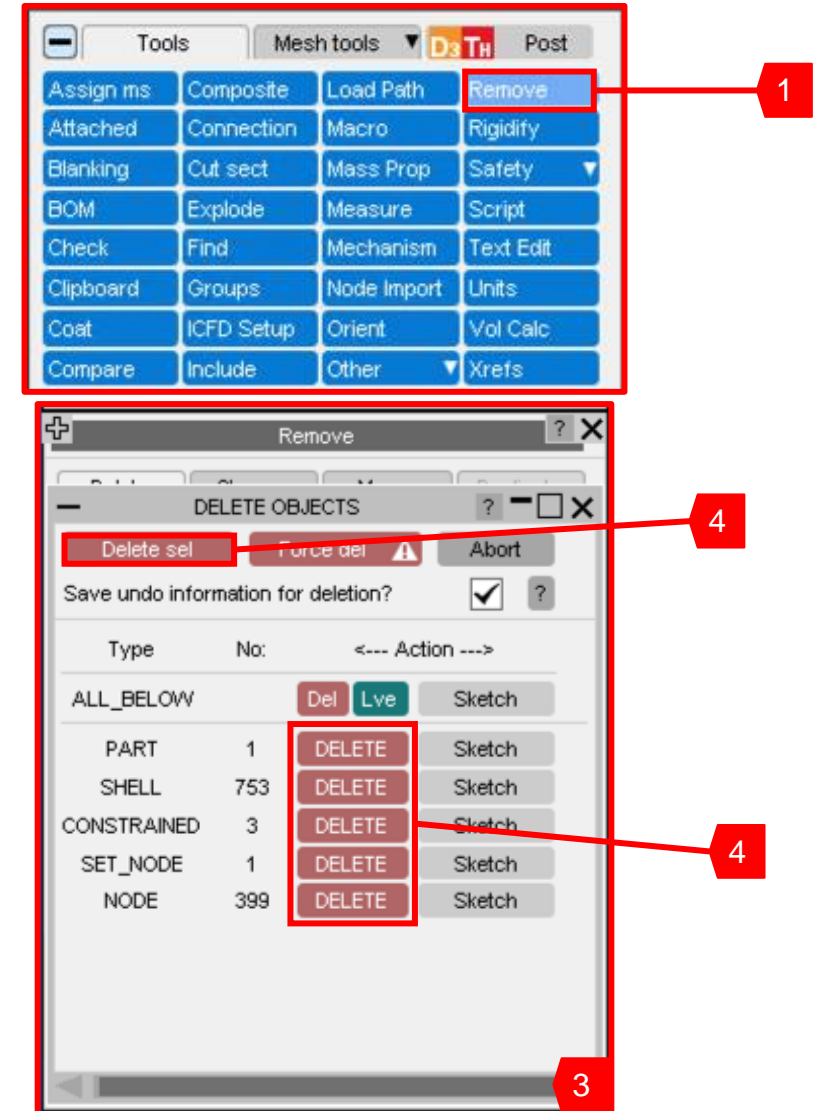
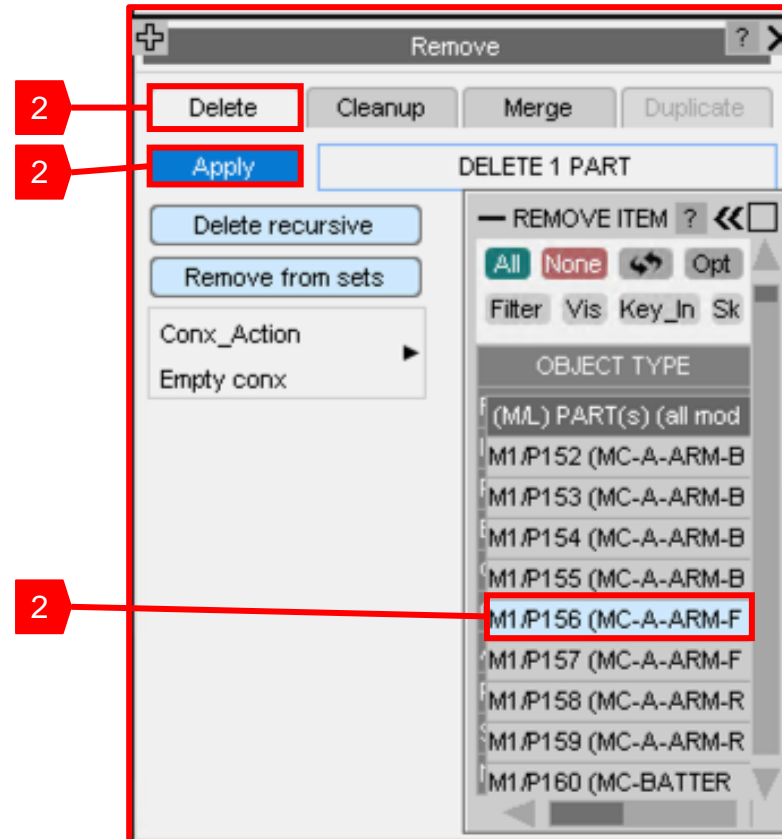
5

Remove

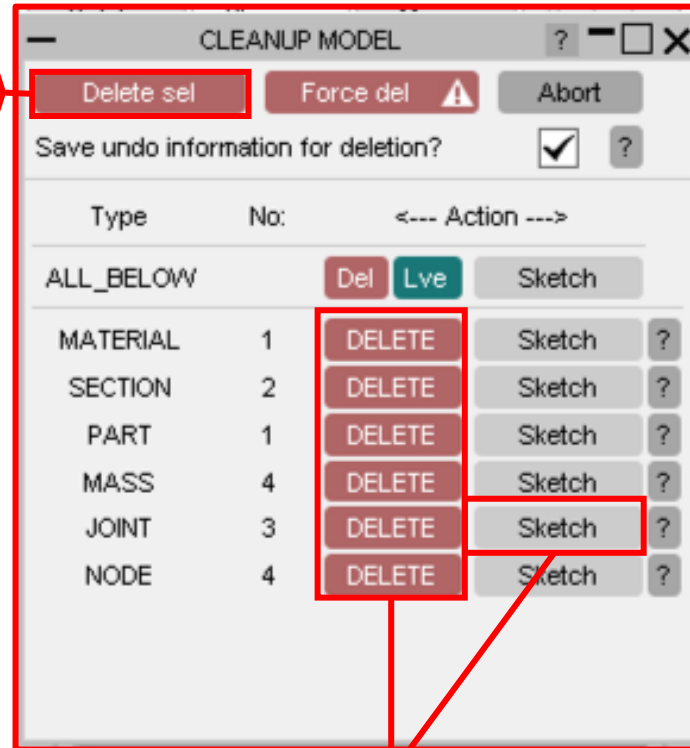
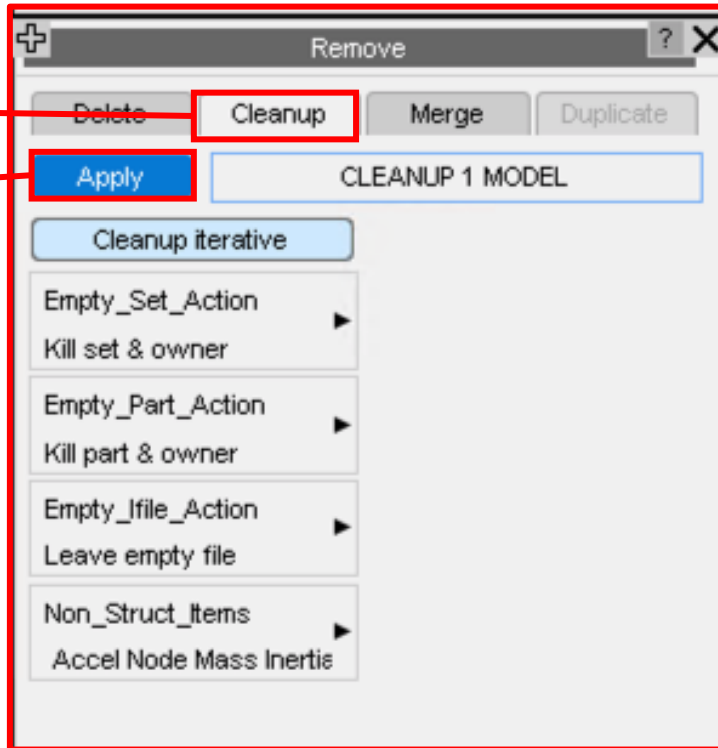
Remove

The remove function can be used to delete entities whilst maintaining the integrity of the model. First, unblank the whole model (U).

1. Go to **Tools – Remove**, to display the remove menu.
2. The **Delete** sub-menu is shown by default. Select Part 156 & **Apply**.
3. The next menu shows what entities PRIMER decides should be deleted together with Part 156. *In this this example some Constrained entities are also highlighted – if left in the model these would cause an error.*
4. The user can Sketch each entry on the list and choose which types to leave undeleted. In this case delete all using **DELETE_SEL**.



Remove



Remove (cont)

The cleanup function identifies data that is no longer needed in the model.

1. Select **Cleanup** sub-menu to remove entities that are no longer used.
2. Click **Apply**.
3. In the next menu we see what PRIMER recommends to be deleted – **Sketch** the joints to see their location. *If these joints are left in the model LS-Dyna will terminate with an error.*
4. Delete all using **DELETE_SEL**.

Orient

6 ways to Orient are available

2

5

3

4

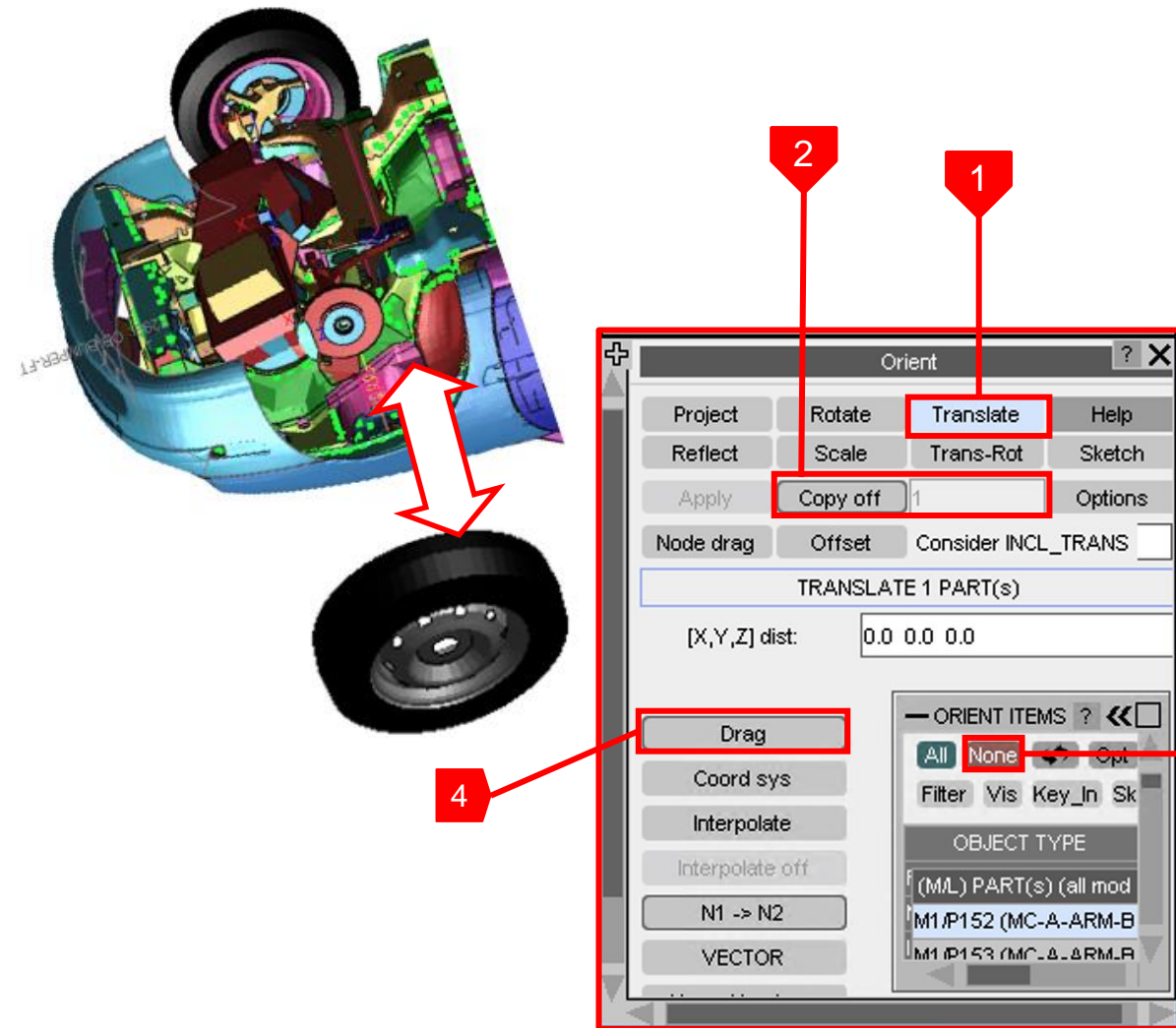
e.g. Trans-rot maps one node triad onto another

Orient

Using the **TOOLS – ORIENT** menu we can move any entity in the model to a different position and (optionally) make copies of the entity during the process

1. Delete one wheel & tyre using the REMOVE function (previous slide)
 2. Open the **ORIENT** menu and choose **Reflect**
 3. Pick a node on the centre-line of the vehicle as a mirror point in the Y-Axis
 4. From the list on the right choose **Part**, drag an area over parts to copy
 5. Select the **Copy on** feature, **1 copy**
 6. Press **Apply** – PRIMER will create a new wheel with new part IDs, elem IDs etc.
- Numbering of the new entities can be specified here, or renumbered later using **MODEL – RENUMBER CONTENTS - SELECTION**

Orient - Dragging

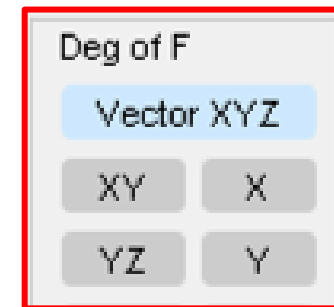


Orient

Orientation can also be done by dragging.

1. Switch the Orient method to **Translate**.
2. Switch off Copy
3. The previously selected parts are still selected in the Object menu. Deselect these using the **NONE** button, then pick the newly-reflected parts. Dismiss the screen-pick menu which is covering some of the Orient menu buttons.
4. Press **Drag**, select option **Y**
5. Drag the wheel into a new position.
6. Press **Apply** – PRIMER will leave the parts in their new position.

Numbering of the new entities can be specified here, or renumbered later using MODEL – RENUMBER CONTENTS - SELECTION



Tabs

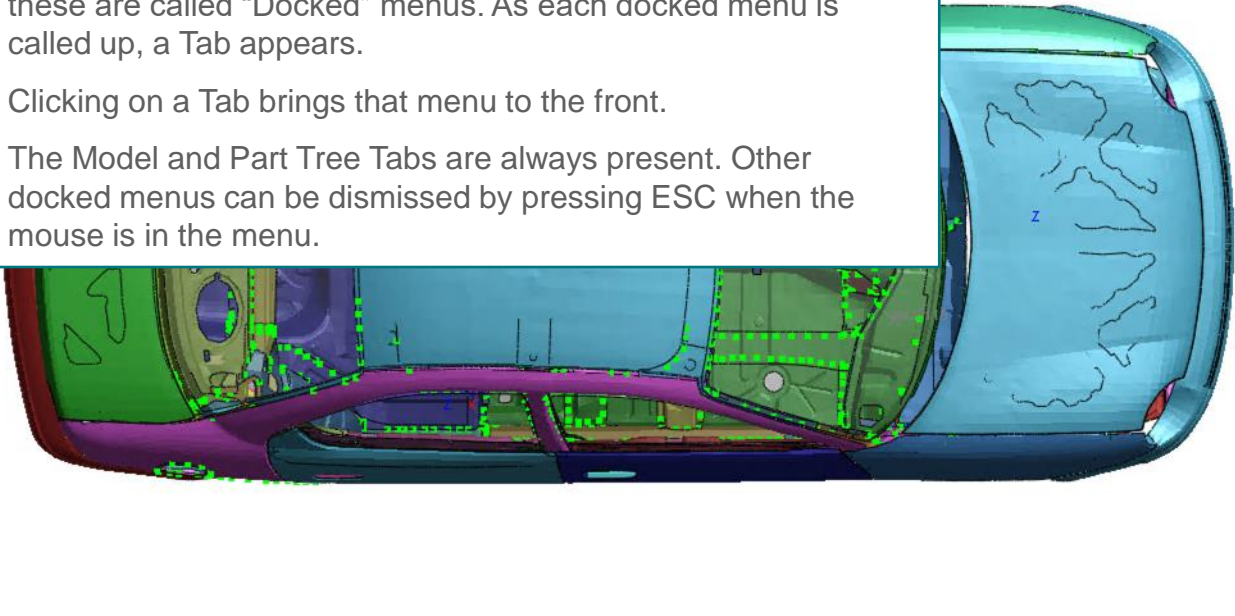
PRIMER 17.0 - 64 bit (build 26840), Licensed to : Ove Arup (AT&R London)

File Keywords Tools Display Images Viewing Options Help Pick NPNT PP Undo

PRIMER: M1: Neon model for Primer Demo

Tabs

1. Many of PRIMER's menus appear in the bottom-right corner – these are called “Docked” menus. As each docked menu is called up, a Tab appears.
2. Clicking on a Tab brings that menu to the front.
3. The Model and Part Tree Tabs are always present. Other docked menus can be dismissed by pressing ESC when the mouse is in the menu.



Y
X

Picked: P236
Short-cut key '' not in use.

Manual CT SI Node plot U H Sh Save P Lock
Stop Timestep ▶ Init Vels (Tr) AC Zoom CN All
Tidy ▶ +XY +YZ +XZ +SO R Views Rev
? -XY -YZ -XZ -SO ← → ↶ ↷ Ent

Tools Mesh tools D T Post

Assign ms	Composite	Load Path	Remove
Attached	Connection	Macro	Rigidify
Blanking	Cut sect	Mass Prop	Safety
BOM	Explode	Measure	Script
Check	Find	Mechanism	Text Edit
Clipboard	Groups	Node Import	Units
Coat	ICFD Setup	Orient	Vol Calc
Compare	Include	Other	Xrefs

Volumes I & II Volume III

AIRBAG	DATABS	INITIAL	RAIL
ALE	DEFINE	INTEGRN	RIGIDWALL
BOUND	DEF_2_RG	INTRFCE	SECTION
CASE	ELEMENT	LOAD	SENSOR
COMMENT	EOS	MAT	SET
CONSTR	FATIGUE	NODE	TERMIN
CONTACT	FREQ	PARAM	
CONTROL	HOURLGL	PART	
DAMPING	INCLUDE	PERTURB	

Model Part tree Measure

M1:Main file

Measure

Point to Point	Point angle	Node to Node	Node angle
Nodal coord	Curve length	Node to Part	Part to Part
Node Curve I	Distance Plot	Node to Plan	

Node to Node - pick nodes

Done Reject last Refresh

<none>

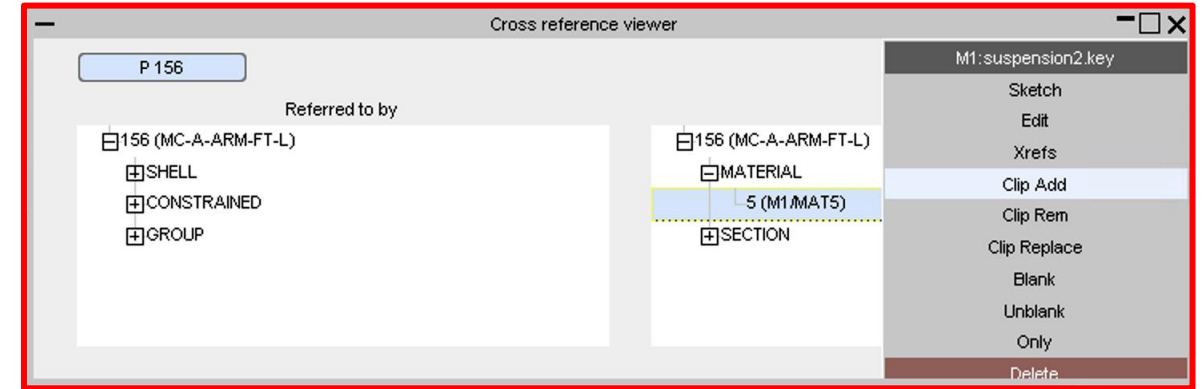
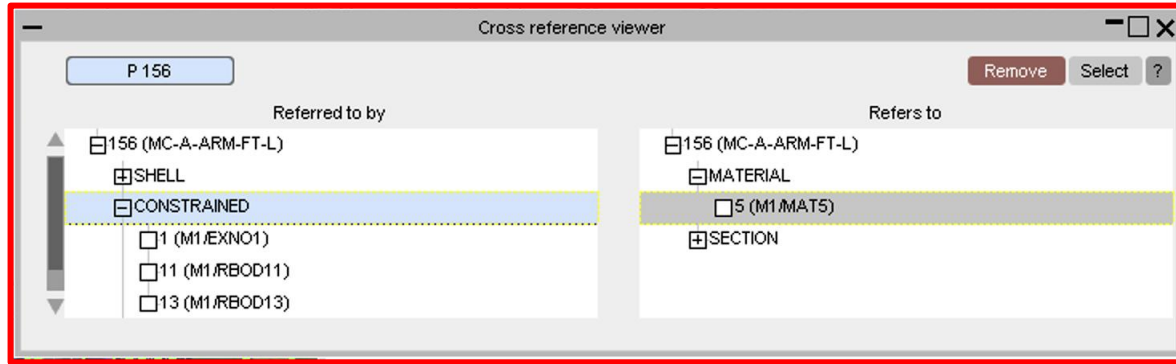
<none>

Vector:

☐ Unit vector

Distance: Undo move

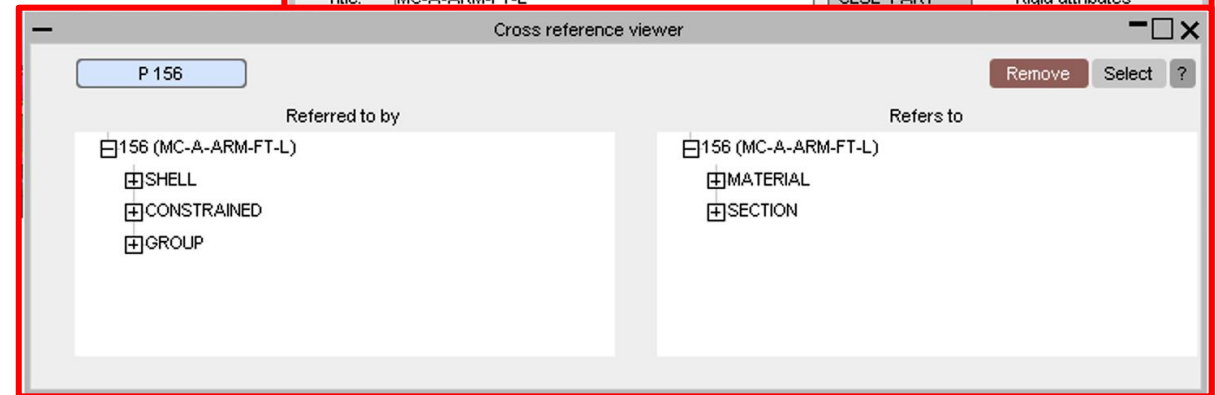
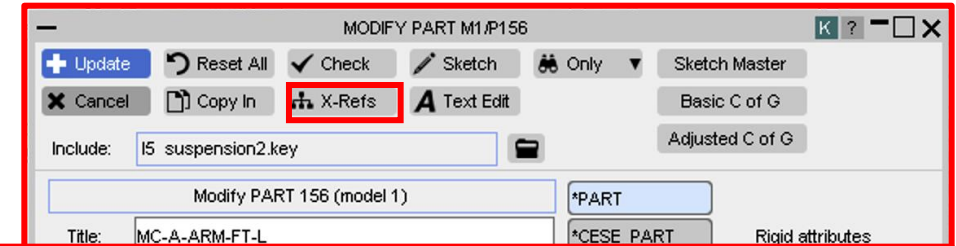
X-Refs



X-Refs (Cross reference)

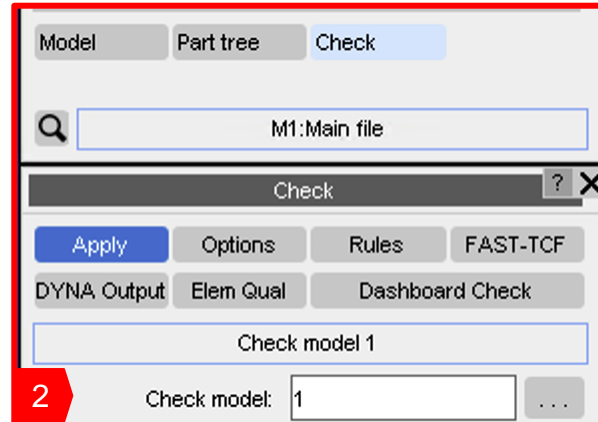
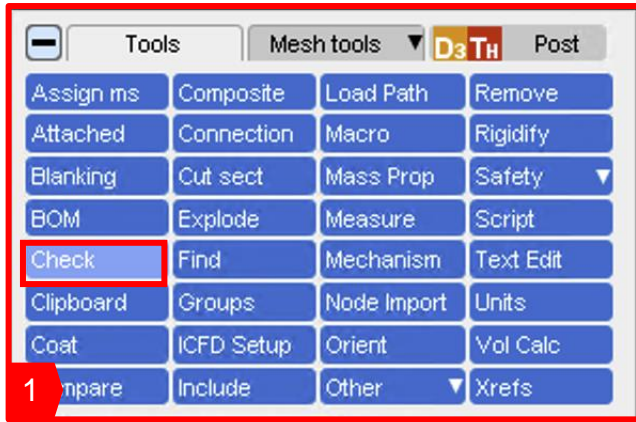
Linkage (cross references) between entities may be viewed using Xrefs.

1. From parts menu, **Edit** part 156 or 157 (one of these may have been deleted!).
2. **VIEW_XREFS** will open the Cross reference viewer
3. The left menu contains entities that refer to this part – look at the CONSTRAINED entities that refer to this part
4. The right menu contains entities that this part refers to – right click & open – we can see that only 1 parts references this material



Model Checking

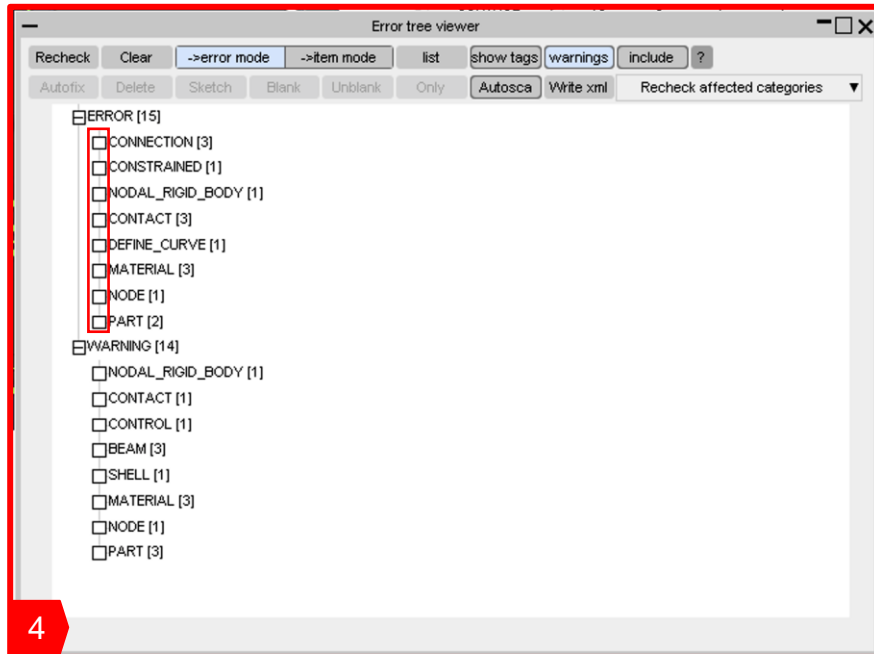
Checking Models



3

The CHECK DYNA3D MODEL dialog box is open, showing the 'Check (model 1)' button and the 'Check model 1' button. A red number 3 is next to the 'Check model 1' button.

Entity type	(No.)	# errors	# warn	# fixable
Grand Totals		442	446	423
AIRBAG	4	0	0	0
ASSIGN MASS	1	0	0	0
CONNECTION	2588	405	0	325
CONSTRAINED	1391	9	8	8
CONTACT	3	3	1	2
CONTROL	10	0	1	1
DATABASE	20	0	0	0
DEFINE	20	1	0	0
ELEMENT	722251	0	317	1
GROUP	3	0	0	0
HOURLASS	3	0	0	0
INCLUDE FILE	40	0	0	0
MATERIAL	321	11	82	82
NODE	296419	3	2	2
PART	375	10	35	2
SECTION	324	0	0	0
SET	128	0	0	0



Checking a Model:

Delete the model and re-read [demo_car4.key](#).

1. Click **Tools** – **Check** to display checking options.
2. Select the model to check, click **Apply**.
3. The checking summary, showing the number of errors and warnings, is displayed. *The distinction between “error” and “warning” is somewhat arbitrary – not all of the “errors” would cause LS-DYNA to terminate.*
4. The Error Tree viewer is also displayed. Each category may be expanded or contracted by clicking on **[+]** or **[-]**

Checking Model

The screenshot shows the 'Error tree viewer' window. The toolbar includes buttons: Recheck, Clear, ->error mode (selected), ->item mode, list, show tags, warnings, include, ?, Autofix, Delete, Sketch, Blank, Unblank, Only, Autosca, Write xml, and Recheck affected. The error tree is expanded to show the following structure:

- ERROR [15]
 - CONNECTION [3]
 - F spotweld is too close to another spotweld (325) *Faults with connections can be fixed using Connections Table*
 - NOT STICKING - Constrained contact not sticking - constraint clash (25)
 - NOT CONNECTED - Spotweld/adhesive is not (fully) in tied contact (55)
 - CONSTRAINED [1]
 - RIGID_BODIES: Separation of rigid bodies exceeds user defined tolerance (3) *User-defined check to catch possible modelling error*
 - NODAL_RIGID_BODY [1]
 - NRB_SPC: Dimensions of nodal rigid body exceed user defined maximum (6) *User-defined check to catch possible modelling error*
 - CONTACT [3]
 - F Constrained TIED/SPOTWELD contact contains rigid part(s) (1)
 - Connection contact does not tie all nodes on slave side (1)
 - F Another constraint unsticks potentially tied nodes in constrained contact (1)
 - DEFINE_CURVE [1]
 - DEFINE_CURVE referenced but not defined (1) *Missing data – will stop LS-DYNA*
 - MATERIAL [3]
 - MAT_24 etc: <LCSS> curves starts at yield stress $Y \leq 0.0$ (4)
 - MAT_24 etc: <LCSS> curve begins at strain $X > 0.0$ (6)
 - MAT_24 etc: table curves cross at strain below failure (EPPF) (1)
 - NODE [1]
 - node on rigid part clashes with spotweld (constrained contact) (3) *“multiple constraint” error*
 - PART [2]

Read the descriptions of the different errors...

Faults with connections can be fixed using Connections Table

User-defined check to catch possible modelling error

User-defined check to catch possible modelling error

Contact quality checks

Missing data – will stop LS-DYNA

Faults with material data that (in our experience) can cause trouble in LS-DYNA

“multiple constraint” error

Checking Models

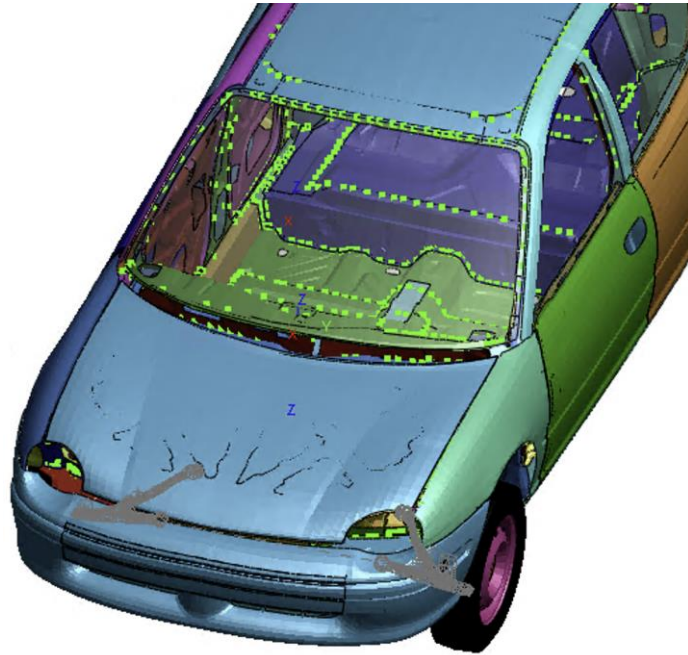
Error tree viewer

Recheck Clear ->error mode ->item mode list show tags warnings include ?

Autofix Delete Sketch Blank Unblank Only Autosca Write xml Recheck affected categories ▼

- ERROR [15]
- WARNING [14]
 - NODAL_RIGID_BODY [1]
 - CONTACT [1]
 - F item does not have explicit label, others do. May cause LSDA/Ascii problem. : Assign an explicit label (optional id) to all items of this type (1)
 - CONTROL [1]
 - F SHELL: Sorting flag for degenerate quads <esort> is unset : Set sorting parameter <esort> to 1 (1) Recommended parameter is not the default
 - BEAM [3]
 - Spotweld beam length is greater than maximum allowed size (232)
 - Beam is part of a spotweld which is longer than maximum allowed length (56) (Spotweld beam) element quality checks
 - Spotweld beam length is less than minimum allowed size (28)
 - SHELL [1]
 - F Duplicate Shell - overlaps existing shell (same part) : Delete overlapping (higher label) element(s) (1)
 - MATERIAL [3]
 - F MAT_24 etc: curve discretisation may lose data : Increase LCINT on CURVE(S) (80) Material type 24 - warnings
 - F MAT_24 etc: yield stress from <LCSS> curve over rules SIGY : Modify value of SIGY for material using LCSS/data value (1)
 - F MAT_24 etc: Stress (Y) characteristic goes -ve at strain < failure (EPPF) : Set failure strain <epff> to prevent -ve stress (1)
 - NODE [1]
 - F node is not in same include as element : Move node into same include as element (2)
 - PART [3]
 - elements of rigid part are not continuously meshed (32) Discontinuous rigid body – could be a mistake?
 - F Part contains elements in different include file : Move all elements and nodes of part into same include as part (1) Model badly organised
 - part is empty - no elements or nodes (1)

Checking Models



Checking with the Error Tree Viewer:

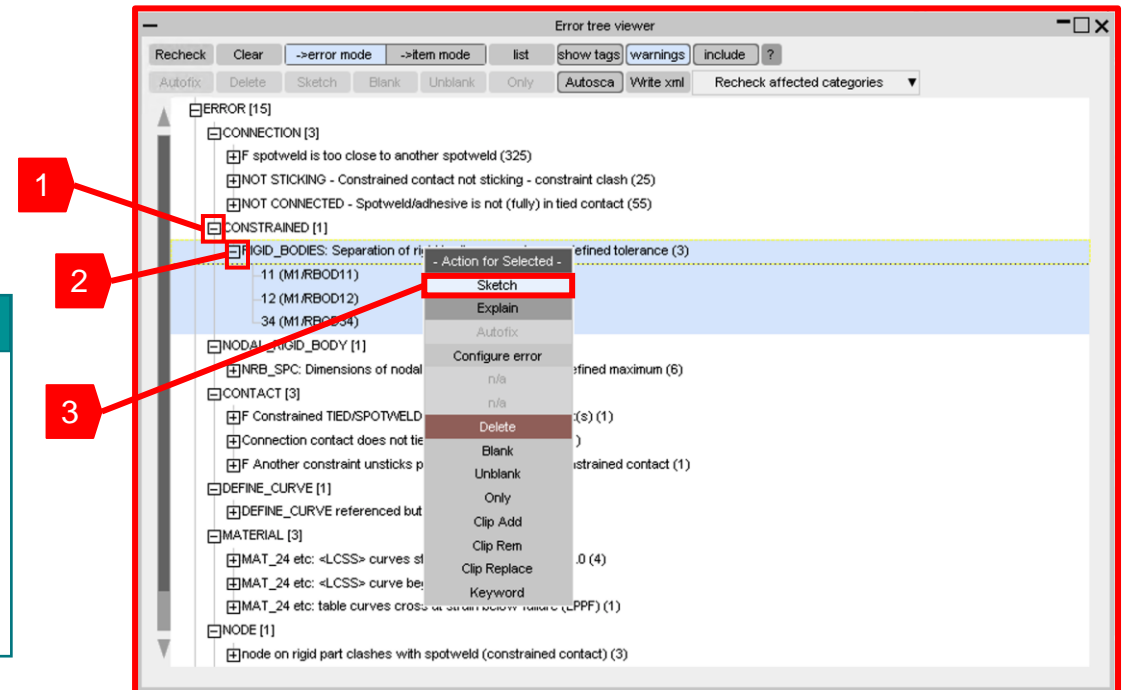
1. Expand the first error category in the tree:
[+] **CONSTRAINED** then [+] **RIGID BODIES**

This is a *user-defined check* for the distance between merged rigid bodies – it could show up a modelling error that the wrong rigid bodies were merged.
Later we will learn how to set preferences for user-defined error checks.

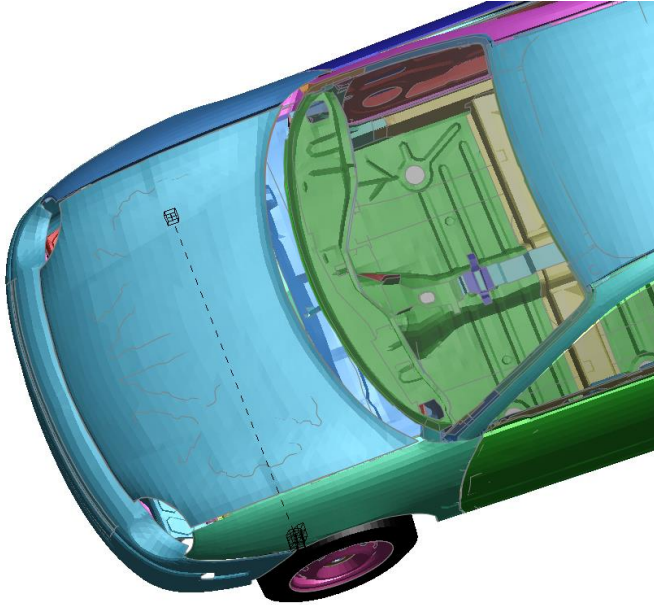
Error Tree Viewer contd:

2. Right-click on the top error heading (**RIGID BODIES**). The pop-up menu give options that apply to all the entities listed under that heading.
3. Click on **Sketch**. You should see several rigid bodies that are subject to a *CONSTRAINED_RIGID_BODIES that fails the *user-defined check*.

Some of these may not be errors – it depends on user defined values – you need to consider each one separately.

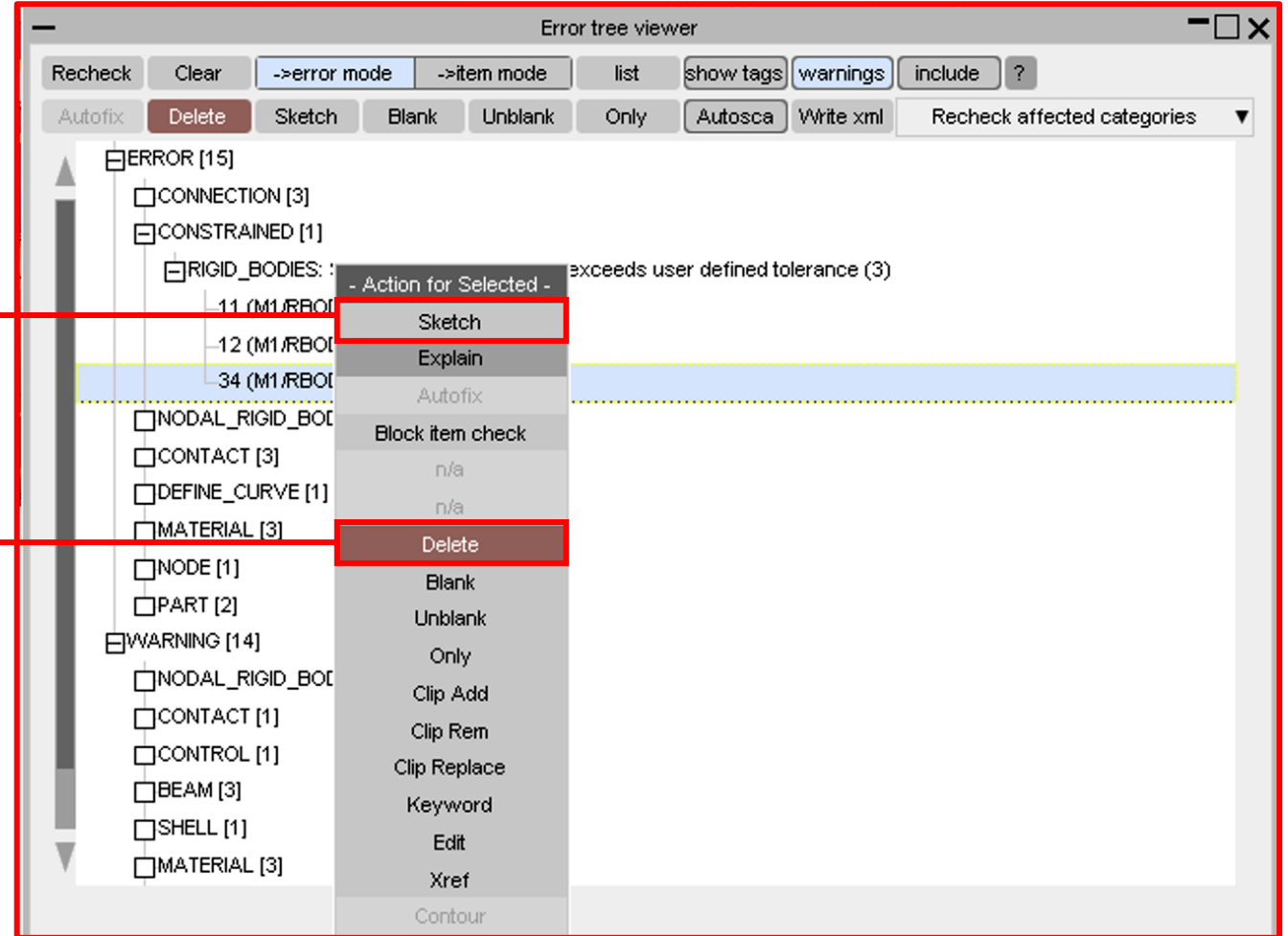


Checking Models

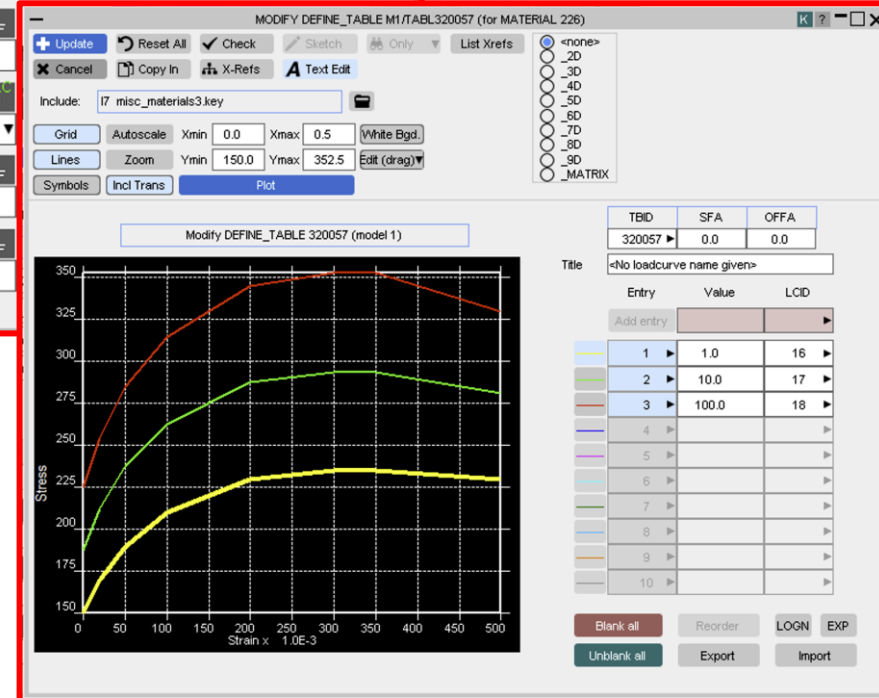
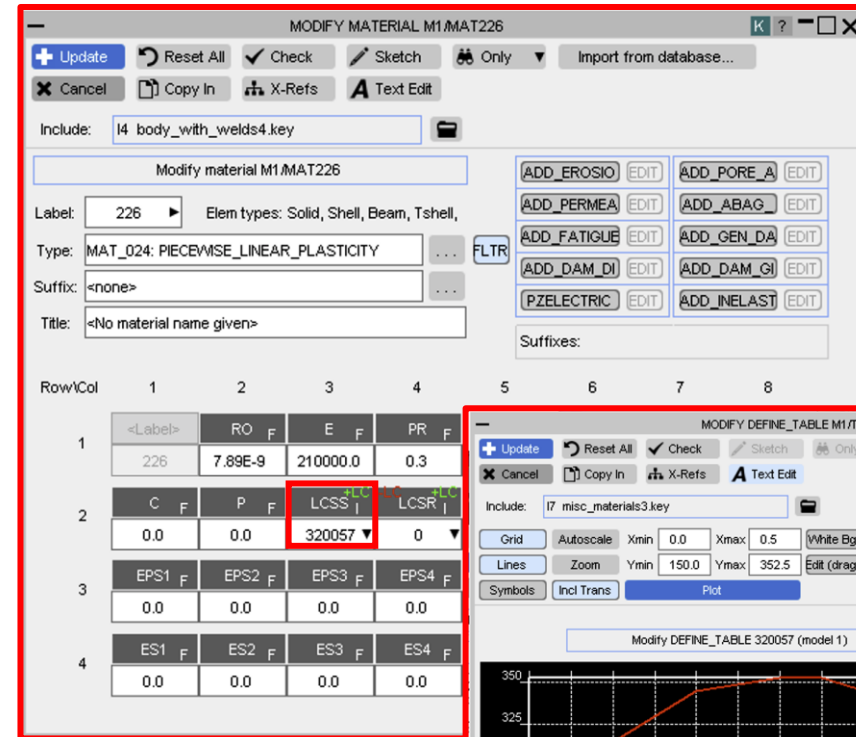
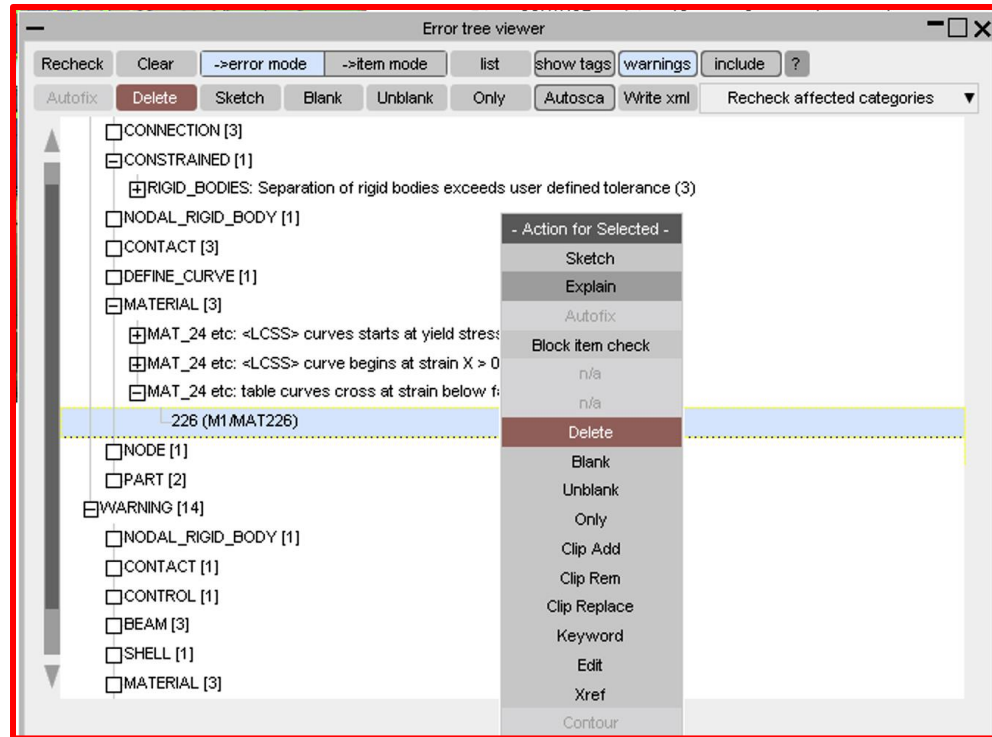


Error Tree Viewer contd:

1. Right-click on the last of the individual entities affected (**RBOD34**). Click on **Sketch**.
2. Rotate the model to see the small rigid bodies that have been merged accidentally – this is clearly an error.
3. Delete the constraint by clicking on **Delete**, then **DELETE_SEL** and **CONTINUE**.



Checking Models



More Checking & Fixing:

1. Right-click on the last **MAT_24** error. Click on **Edit** This will open the **MODIFY MATERIAL** menu for MAT 226
2. Right-click on the stress-strain curve table **LCSS**, **Edit**.
At large plastic strain, the stress-strain curves for different strain rates will cross. This can cause instability.

Checking Models

More Checking & Fixing contd:

3. This can be fixed by making the material fail before the curves cross – set **FAIL**=1.0 (100% plastic strain)
4. Check again using **Check** button
5. Erase the **SIGY**=400 – this is ignored by Dyna when the stress-strain curves start at a different value. **Check** again – should be no errors for MAT 226

MODIFY MATERIAL M1/MAT226

Buttons: + Update, Reset All, **Check**, Sketch, Only, Import from database..., X Cancel, Copy In, X-Refs, Text Edit

Include: I4 body_with_welds4.key

Modify material M1/MAT226

Label: 226 Elem types: Solid, Shell, Beam, Tshell

Type: MAT_024: PIECEWISE_LINEAR_PLASTICITY

Suffix: <none>

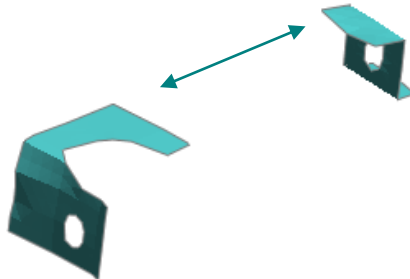
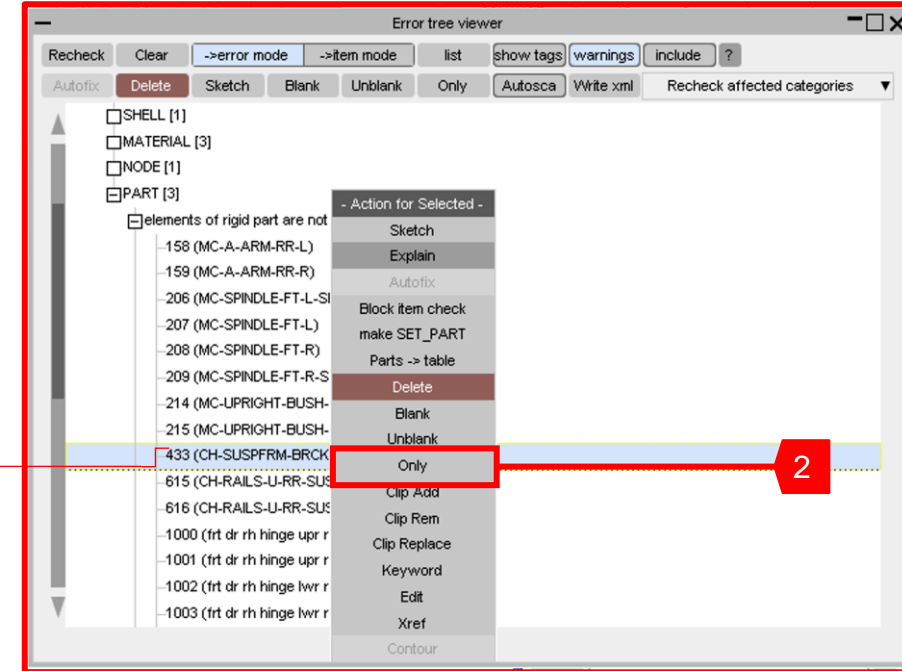
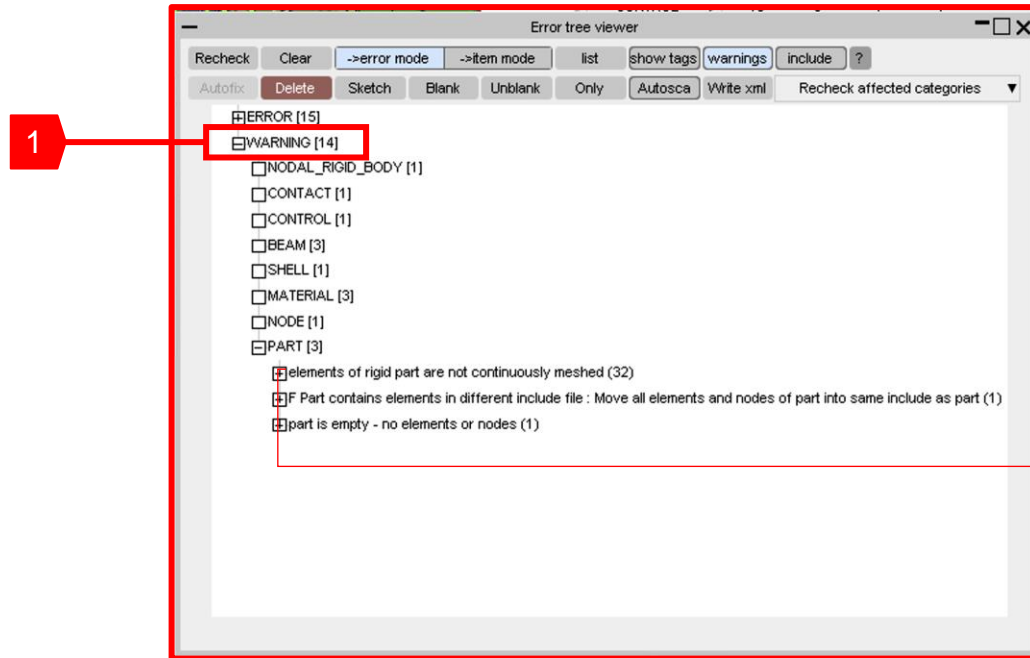
Title: <No material name given>

Buttons: ADD_EROSIO, ADD_PORE_A, ADD_PERMEA, ADD_ABAG, ADD_FATIGUE, ADD_GEN_DA, ADD_DAM_DI, ADD_DAM_GI, PZEELECTRIC, ADD_INELAST

Suffixes:

Row\Col	1	2	3	4	5	6	7	8
1	<Label>	RO F	E F	PR F	SIGY F	ETAN F	FAIL F	TDEL F
	226	7.89E-9	210000.0	0.3	0.0	0.0	1.0	0.0
2	C F	P F	LCSS I	LCSR I	VP F			
	0.0	0.0	320057	0	0.0			
3	EPS1 F	EPS2 F	EPS3 F	EPS4 F	EPS5 F	EPS6 F	EPS7 F	EPS8 F
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
4	ES1 F	ES2 F	ES3 F	ES4 F	ES5 F	ES6 F	ES7 F	ES8 F
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

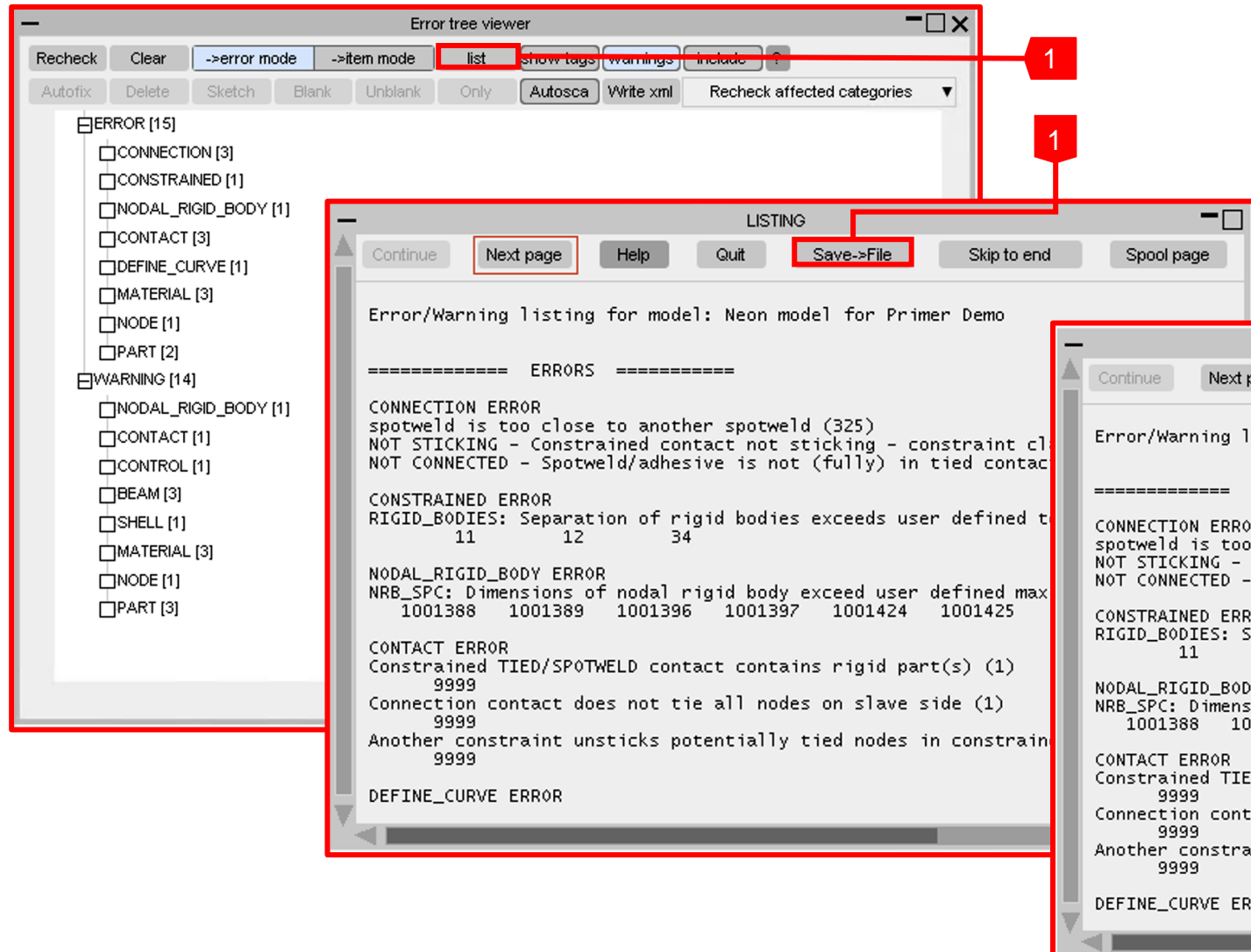
Checking Models



More Checking:

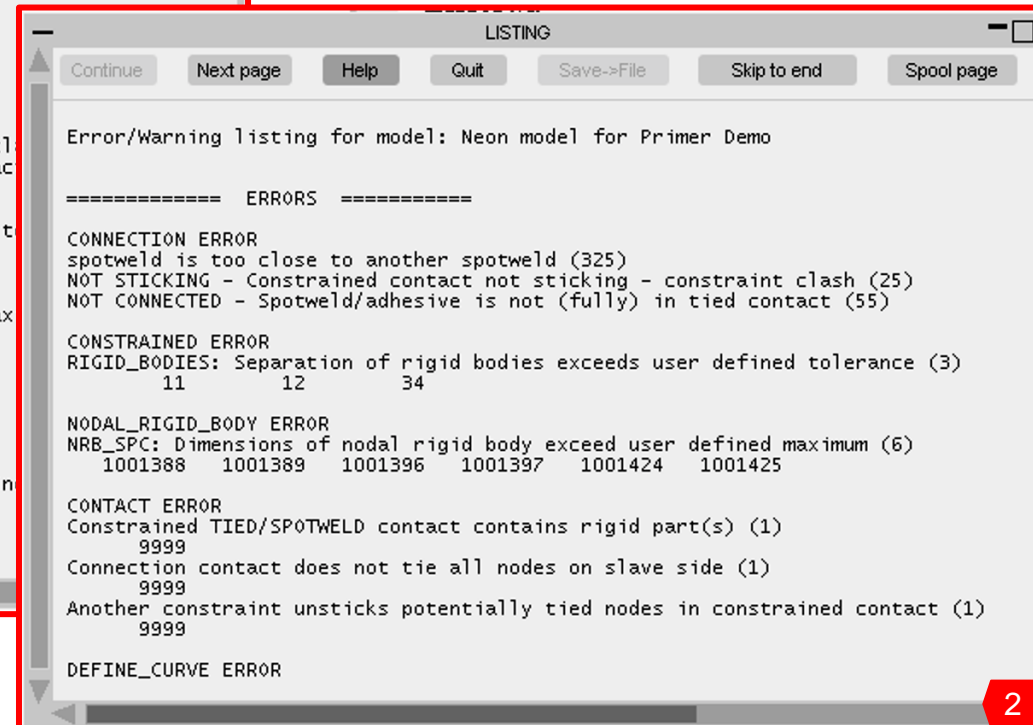
1. Expand the **WARNING** section and look at the first **PART** warning
2. Many parts are listed under this warning – use the right-click **ONLY** option to show only part 433. *This is a quick way to see where the parts are in the model.*

Writing an Error Summary File



Saving the Errors:

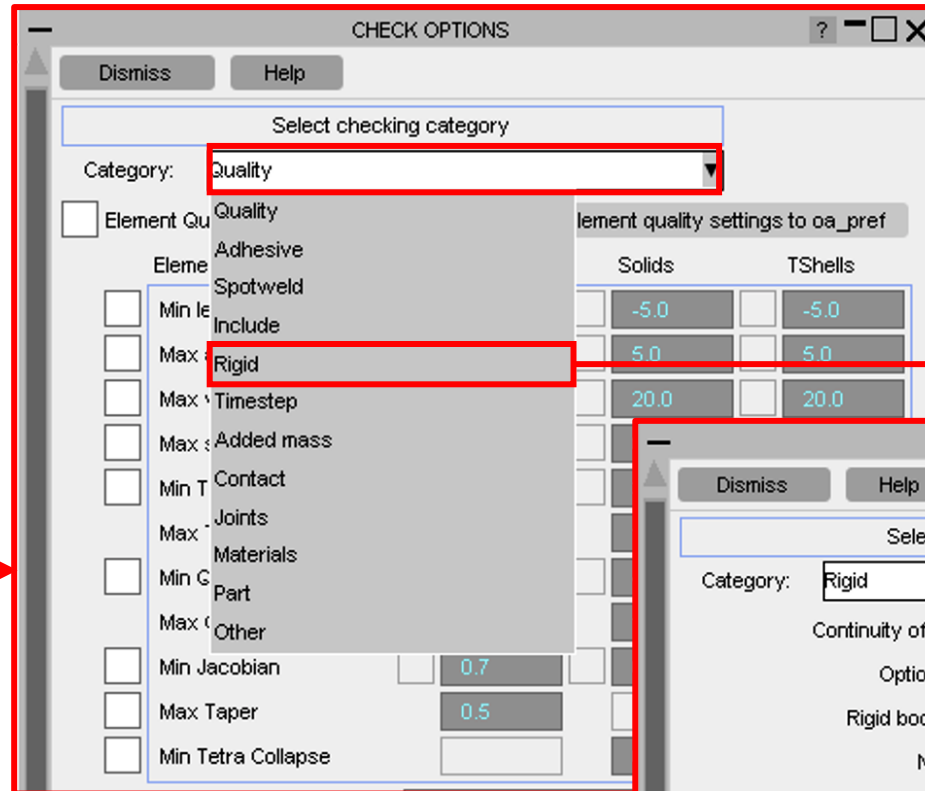
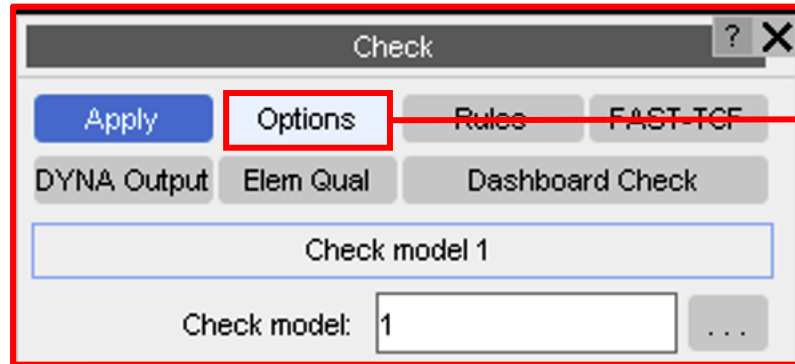
1. Write an error summary file using **List**.
2. View it using a text editor. *This file can also (optionally) be created during model write-out.*



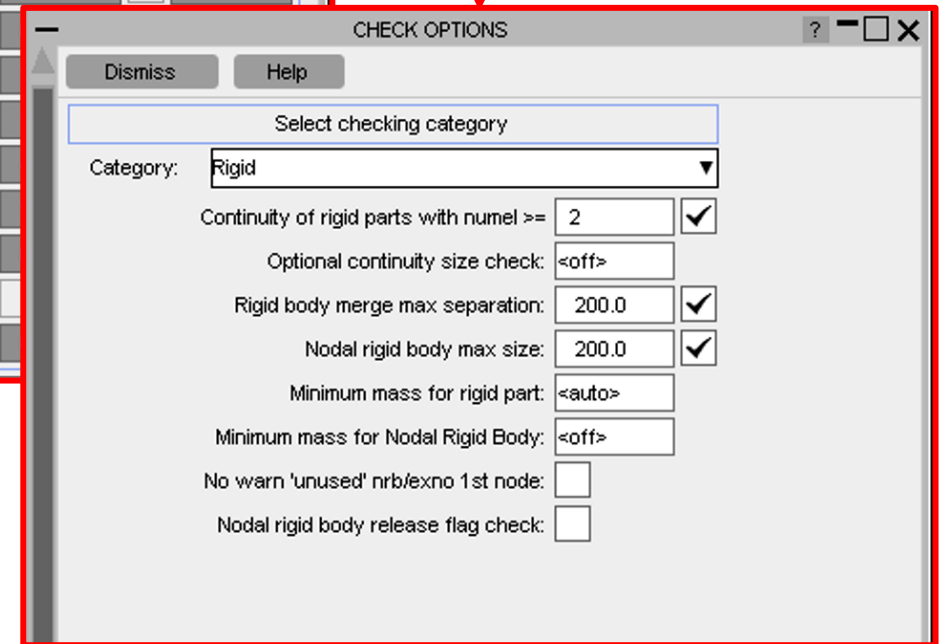
User-defined Checking Options

User Defined Options:

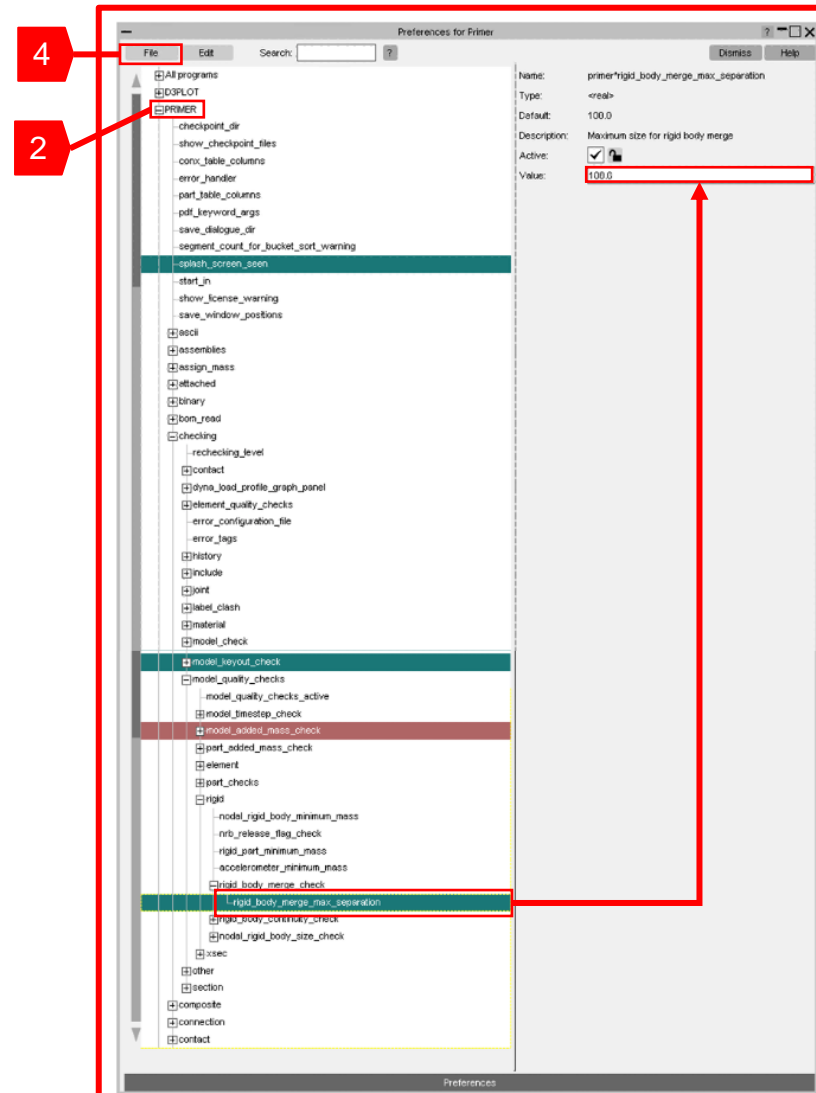
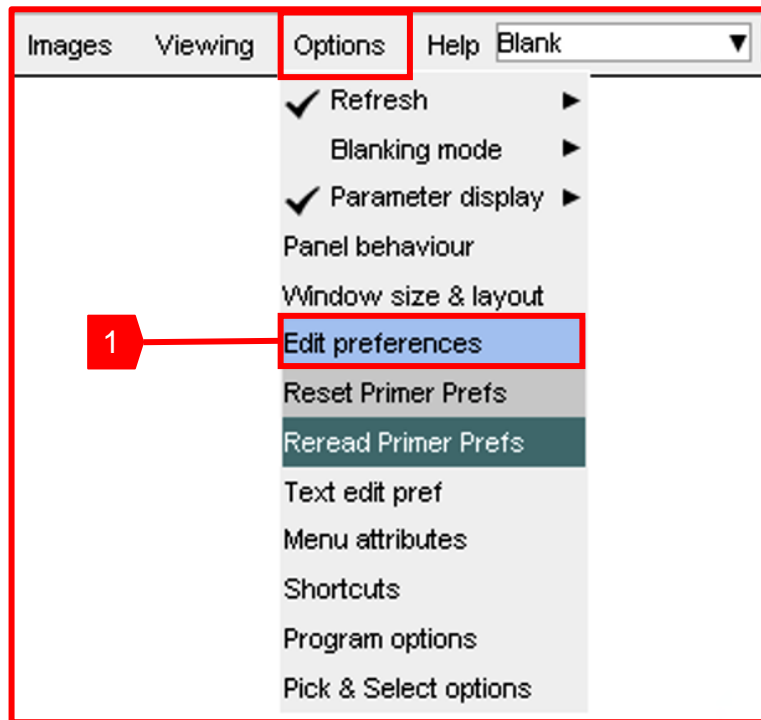
1. To set user-defined values for some of the Error & Warning checks, click **Options** from the main check menu
2. Tick the boxes and set the desired limits – *this will apply to the current PRIMER session only*



Use drop-down to change category



User-defined Checking Options



User Defined Options:

1. For a permanent setting, use **Options, Edit prefs** from the top tool bar. *Note – this does not affect the current PRIMER session unless you then “Refresh preferences”*
2. Expand **[+] PRIMER** → **[+] checking** → **[+] model quality checks** → **[+] rigid** → **[+] Rigid body merge check** → **rigid body merge max separation**
3. Change the value to 250
4. Save settings & exit from the top **File** option. *The value of 250 will now be used in future sessions of PRIMER.*

User-defined Checking Options

- PRIMER has a further capability for customising error checking.
- A user-written file can change the status of selected errors, and add extra messages which will appear in the error log file. The user's scripts can then scan the error file for these messages and take action accordingly.

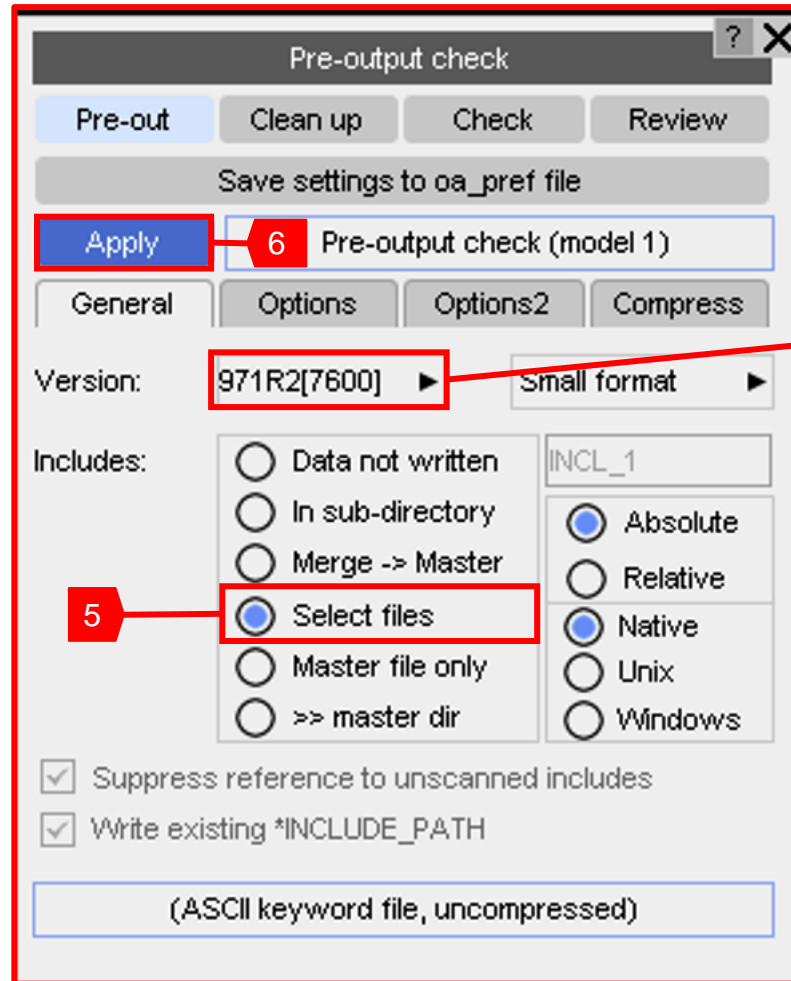
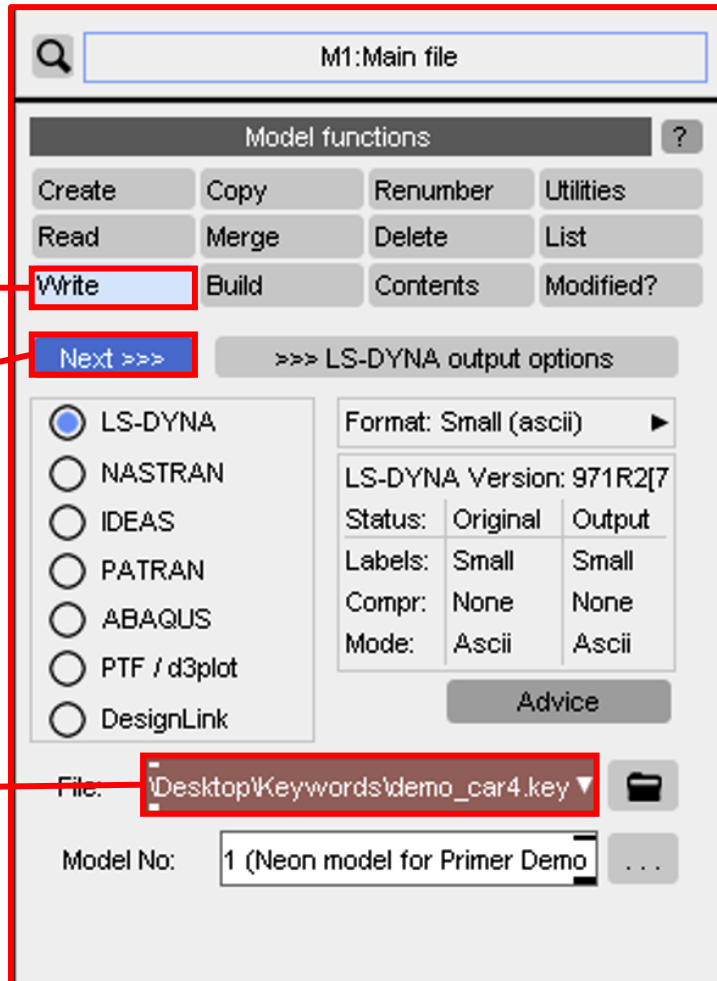
Example file:

```
PART_122,      ERROR,      Fatal - do not run this model  
M_ST_24_07,    WARNING,    Demoted to warning  
M_ST_24_10,    IGNORE,
```



Error tags – identifying the selected error. The tags can be made visible in the error tree.

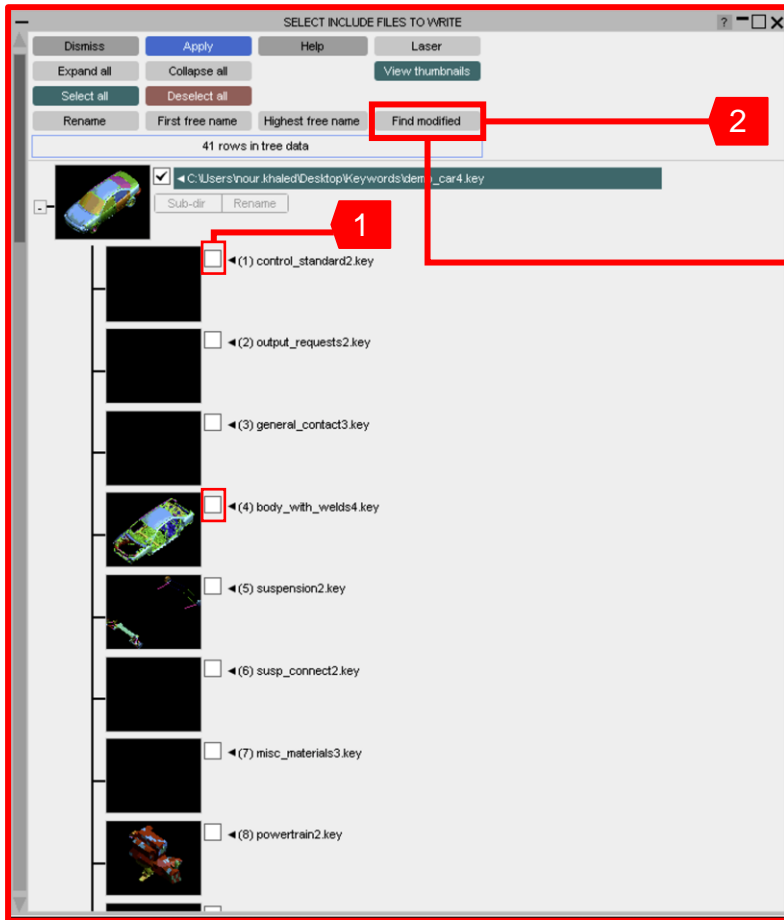
Writing Out the Corrected Model



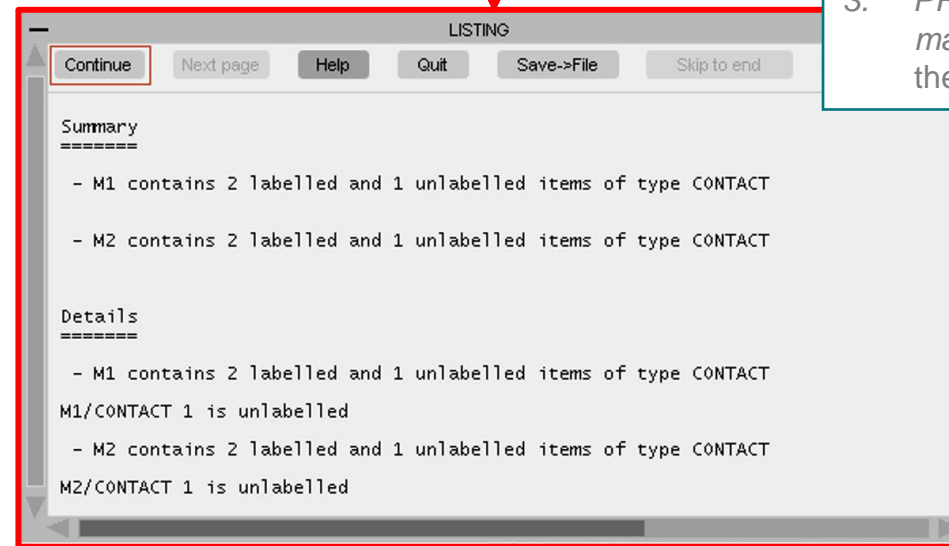
Model Write:

1. To write out the model, use **Write** from the main model menu.
2. Give a file name. *As the model has INCLUDE files, this will be the master keyword file.*
3. Press **Next >>>** to move to the next menu
4. Set the desired **Version** of LS-Dyna. *Input data not valid for this version (e.g. new KEYWORDS) will be omitted and a warning message printed.*
5. Pick the **Select Files** option – to choose which INCLUDE files are written in the next step
6. After selecting all the required options, press **Apply**

Writing Out the Corrected Model



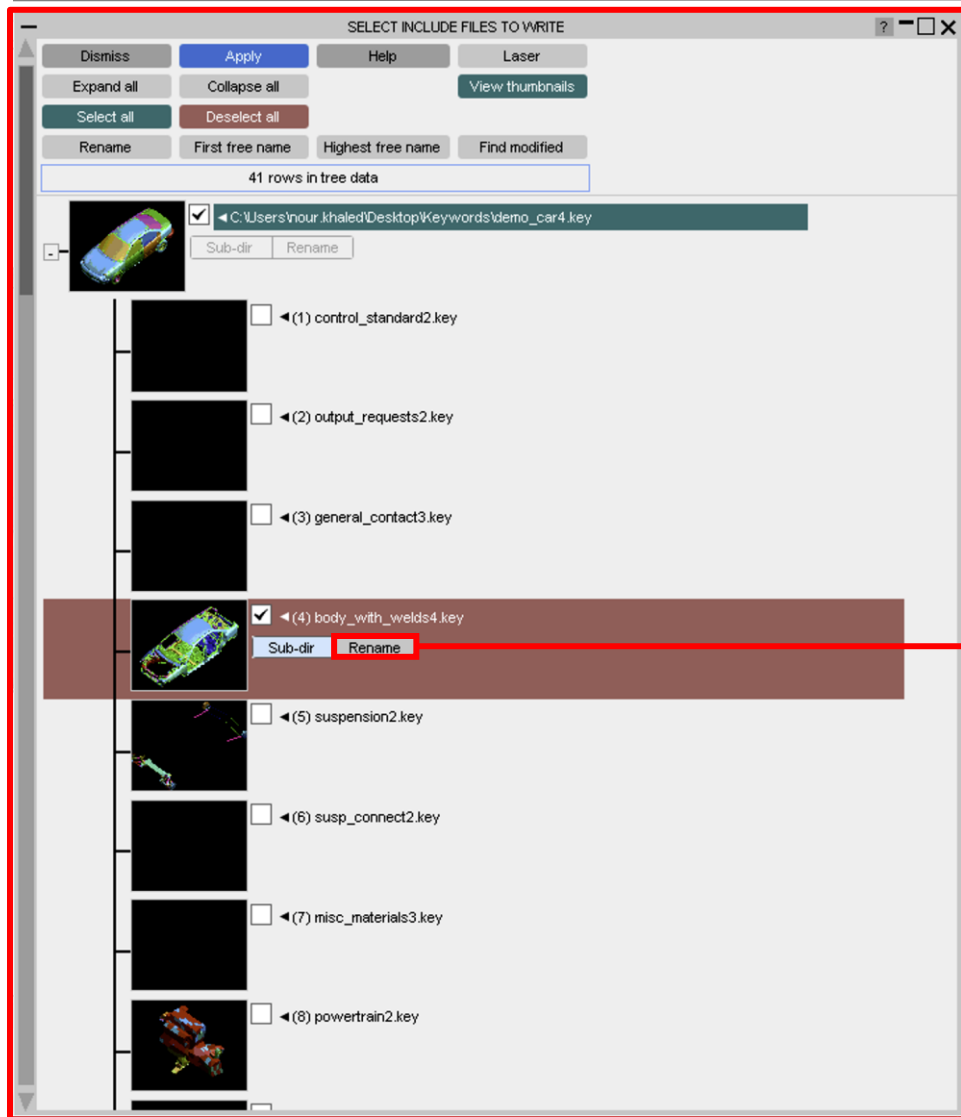
This list may be slightly different – depending on what modifications you've made



Model Write contd.:

1. Either click the boxes to tell PRIMER which INCLUDE files to write...
2. ... or ask PRIMER to **Find modified** INCLUDE files. *This process will take one or two minutes as PRIMER compares with the files previously read off disk.*
3. PRIMER will show a list of all the changes you've made in this session. **Skip to the end** of the listing then **Continue** to the next menu

Writing Out the Corrected Model



Model Write contd.:

1. The modified INCLUDE files are shown in red & automatically selected ☒.
2. To write the new files in the same directory as the originals, use **RENAME** and give a new file name.

Other Methods of Checking a Model

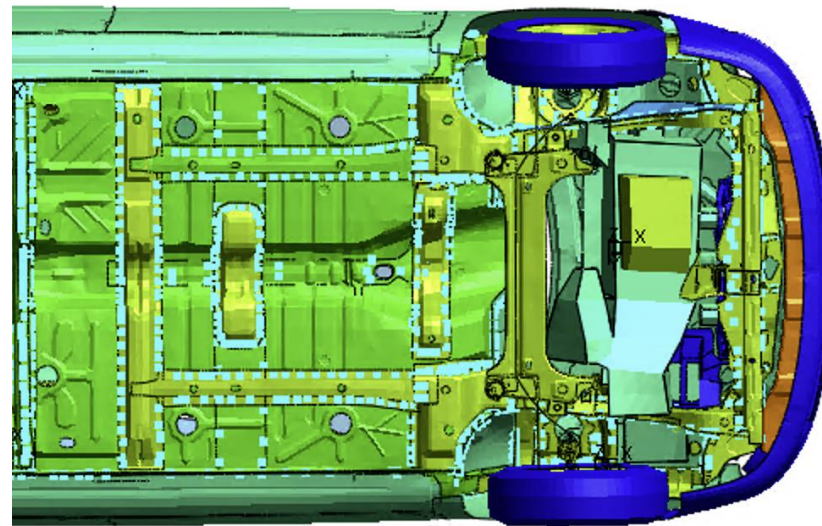
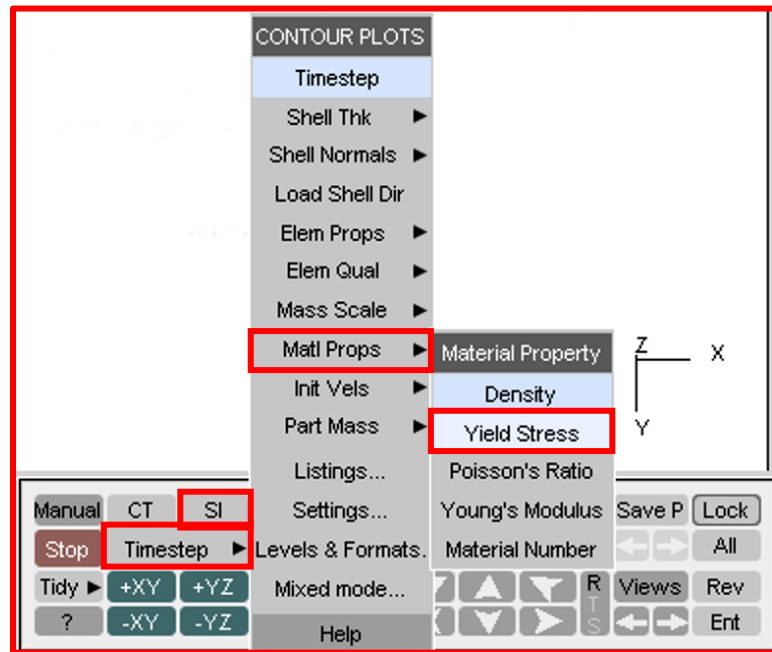
Visual Checking - Contouring

Ensure that only the body structure is visible.

Select component Matl Props, Yield Stress.

Click SI to perform a Shaded Image plot. The parts are now coloured according to yield stress.

Can you see the error? The part highlighted should be high-strength steel – green, like the part on the other side.



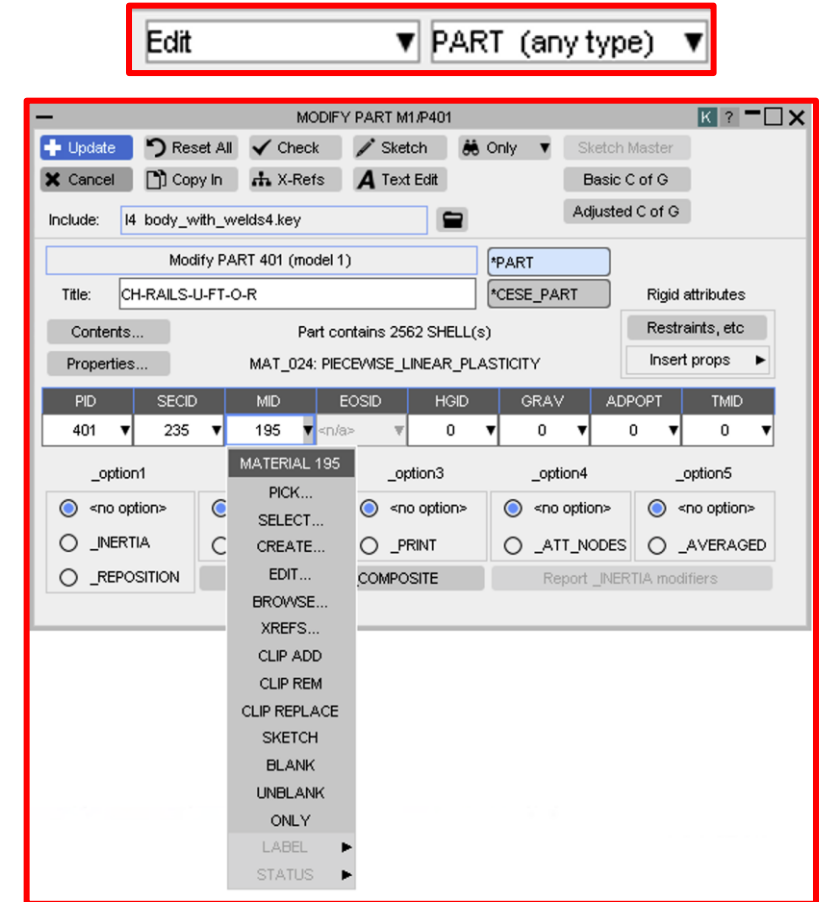
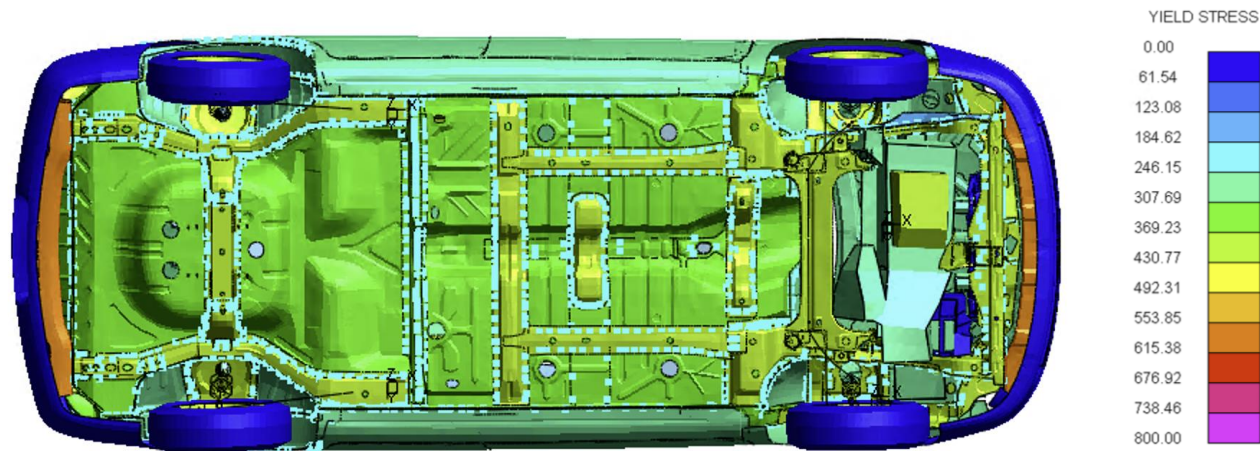
YIELD STRESS

0.00
61.54
123.08
184.62
246.15
307.69
369.23
430.77
492.31
553.85
615.38
676.92
738.46
800.00



Visual Checking - Contouring

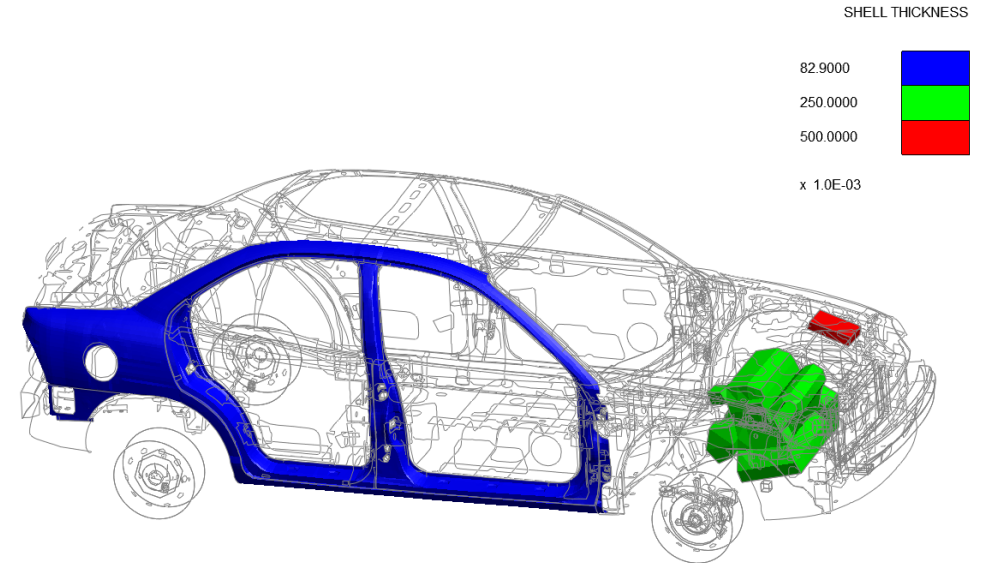
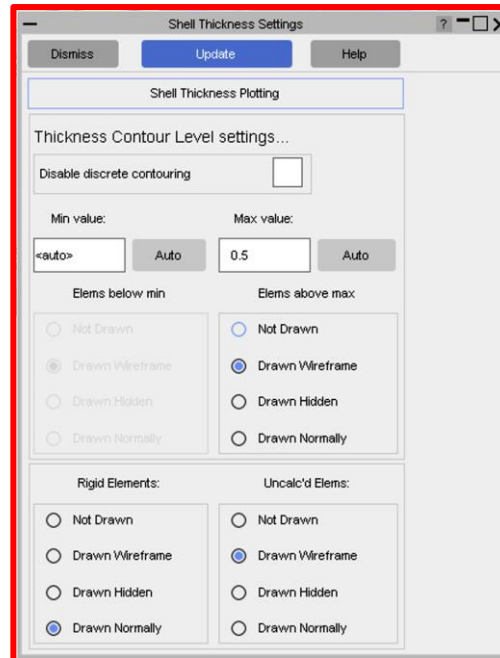
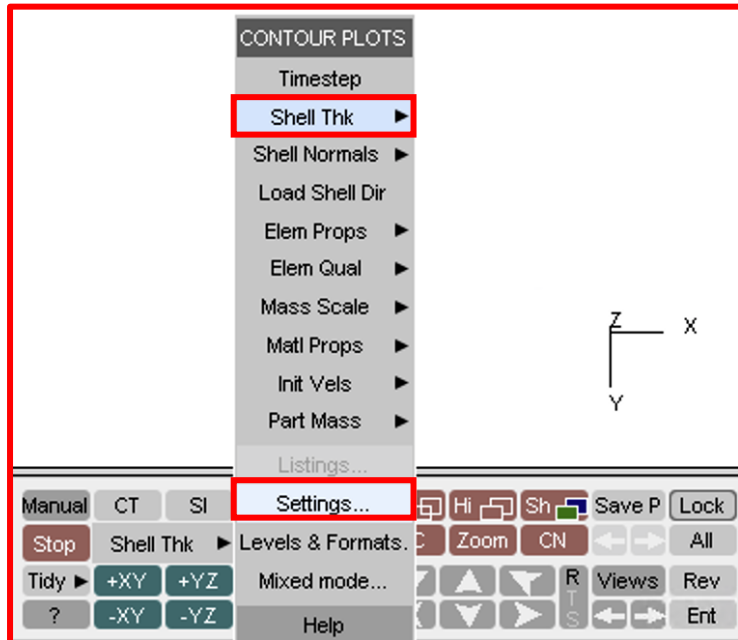
- To fix the error, we need to edit the Part data, and reference the correct material.
- Change the quick-pick settings to Edit Part. Click the offending Part to bring up the Part editing panel.
- Right-click on the Material ID, click PICK.
- Now click on the symmetrically-opposite part. The material ID should change to 234.
- Press UPDATE.
- Press SI. The plot should now show the correct symmetrical pattern of high-yield material.



Visual Checking - Contouring

Thickness contouring:

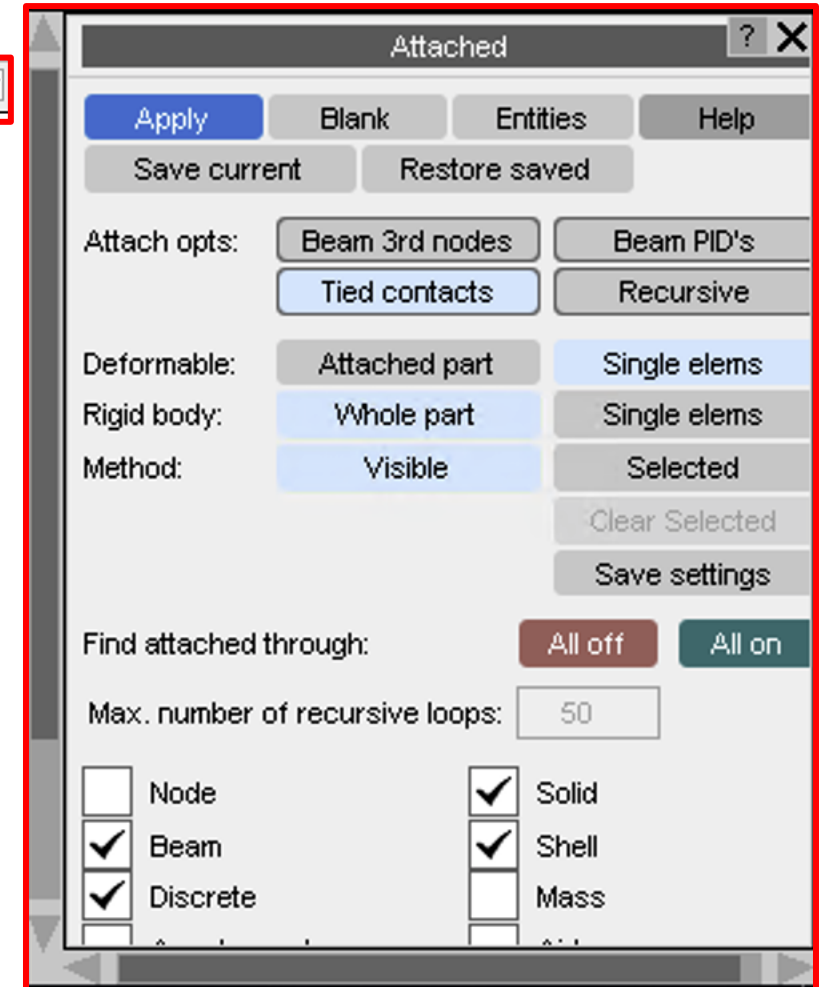
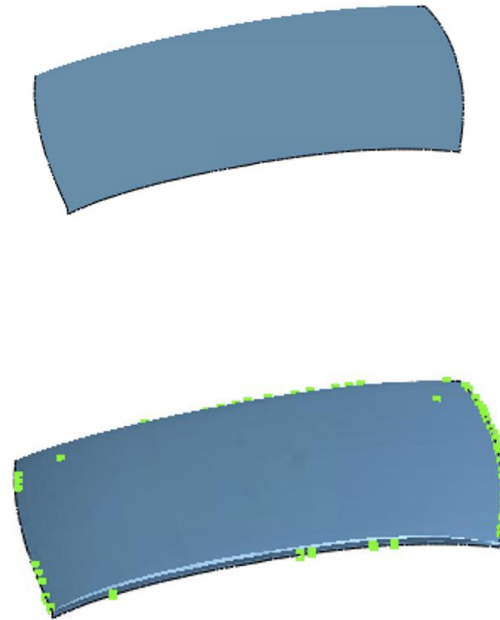
1. Ensure that only the body structure is visible.
2. Select component Shell Thk, do an SI plot
3. Note the contour levels – somewhere there is a panel with a thickness of only 0.083mm – this is probably an error.
4. To find it, set the contour levels: press Settings..., enter a contour maximum value 0.5. Press SI.
5. The part containing the error is now obvious – right-click on the part, edit, edit the Section: change the thickness to 0.83.
6. Experiment with plotting some other data components.



Visual Checking – Find Attached

To check that all parts are connected, we use the ATTACHED function. First, display only one part (e.g. the roof). To do this, right-click on the roof, click Only.

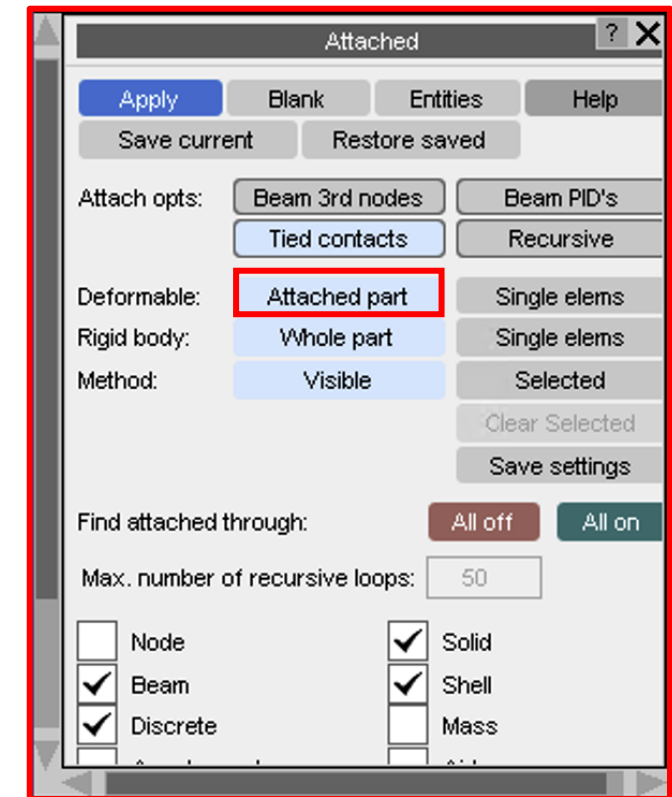
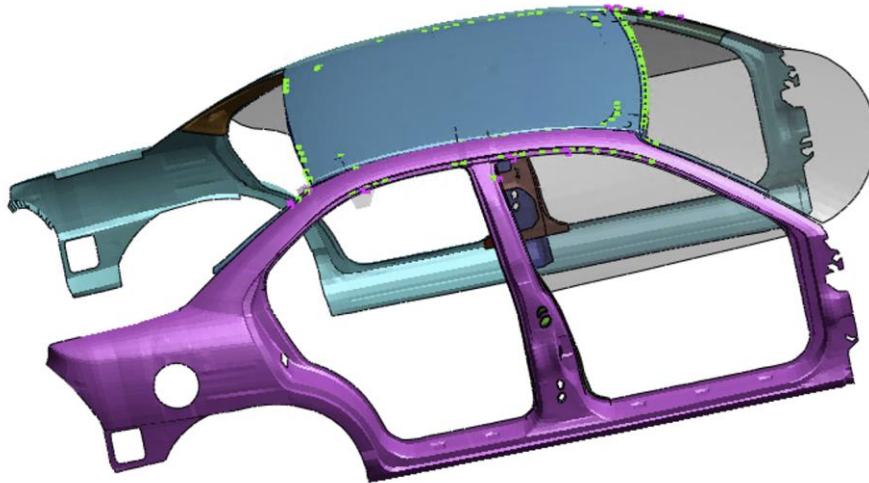
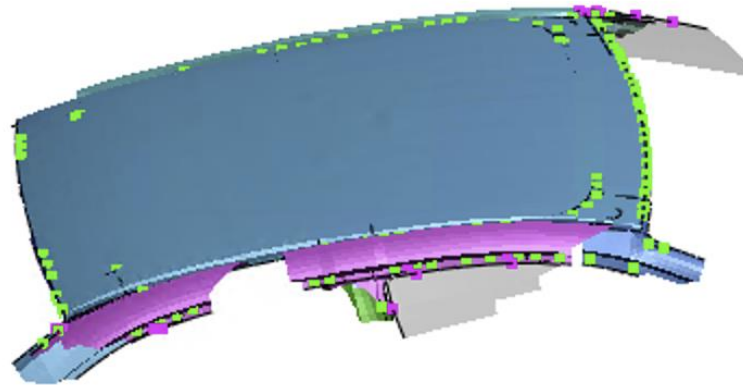
Under Tools, press **ATTACHED**. In the attached menu, press **APPLY**. PRIMER finds and displays any entities joined to the entities already displayed – in this case, mesh-independent spotwelds that are joined by a tied contact. PRIMER uses the same thickness tolerances etc as LS-DYNA to decide which nodes are tied by a tied contact.



Visual Checking – Find Attached

Press APPLY a few more times. Elements of the panels attached to the spotwelds appear.

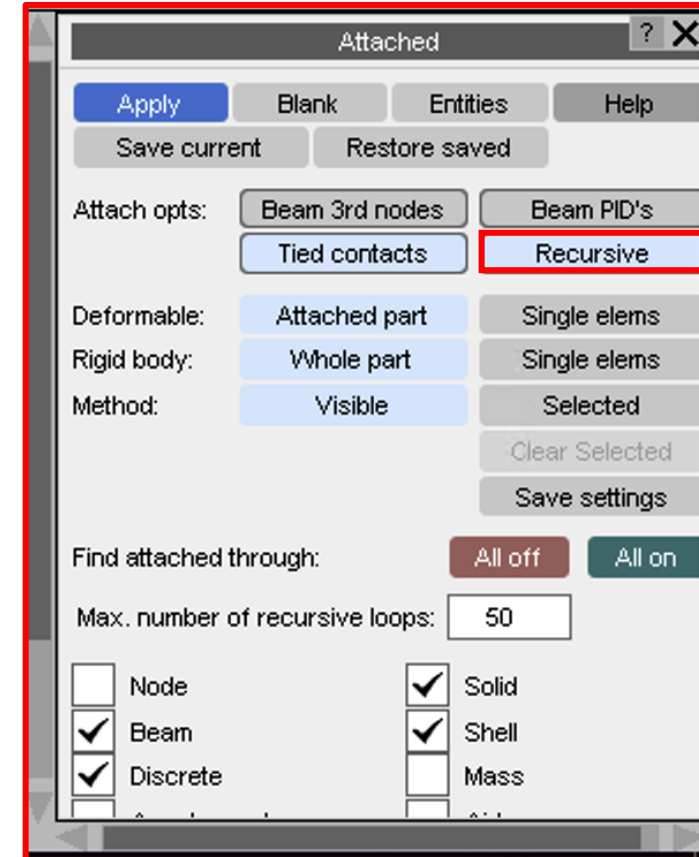
Now click Attached Part, and press APPLY. The whole of each attached part appears.



Visual Checking – Find Attached

To find every connected entity (without pressing APPLY many times), click Recursive. Press APPLY. PRIMER automatically continues to find attached entities until no more are found.

Finally, press R to reverse the blanking. Any loose part will now be displayed. These need to be fixed to the car.



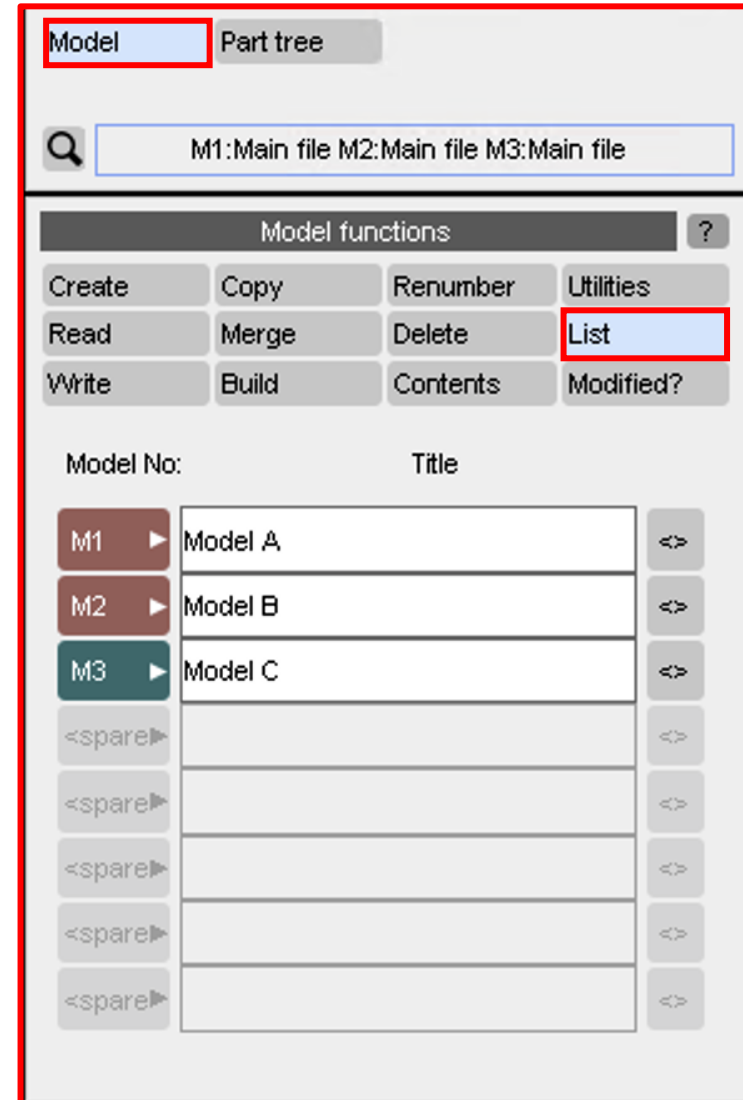
Working with Multiple Models and Merging Models

Working with Multiple Models

Read in odb_rigid_2.key and rigid_wall_conds_ffb.key. These now exist in PRIMER in addition to the first model that was read in; the three models are completely independent of each other, there can be no connection or inter-relation between them.

Use Model List to make each model visible (green) or invisible (red).

Try using the up/down arrow keys to make one model visible at a time. Leave only M3 visible.

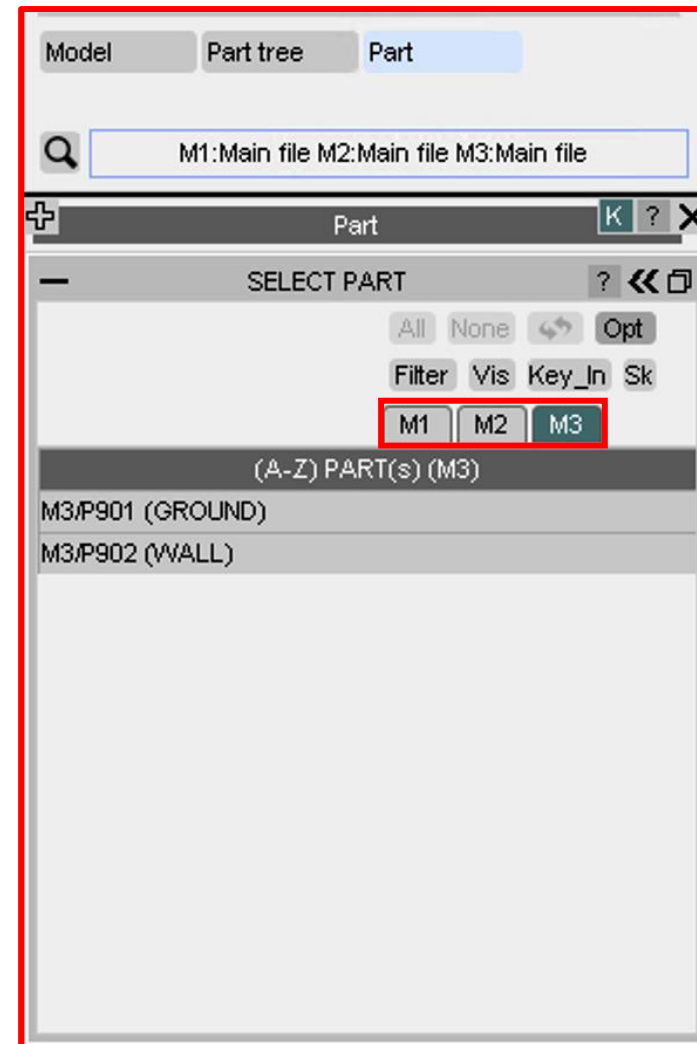


Working with Multiple Models

Go to Keyword=>Part=>Modify.

The object menu has tabs for each model. By default only the parts in active models are offered.

Try clicking the tabs, see the effect on the list of parts offered.

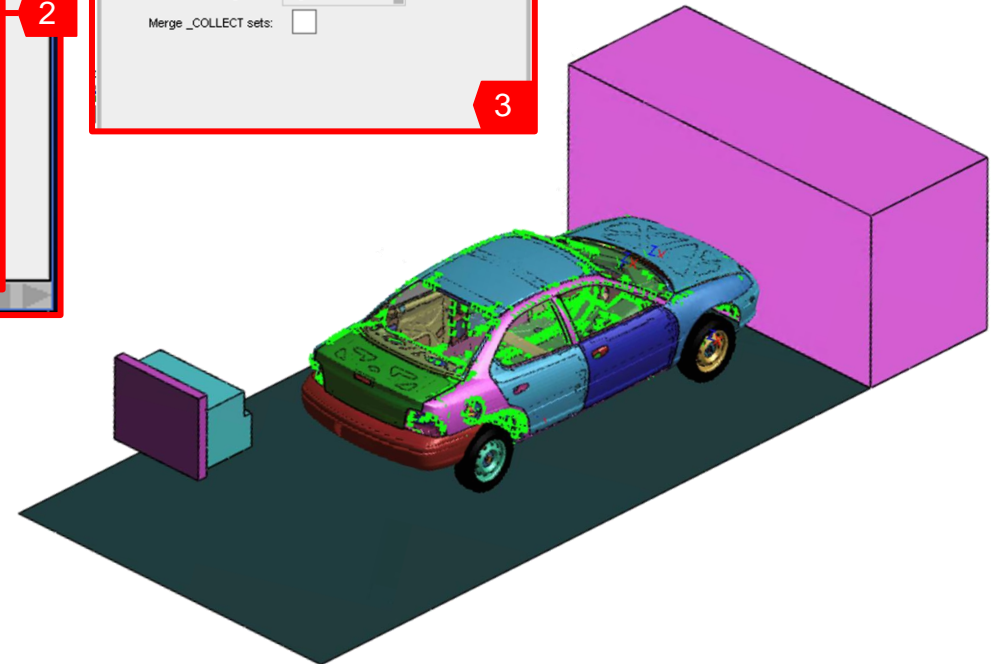
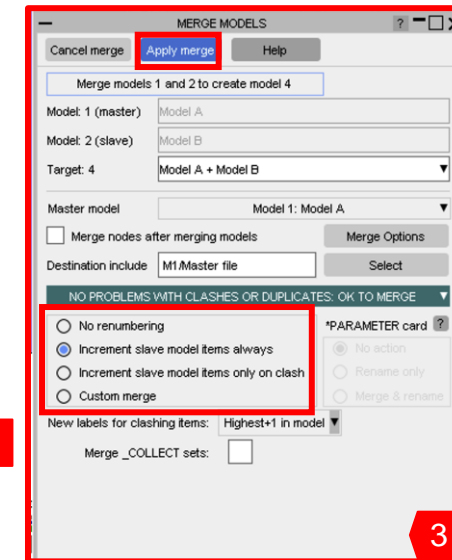
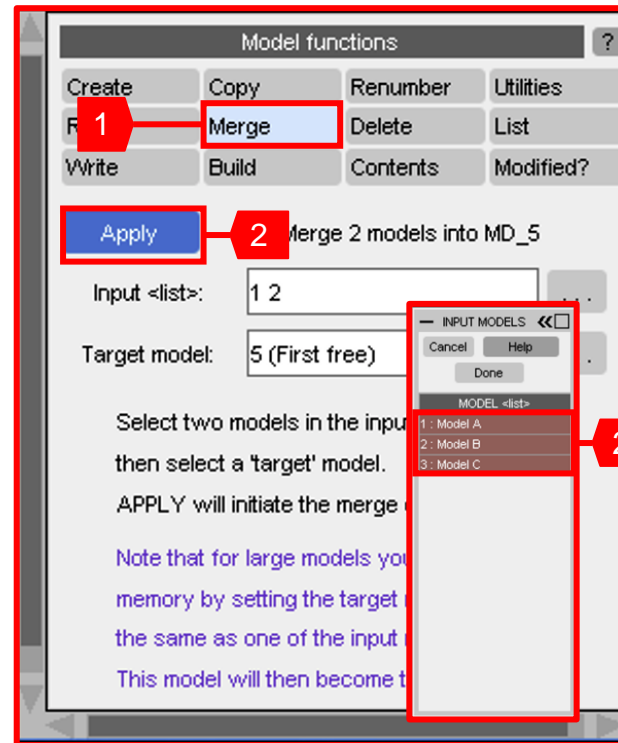


Merging Models

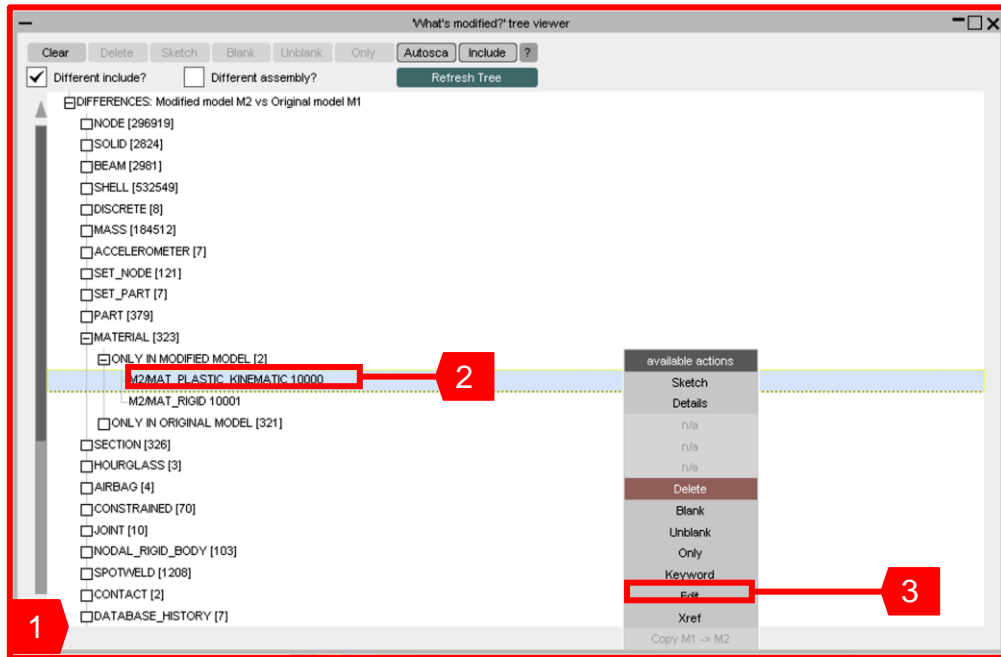
Merging Models

Multiple models can be read into PRIMER and kept separate unless the user merges them.

1. In the Model menu, click **Merge** to create a new model by merging existing models.
2. Select models 1 and 2 (the vehicle and ODB), click **Apply**.
3. Usually there will be clashing labels in the two models that need to be resolved – click “Increment slave model always”, then **Apply Merge**. A new model M4 will be created.
4. Check the contents of M4 using Model List to make it visible, and look for its parts in the Part Tree.
5. Delete models 2, 3 and 4 ready for the next workshop example.

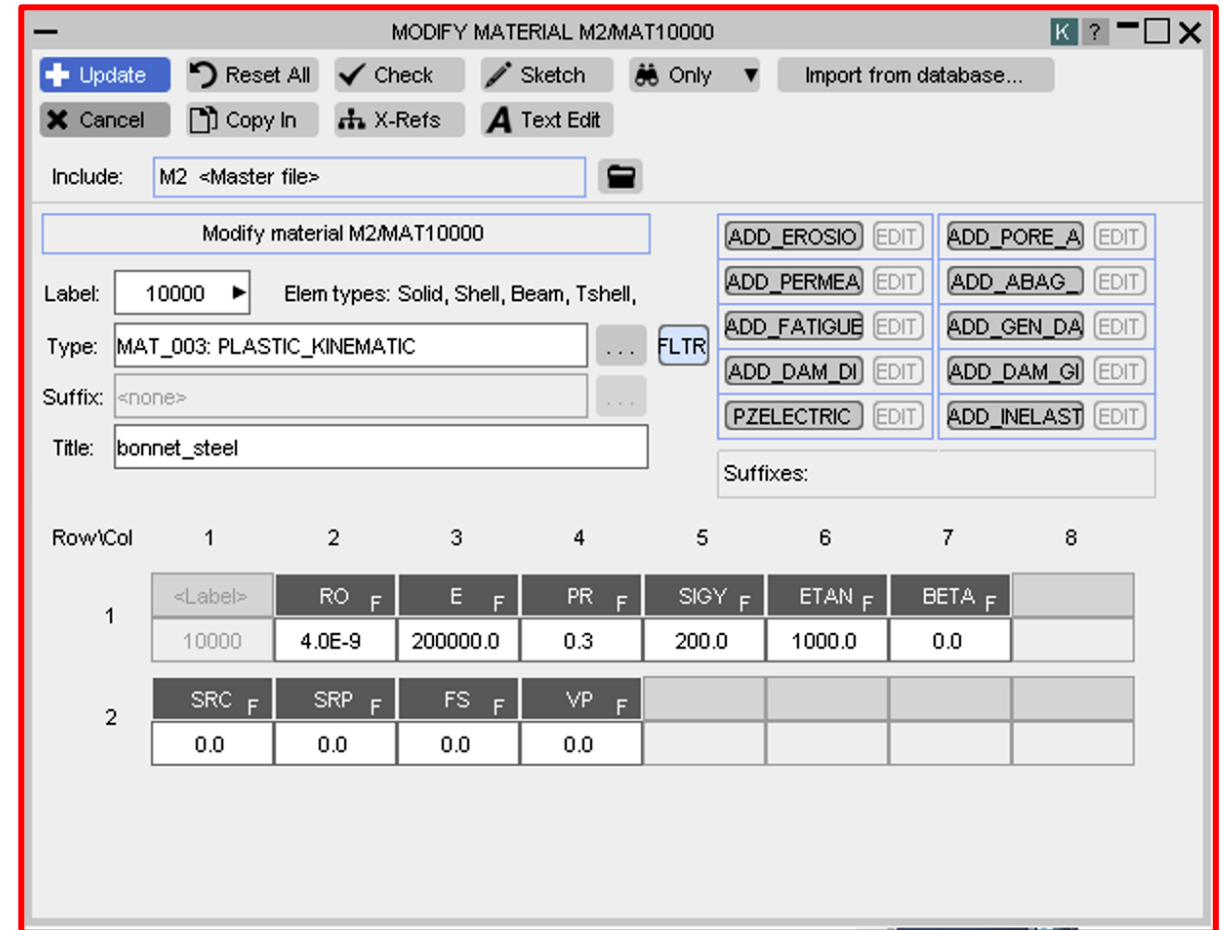


Exercise: Diff Function (Find Modified Keywords)



Modified?

1. The **What's modified tree viewer** pops up.
2. Expand the menus by clicking on the white boxes.
3. Right click on the difference found and click on **Edit**
4. This pops up the material where the difference was found. You can then interrogate what the difference is



Replace Part

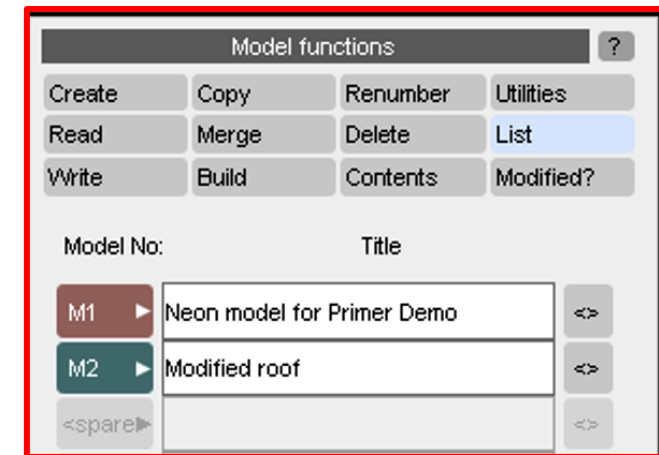
Replace Part



REPLACE PART is used to introduce a remeshed part into an LS-DYNA model.

In this example, the design of the roof has been changed. It is an extreme example, so that the change will be clearly visible in the pictures.

Read the model roof_modified.key. There are now 2 models in PRIMER. Use the red/green buttons in the Model List menu to make only Model 1 visible.



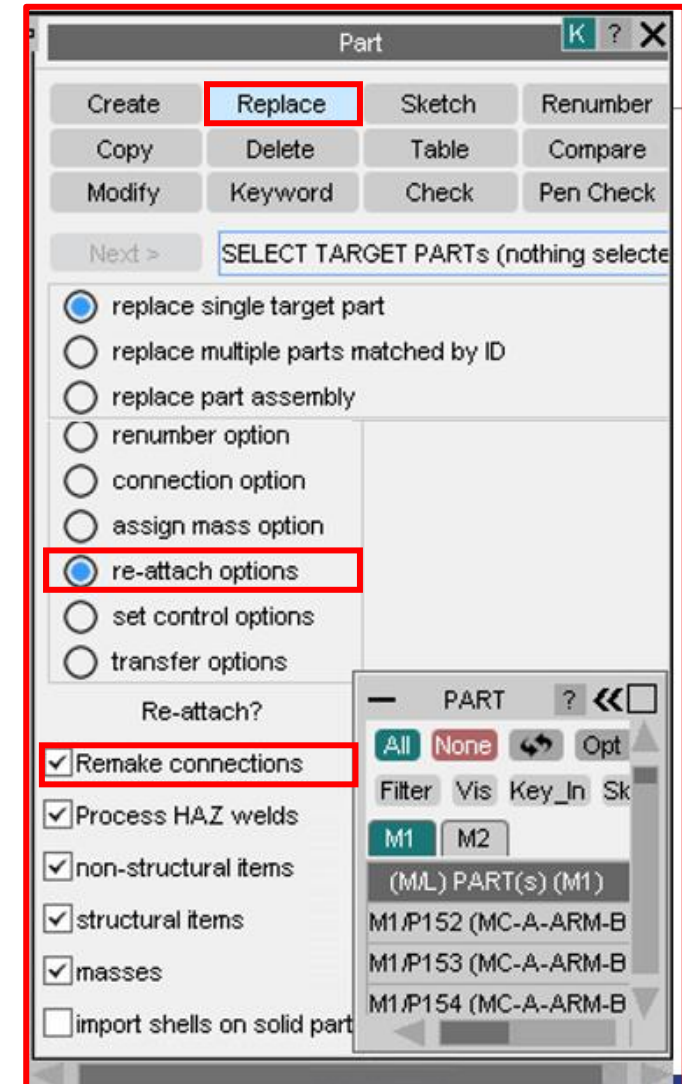
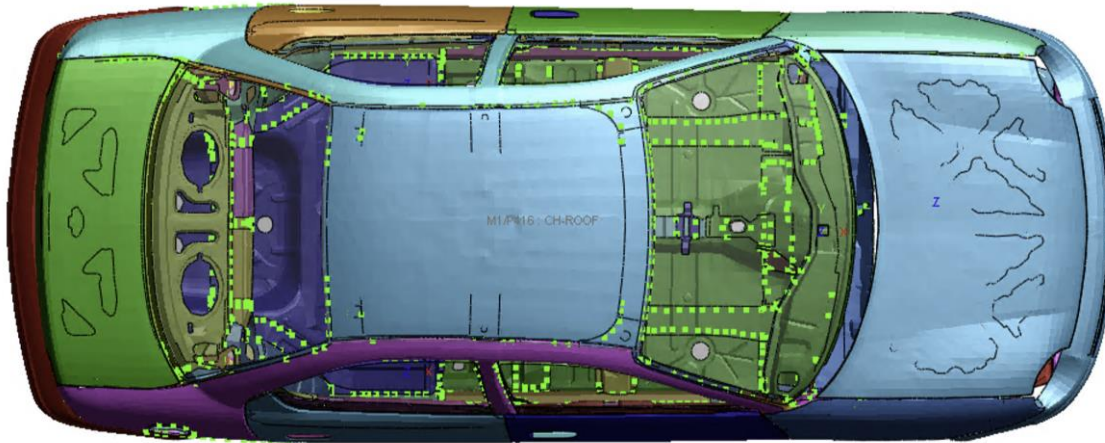
Replace Part

From the Keyword menu click Part, then Replace.

Leave the menu in “replace single part” mode.

Under “re-attach options” make sure “remake connections” is on, to ensure that mesh-independent spotwelds will re-connect to the new roof panel.

Click on the roof panel of Model 1, to select which part should be replaced. Click NEXT.



Replace Part

PRIMER now offers the parts in Model 2. Select P416 (the new roof). Press Next.

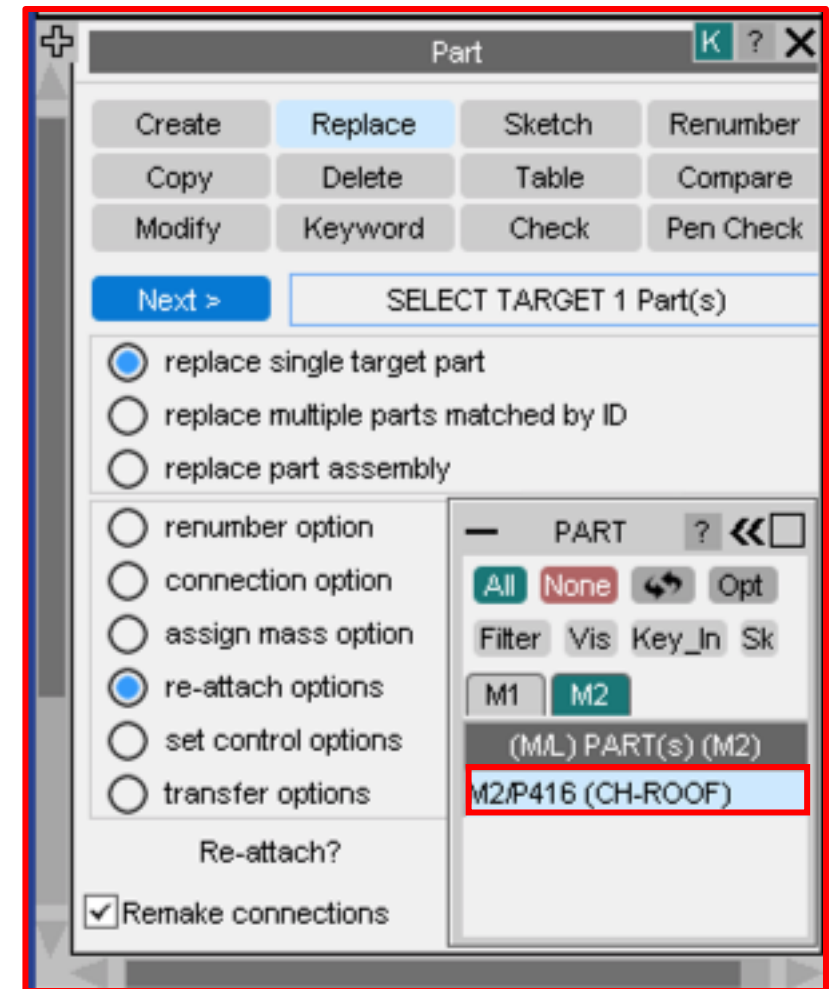
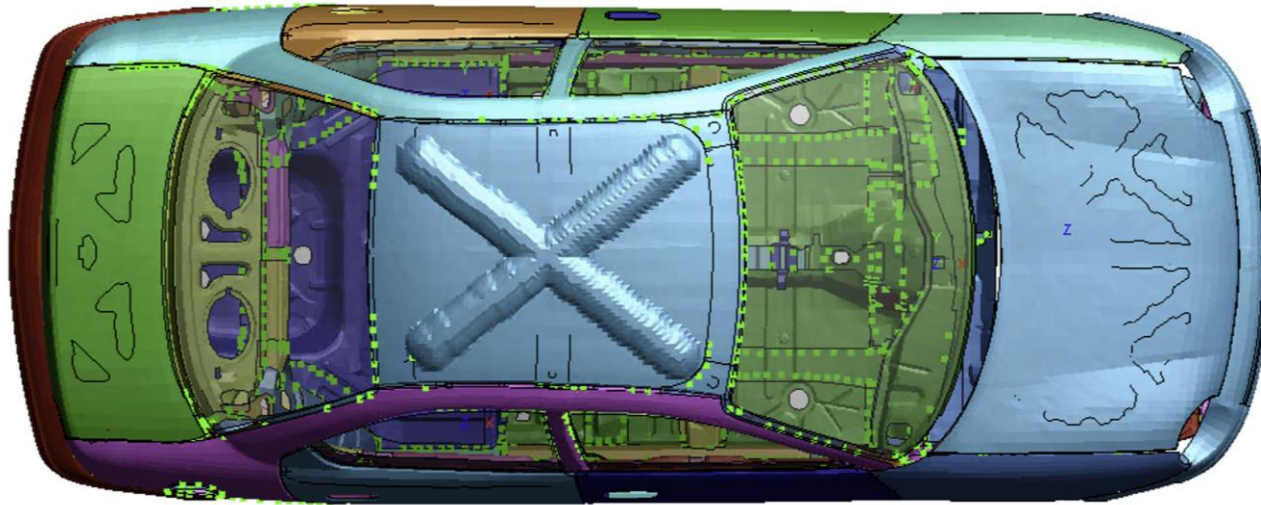
The new roof is now included in the car model.

PRIMER automatically fixes numbering clashes (e.g. same node ID in new mesh and in another panel).

Delete Model 2 (the separate roof panel) from PRIMER.

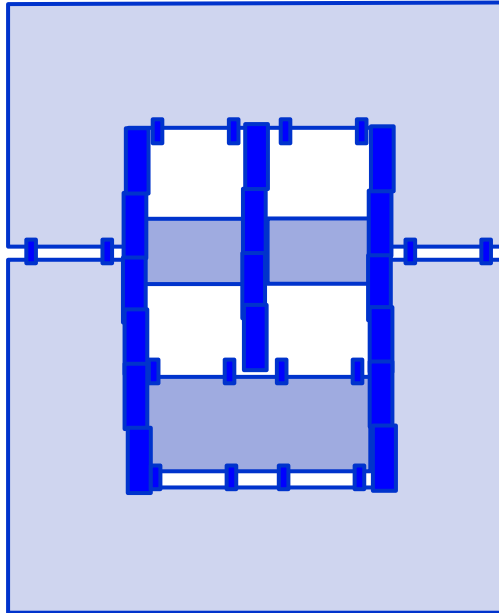
Part Replace will also re-attach any items connected to the old mesh (in this case, there aren't any such items).

Multiple parts may be replaced in one operation, provided that the Part IDs match.

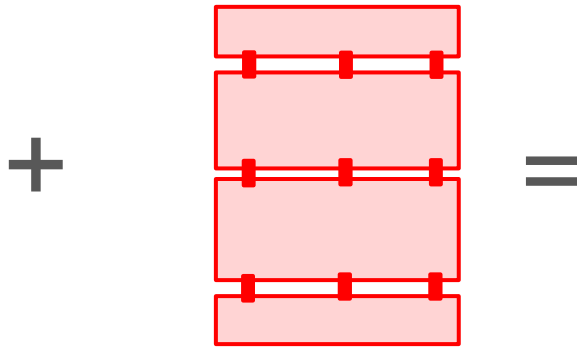


Replace Assembly

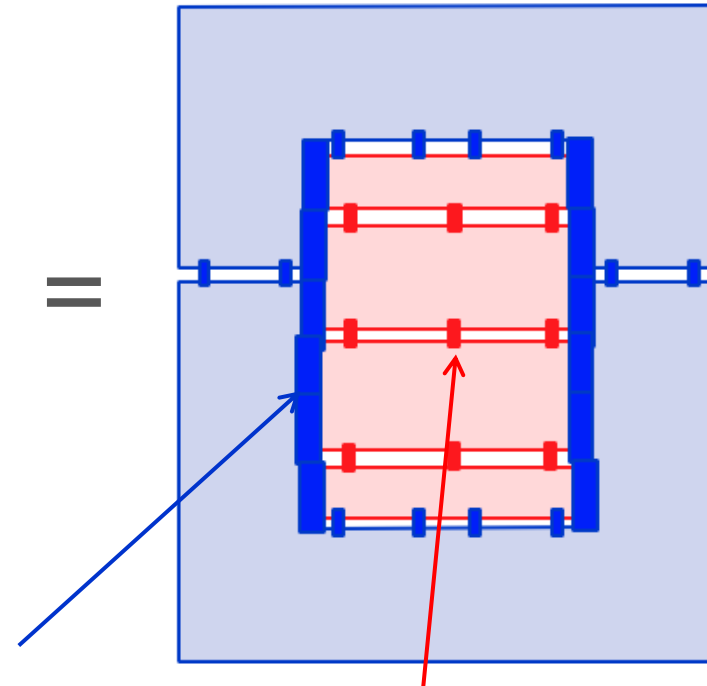
Old model with spotwelds



New mesh of assembly,
spotwelded, may have
different number of parts



Old model with spotwelds



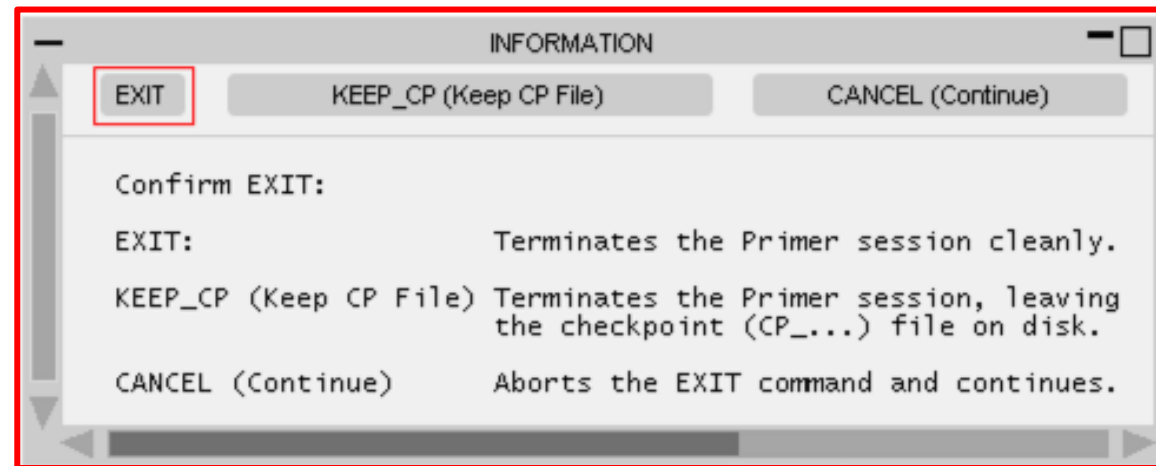
Spotwelds “around the edges” of the assembly are taken from the old model, and re-connected.

Spotwelds “inside” the assembly are taken from the new mesh

Checkpoint files, command files, macros and oa_pref files

Checkpoint Files

- A checkpoint file is automatically written by PRIMER during each session.
- Example filename: cp_PRIMER_rc1_(build604)_2314
- This is a record of what buttons you pressed, and where on the screen you clicked.
- A checkpoint file can be replayed, to recreate the exact same actions using the same model.
- Useful in the rare event of PRIMER crashing, or being terminated abnormally by the computer system – can recover any unsaved work.
- Also useful if your last action was a mistake – you can choose how many of the stored commands should be replayed.
- Checkpoint files are not editable and cannot be modified in any way.



Command Files

- A limited range of PRIMER functions are available in command files.
- PRIMER may be run in batch mode with a command file.
- For example, your LS-DYNA submission script could include a command to read the model into PRIMER, run the error checker, write an error report. The submission script could then read the error report and decide whether to submit the job.
- The PRIMER manual (Appendix XII) lists the available commands
- New commands may be available in addition to those in the manual – try typing H in the dialog box to see the available options

Command to run PRIMER in batch mode with a command file (see Manual Section 1.4 and 1.5):

```
$<pathname>/primer91.exe -d=batch -cf=my_command_file -exit
```

Example command file:

```
/READ DK (path)/model_name.key 1
```

```
/CHECK checkfile filename.dat
```

```
MODEL 1 APPLY
```

```
/EXIT
```

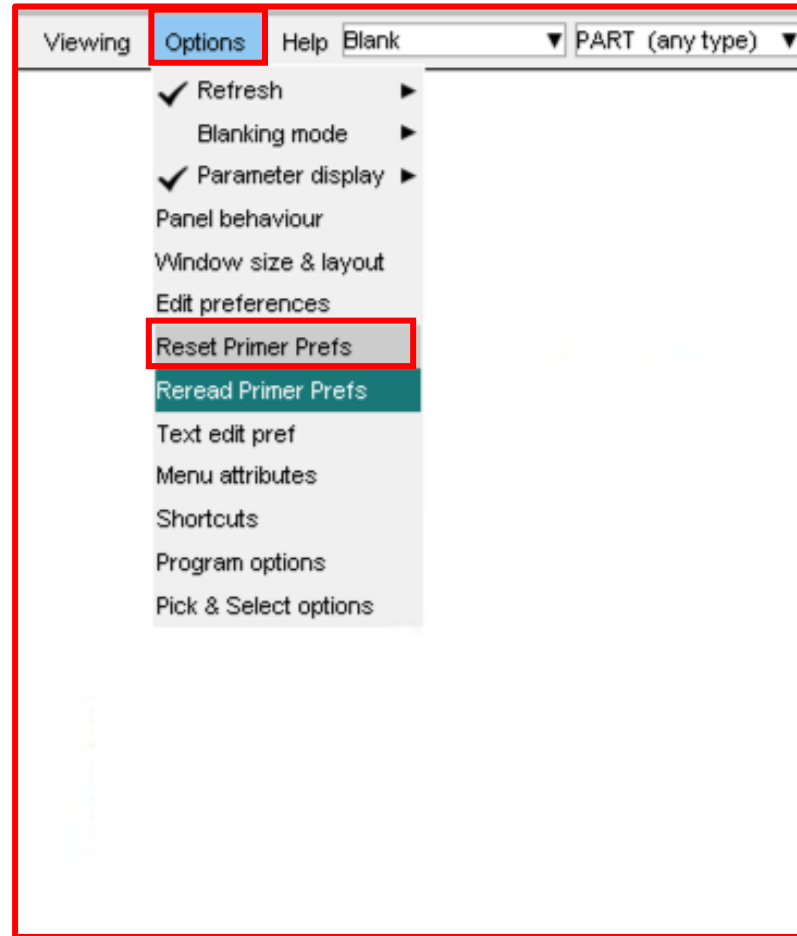

Preferences - (oa_pref files)

- oa_pref files are text files saved on your system that contain preferences used by Oasys software.
- These are useful so you don't have to change settings every time you open PRIMER/D3PLOT etc.
- The file will contain lines that look like this:

```
primer*display_factor:          1.40
primer*display_brightness:      1.00
primer*display_saturation:      1.00
primer*button_gradation:        0.00
primer*font_size:               default
primer*font_type:               helvetica
primer*background_colour:       WHITE
```

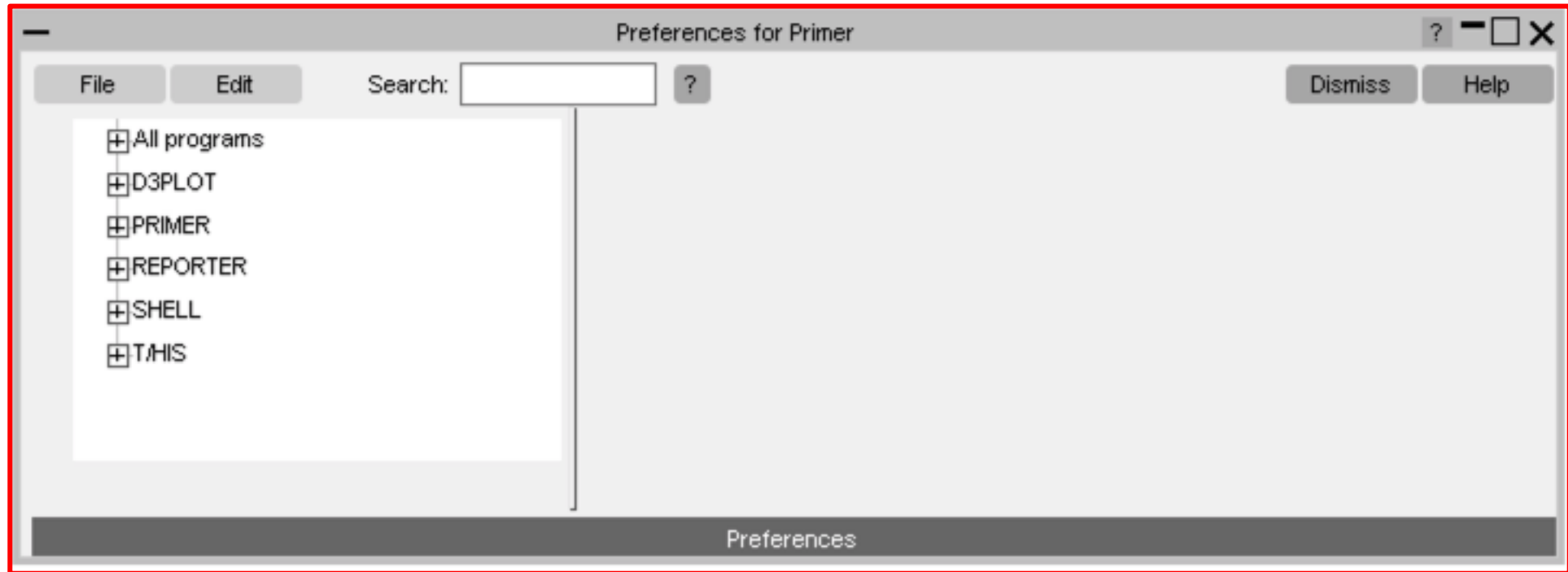
Preferences - (oa_pref files)

- Preferences can be set in PRIMER through the “Options” menu:



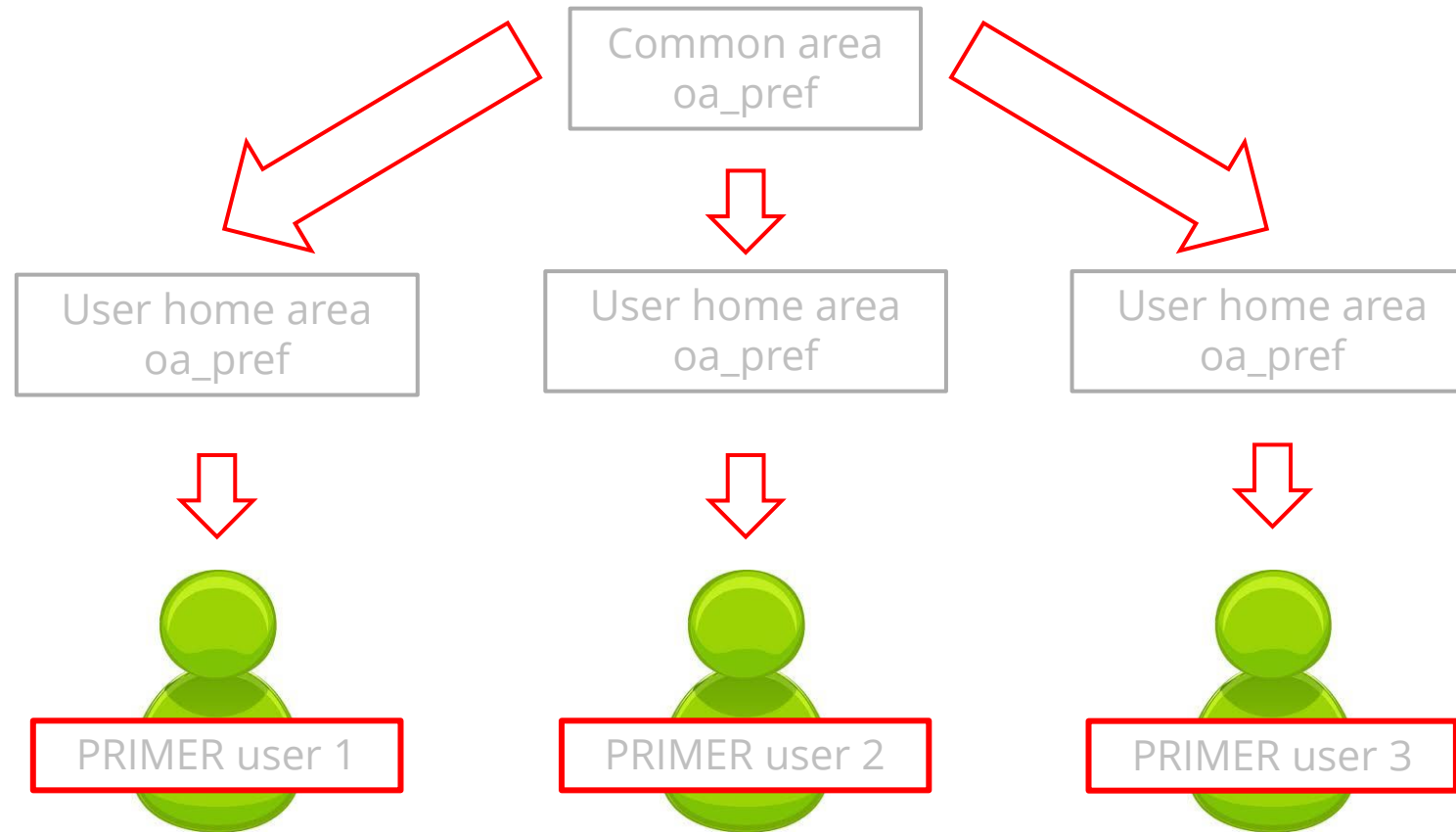
Preferences - (oa_pref files)

- When opening the preference panel, you may get a message about modifying the install preferences – this is because preferences can be saved in a number of places:



Preferences - (oa_pref files)

- When opening the preference panel, you may get a message about modifying the install preferences – this is because preferences can be saved in a number of places:

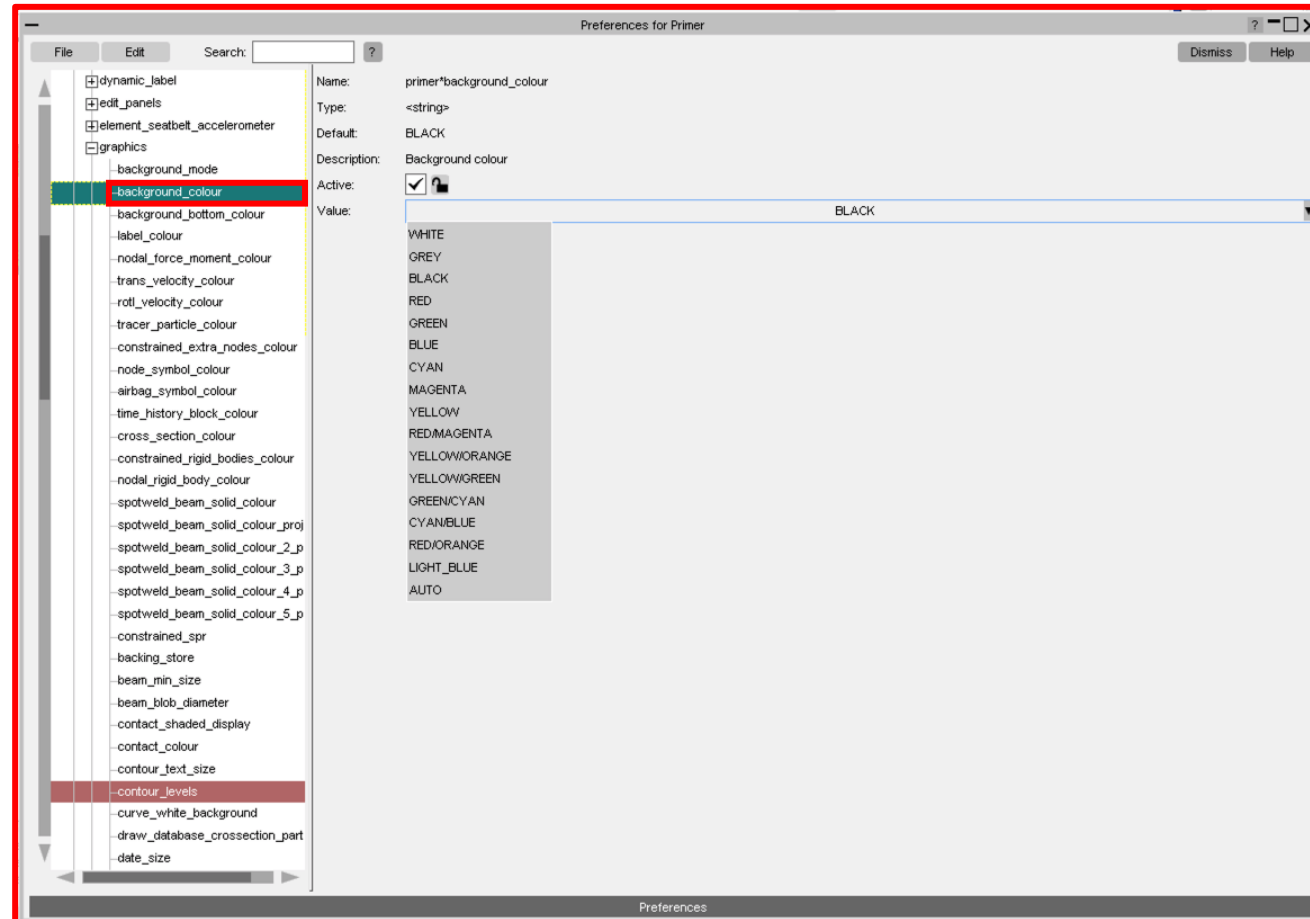


Preferences - (oa_pref files)

- oa_ref files can be stored in 4 locations:
 - OA_ADMIN – top level configuration – often the same as OA_INSTALL
 - OA_INSTALL – Installation level (location of executables)
 - HOME – users personal home area
 - Current working directory (rarely used)

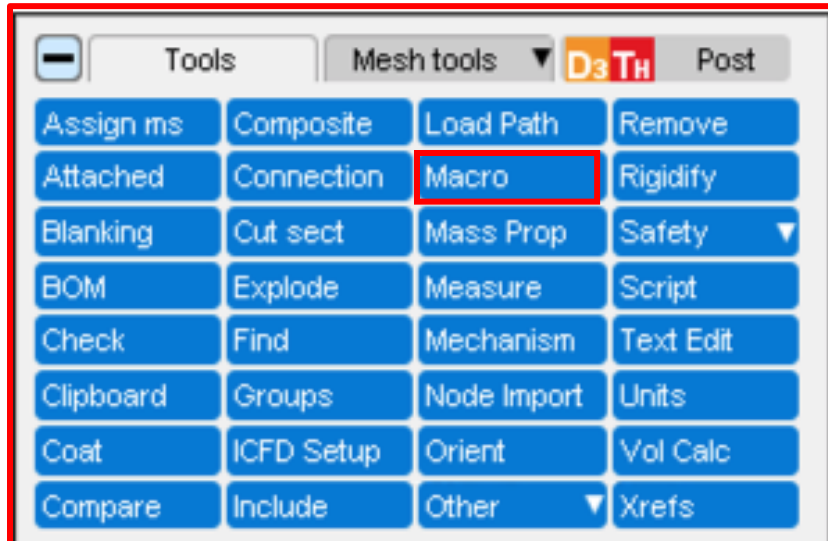
Preferences - (oa_pref files)

- Preferences can be changed in the preference panel which opens. The example below shows how to change the background colour of the graphics window. Once you have made changes, click File > Save



Macros

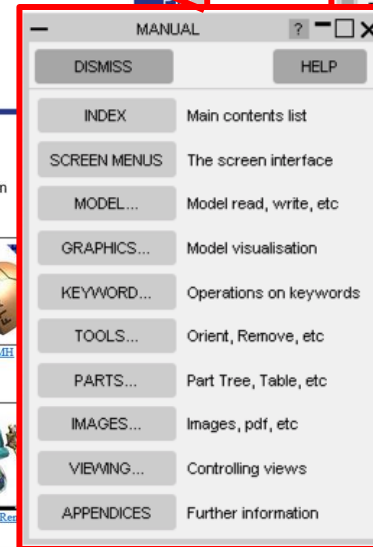
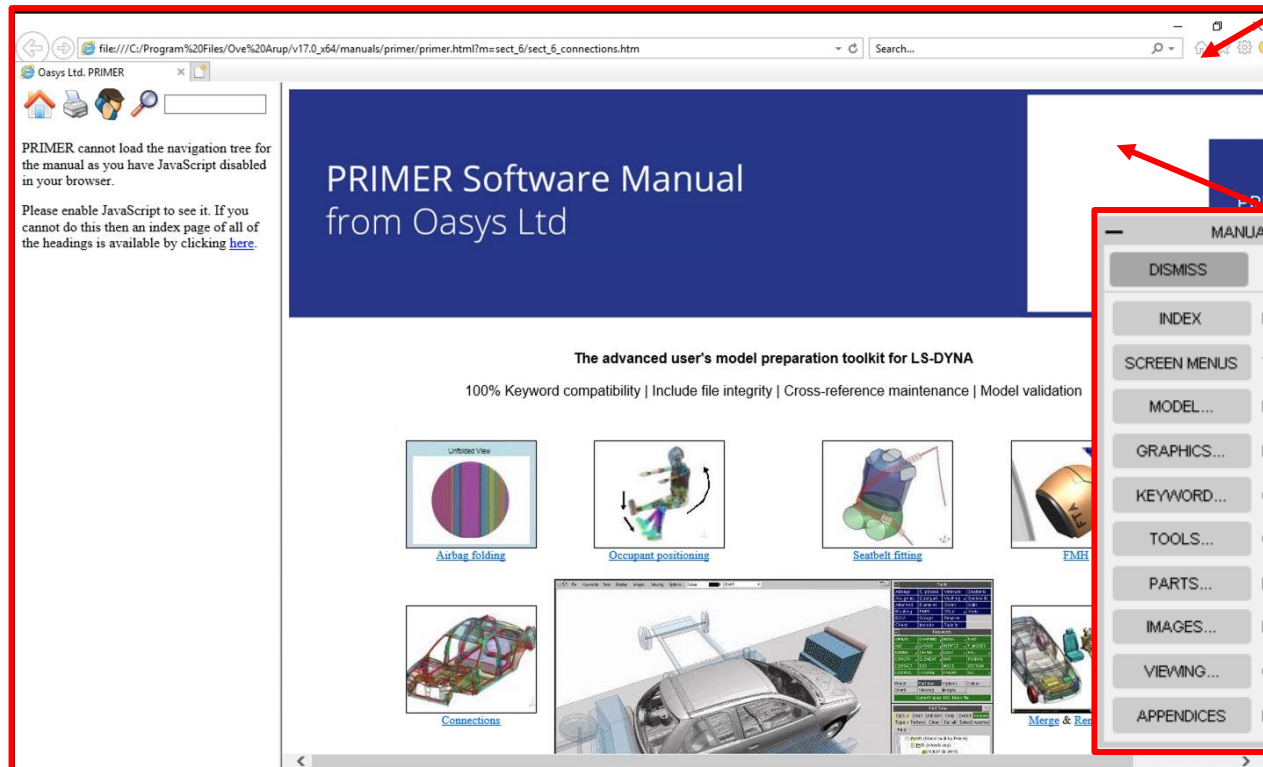
- Macros are readable, editable, recordable command files. They record which buttons were pressed in which menus.
- Macros offer the most flexible method of capturing a process, then applying the process to a different model.
- The “Pause” command (added by hand-editing the macro file) allows instructions to be passed to the end-user, who may then take control to carry out some actions interactively.



```
Window("Contact").Button("Create")
Window("CREATE CONTACT in model 1").Popup("Label:")
PopupWindow1().Button("Highest+1 Layer")
Window("CREATE CONTACT in model 1").Button("Contact Type")
Window("CREATE CONTACT in model 1").Menu("CONTACT TYPE").Select1("AUTOMATIC_SURFACE_TO_SURFACE")
In Window("CREATE CONTACT in model 1")
    .Textbox("Static friction (fs)") = "    0.2"
    .Textbox("Dynamic friction (fd)") = "    0.2"
    .Button("Optional data..")
    .Textbox("Soft constraint opt (soft)") = "    1"
    .Button("==> Main Panel")
    .Popup("Set type (sstyp)")
End In
```

When You Need Help

- Help text is available from most menus
- The on-line manual can be accessed from within PRIMER, provided that it has been installed with the software
- Support line – call us (or your distributor).

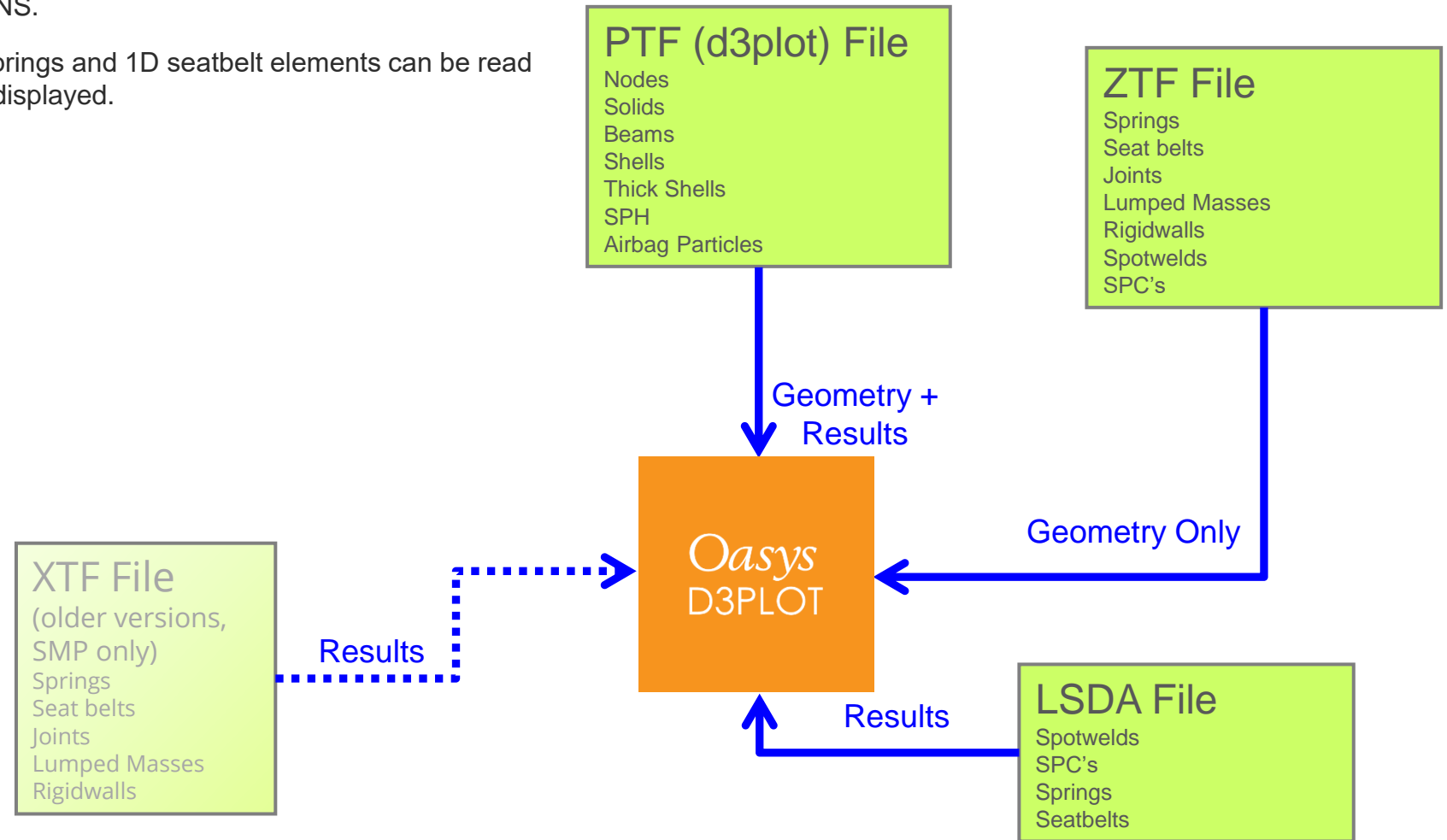


ztf File

- Certain information (e.g. beam sections) is not contained in the d3plot/ptf file.
- The ztf file is a file that passes such information to D3Plot.
- Several features in D3Plot benefit from the ztf file.
- It can be automatically generated when you open D3Plot (see “Opening Models”) or can be written using PRIMER.
- The ztf file must have the same filename as the ptf being opened.

Data Read into D3PLOT

- The ZTF file created by PRIMER includes data for SPC's, spotwelds, *DATABASE_CROSS_SECTIONS.
- Results for spotwelds, SPC's, springs and 1D seatbelt elements can be read from the LSDA (binout) file and displayed.

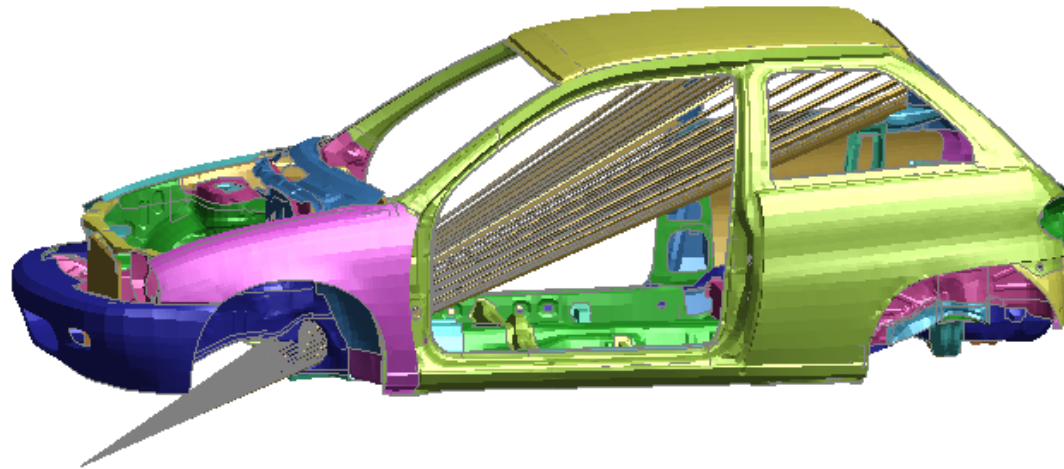


Working with INCLUDE files

Good Modelling –Practice 1

Self-contained INCLUDE files:

- When an INCLUDE file contains *PART, all the elements and nodes contained in that part should be in the same INCLUDE file.
- LS-DYNA data referencing entities in another INCLUDE file should be minimised (“connections”)
- To check, read each INCLUDE file as a separate model.
- This type of image is seen when elements are in one INCLUDE file (that has been read into PRIMER) reference nodes in another file that has not been read in. “Latent” nodes are drawn at (0,0,0).



Good Modelling –Practice 2

Define a numbering scheme for each INCLUDE file. This will minimise the chance of clashes when building models, and allow large teams to work in parallel on the different INCLUDE files.

This is a very simplified example!

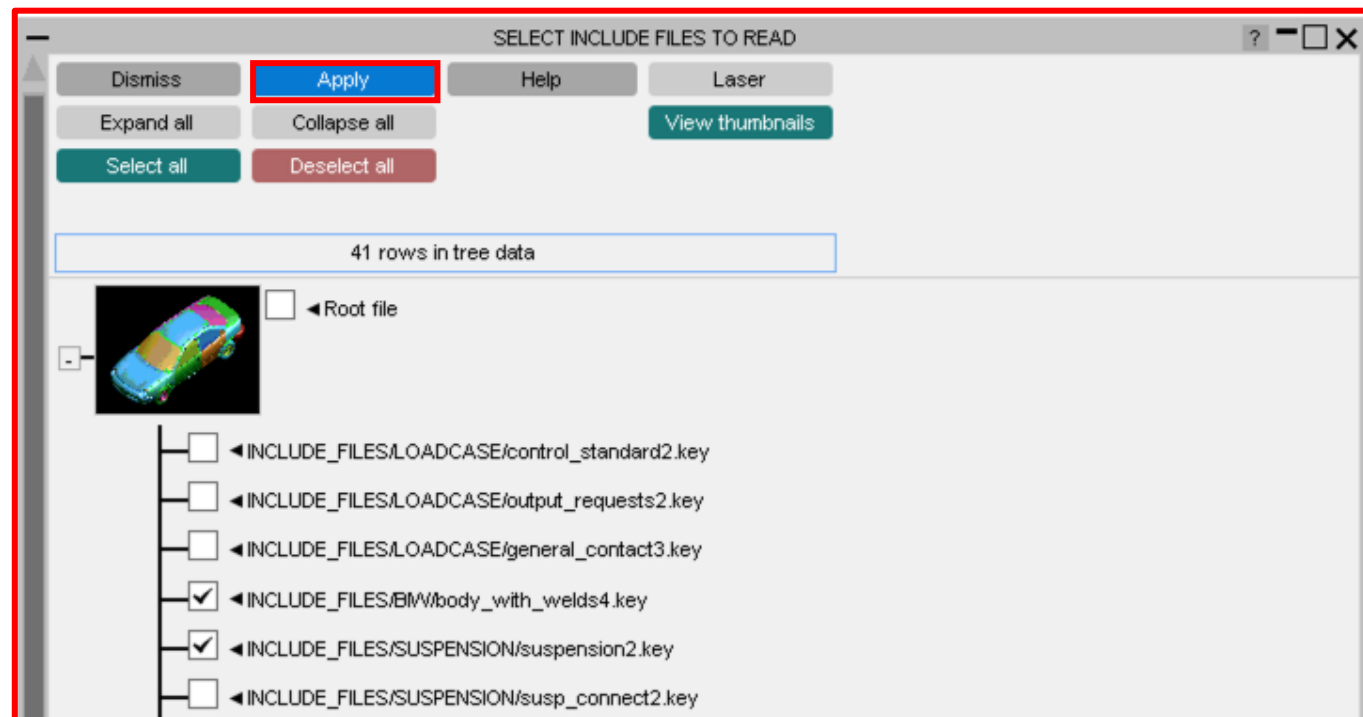
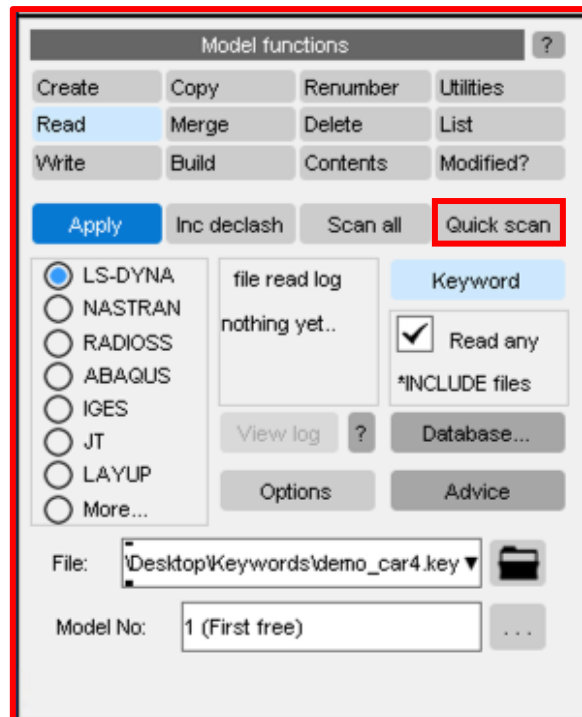
Component	Node, element, NRB, node set	All other data e.g. Parts
Barriers	1 to 499,999	1 to 4,999
Dummies (driver side)	500,000 to 999,999	5,000 to 9,999
Dummies (passenger side)	1,000,000 to 1,499,999	10,000 to 14,999
Airbags	1,500,000 to 1,599,999	15,000 to 15,999
Seats	4,500,000 to 4,999,999	45,000 to 49,999
Powertrain	5,000,000 to 5,999,999	50,000 to 59,999
Suspension	6,000,000 to 6,999,999	60,000 to 69,999
Closures	8,000,000 to 8,999,999	80,000 to 89,999
Body structure	10,000,000 to 19,000,000	100,000 to 189,999
Body structure spotwelds	19,000,000 to 19,999,999	190,000 to 199,999

Reading Selected INCLUDE Files

Scan and Quick-scan

Include files may be selected on reading a model.

1. In the Model menu, click **Read**. Browse for demo_car4.key. Press **Quick scan**.
2. Click on the body and suspension INCLUDE files. Press **Apply**.
3. PRIMER reads in only the selected INCLUDE files – this will be Model 2.

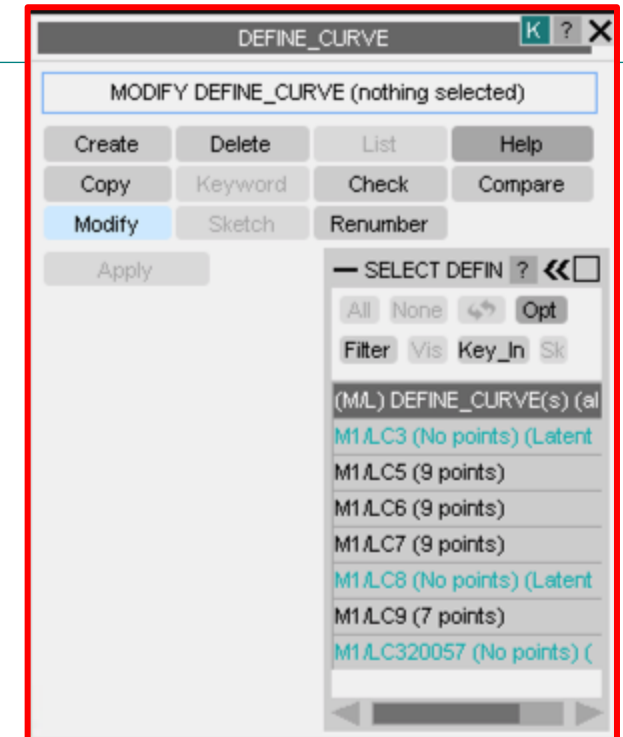
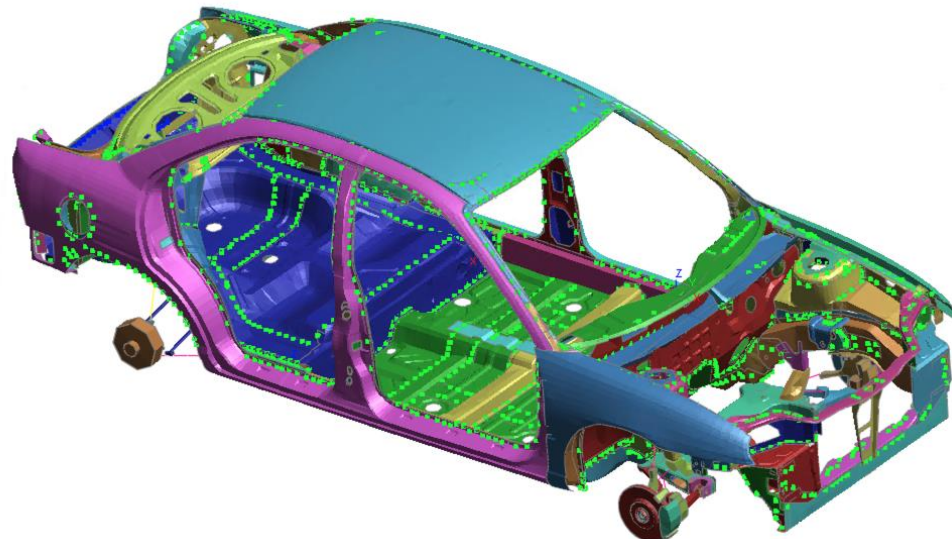


Reading Selected INCLUDE Files

Some data referenced in the INCLUDE files that we have read in, was defined in other INCLUDE files that we have not read in. The model is incomplete. Go to Keyword=>DEFINE_CURVE=>Modify. Note the curves listed in pale blue type – these are called “latent”, i.e referenced but not defined. These would also show up as errors during a Model Check.

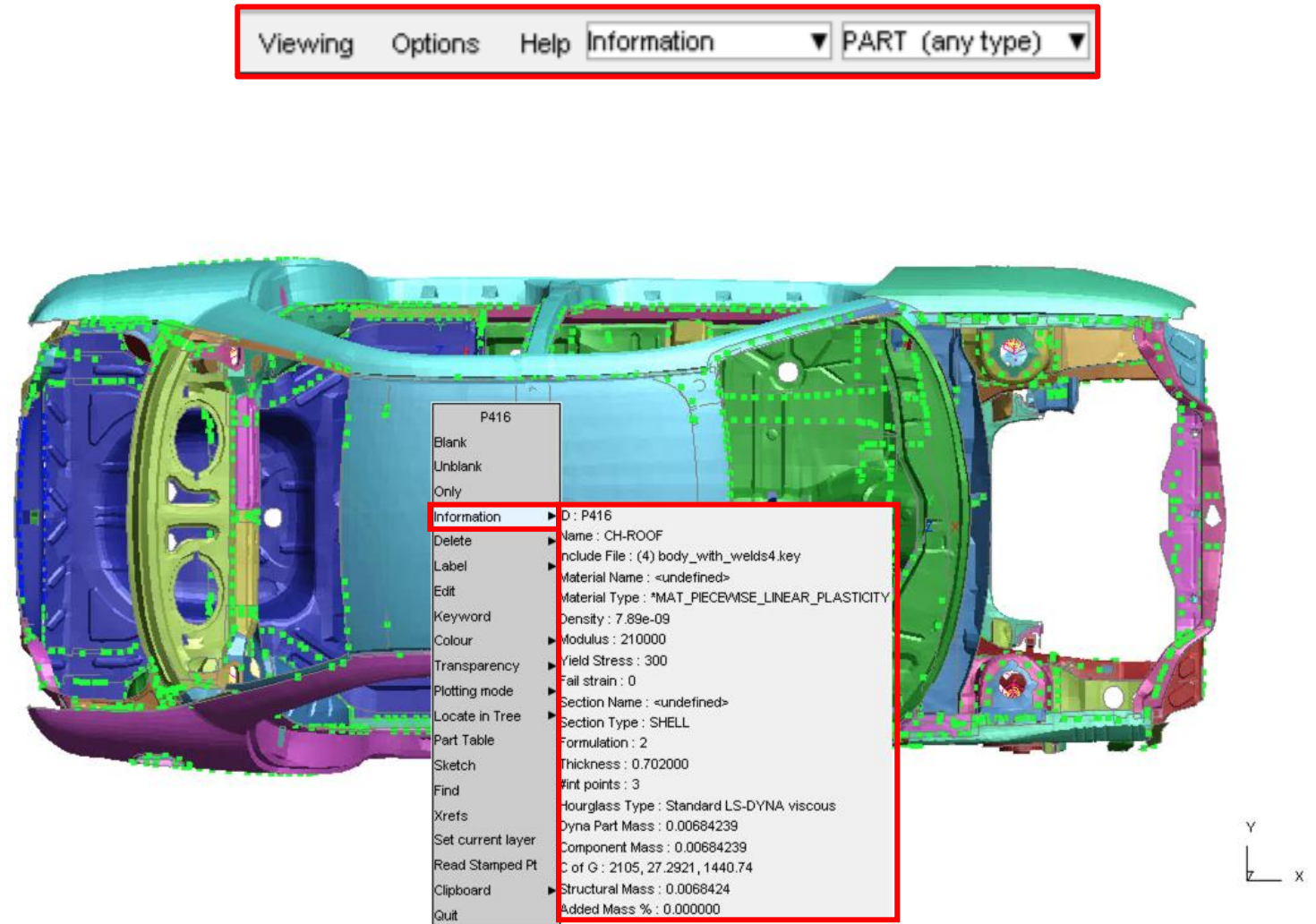
It is not good practice to create INCLUDE files like this – they should be as self-contained as possible, except for connections that join across INCLUDE files.

Now delete all models in PRIMER except Model M1.

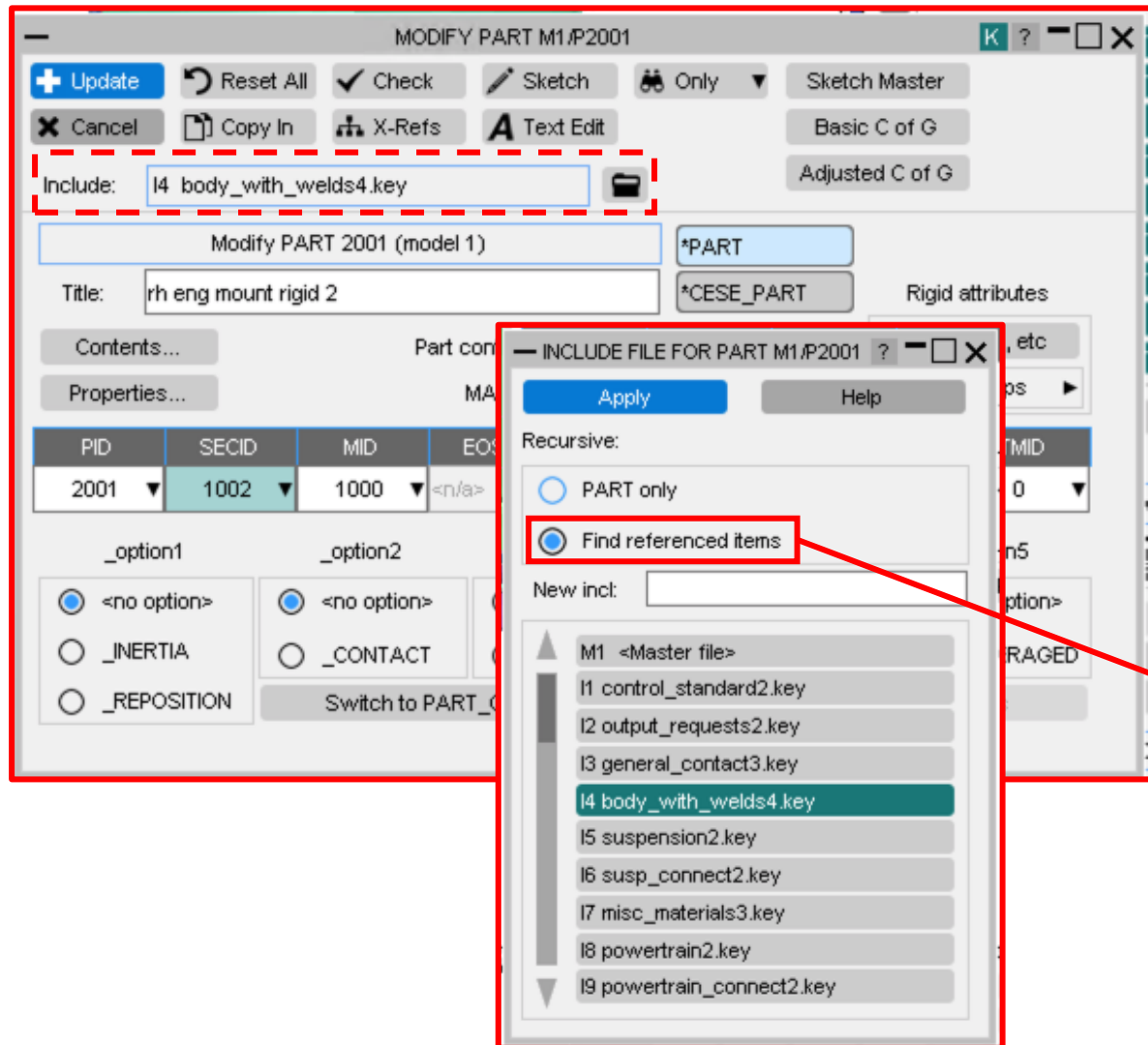


Methods to Find Which INCLUDE File an Item is in

- If you can see the item on the screen,
 - Use Quick-pick “Information”, click on the item
 - Alternatively, if the item is a part, use Quick-pick “Locate in Tree” to see it in the Part Tree under its Include file
- If you know the name or ID of a part
 - Part Tree “Find”
- If the entity type has a Modify menu, use it to edit the entity – the menu tells you which INCLUDE file it is in
- Or,
 - Put the item on the clipboard, right-click and LOCATE

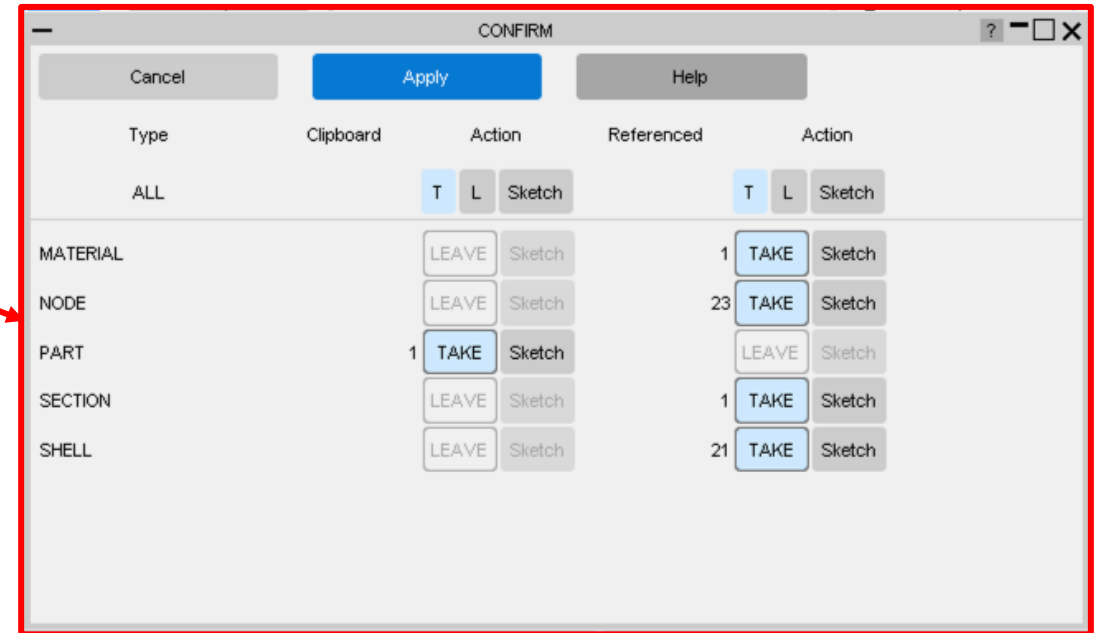


INCL File Location



The EDIT menus offer viewing and modification of the INCLUDE file location of each entity.

Try editing Part 2101, change the INCL location to body_with_welds. Check the “find referenced items” button to ensure that nodes and elements are also moved across.



Moving Entities to a Different INCLUDE File

2

3

2

5

4

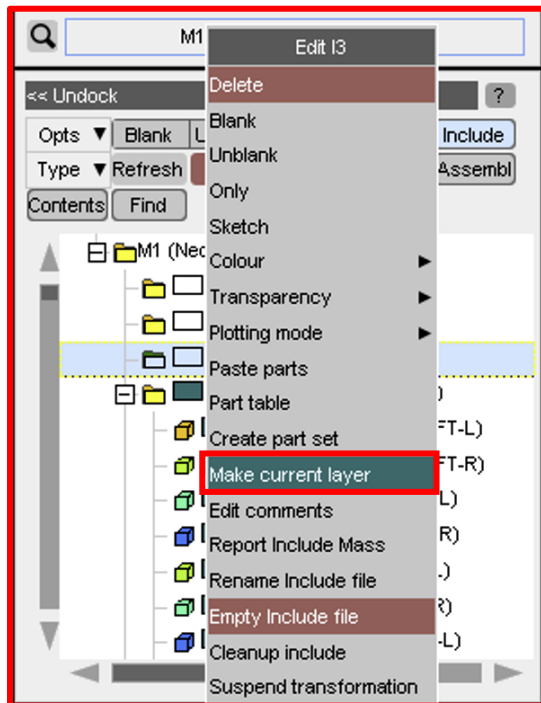
Clipboard

The clipboard can be used to store a selection of entities. Its contents can be accessed when selecting from other menus. There are also certain model management functions available within the clipboard menu.

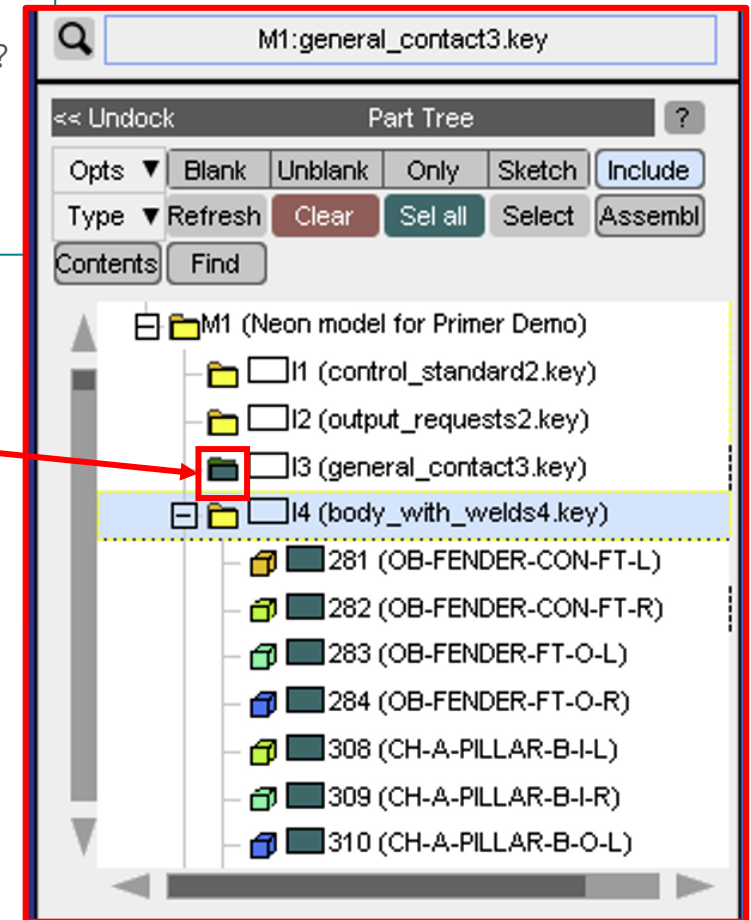
1. Go to **Tools – Clipboard**, to display the clipboard.
2. Click **+(ADD)**, and select object type **PARTS**. Select some parts.
3. Click **Move to INCLUDE file**.
4. **Browse** for the INCLUDE file. Activate the **Find referenced items** button, and click **Apply**.
5. The user can select which entity types will be moved.

INCLUDE File for Newly Created Entities

- If the user copies or modifies data, the modified or new data is put into the same INCLUDE file as the original data.
- When the user creates new data, how does PRIMER decide into which INCLUDE file it should be put?
- Answer: the “Current Layer” means the INCLUDE file for newly created data.
- To set the current layer, right-click on Include file in Part Tree, Make Current Layer.



Current layer is displayed in green

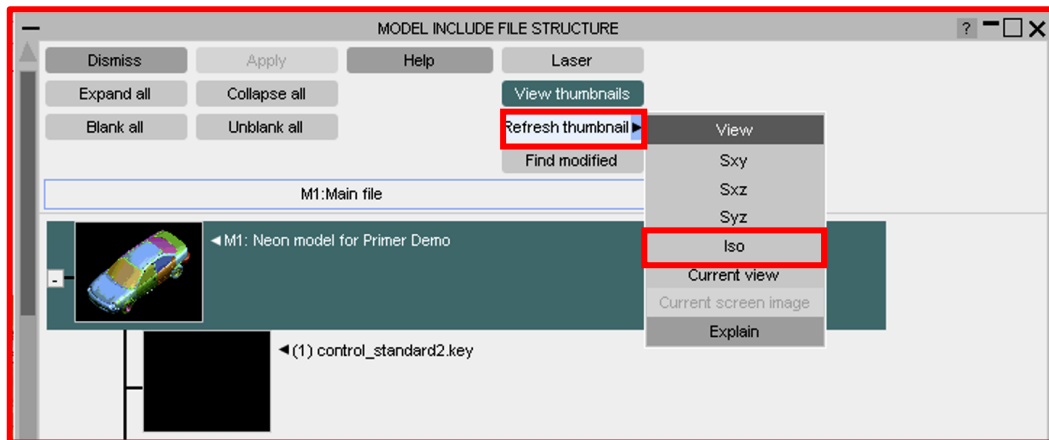
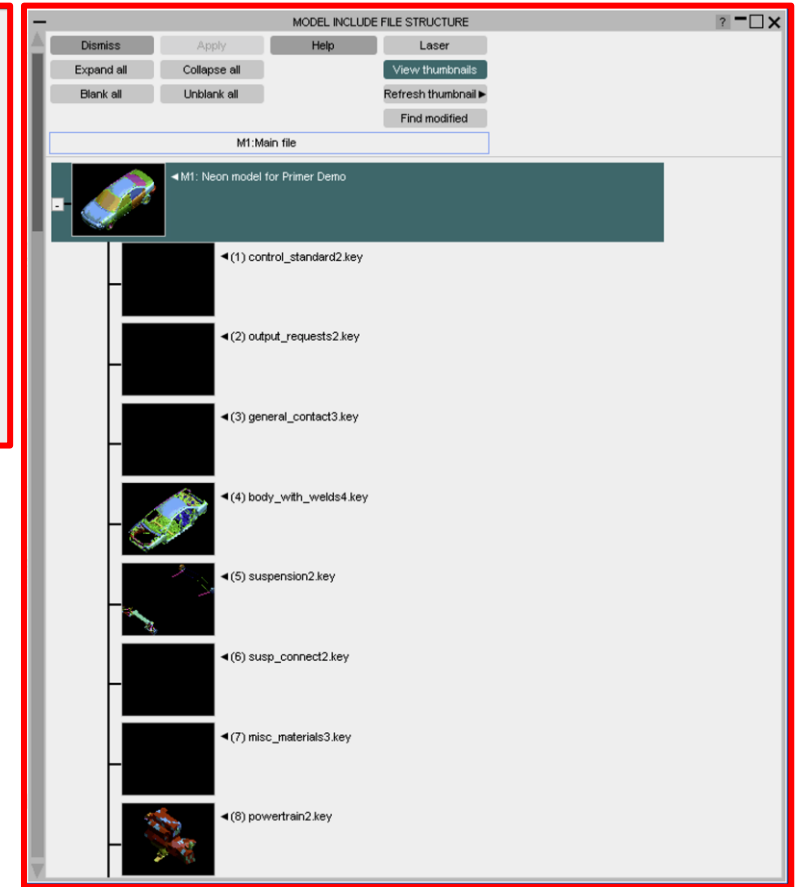
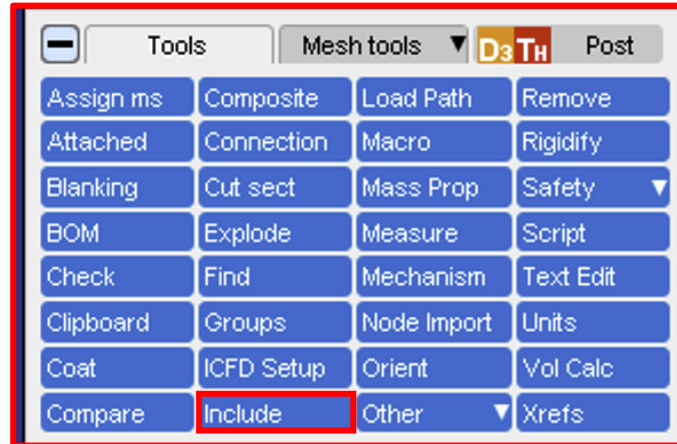


The INCLUDE Menu

Go to Tools=>INCLUDE. The model's INCLUDE file structure appears. This menu is similar to the Part Tree but with greater functionality.

The pictures are called Thumbnails. The blank images are for INCLUDE files that have no drawable data – e.g. the first INCLUDE contains only control cards. Try pressing Hide Thumbnails, then View Thumbnails.

The thumbnails are stored in the keyword files. Right-click on Refresh Thumbnail, choose ISO view. PRIMER creates a thumbnail for each INCLUDE file.



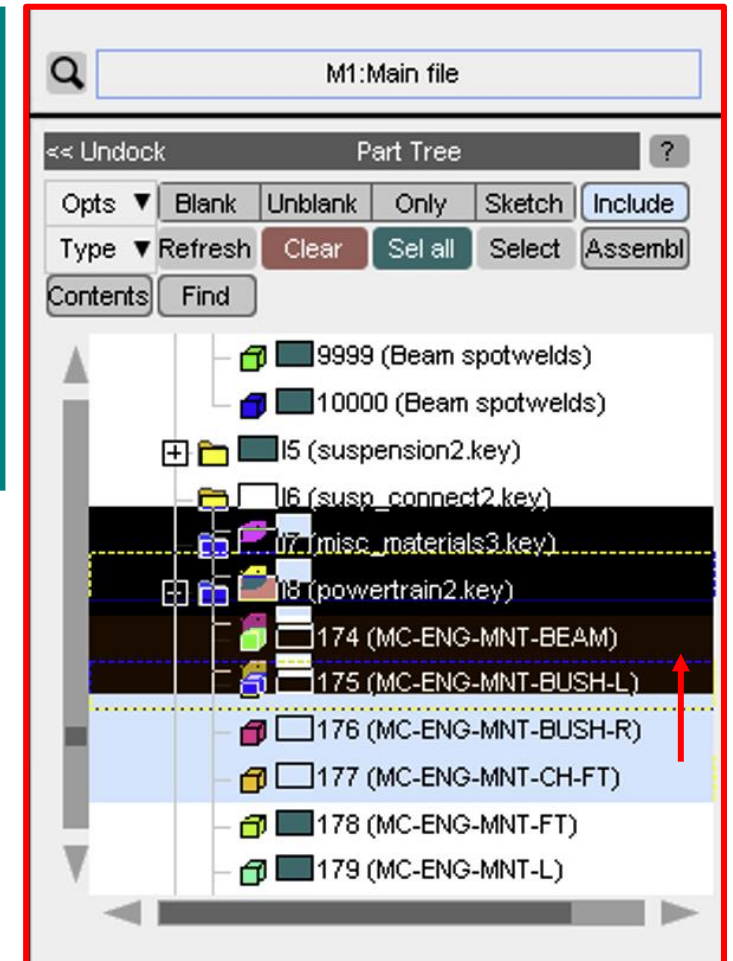
Splitting the Model into INCLUDE Files

A simpler method of moving Parts to a different INCLUDE file (with their nodes and elements) is to drag them in the Part Tree. Multiple parts can be selected by shift-click or CTRL-click.

While dragging, the cursor symbol changes when a valid “drop point” is reached. The parts will be put into the Include file that is under the cursor when the mouse button is released.

Alternatively, select as before => right-click => Cut => select INCLUDE file, => right-click => Paste.

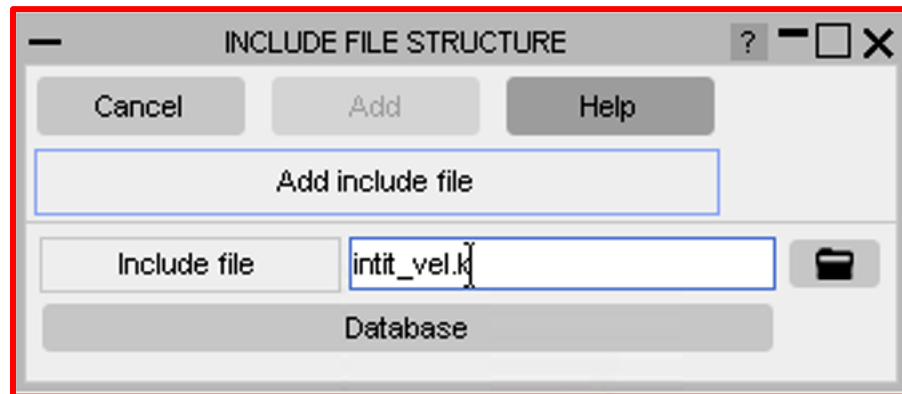
This process gives the same effect as Clipboard => Move to Include with option “Find Referenced Items”



Adding a New INCLUDE File

We will create a new INCLUDE file in which to put the Initial Velocity definition.

1. Right-click off the Master Model. Select ADD NEW CHILD, *INCLUDE.
2. Enter a name for the new INCLUDE `init_vel.k`, press ADD.
3. PRIMER automatically makes the new INCLUDE file the Current Layer (shown in this menu by the green stripe).

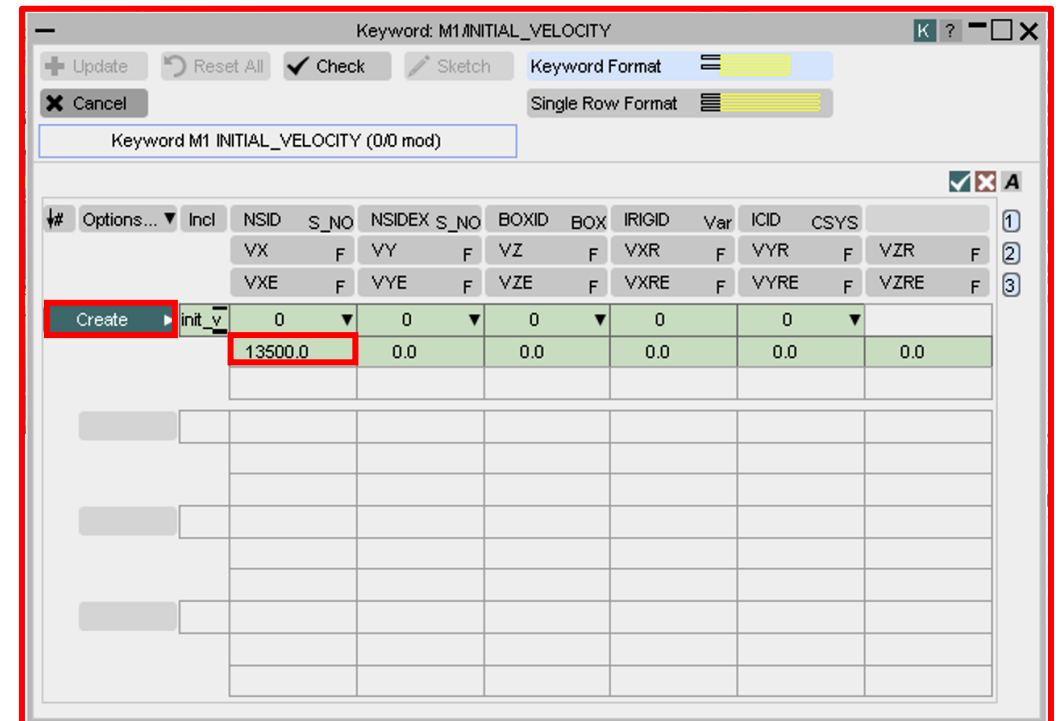
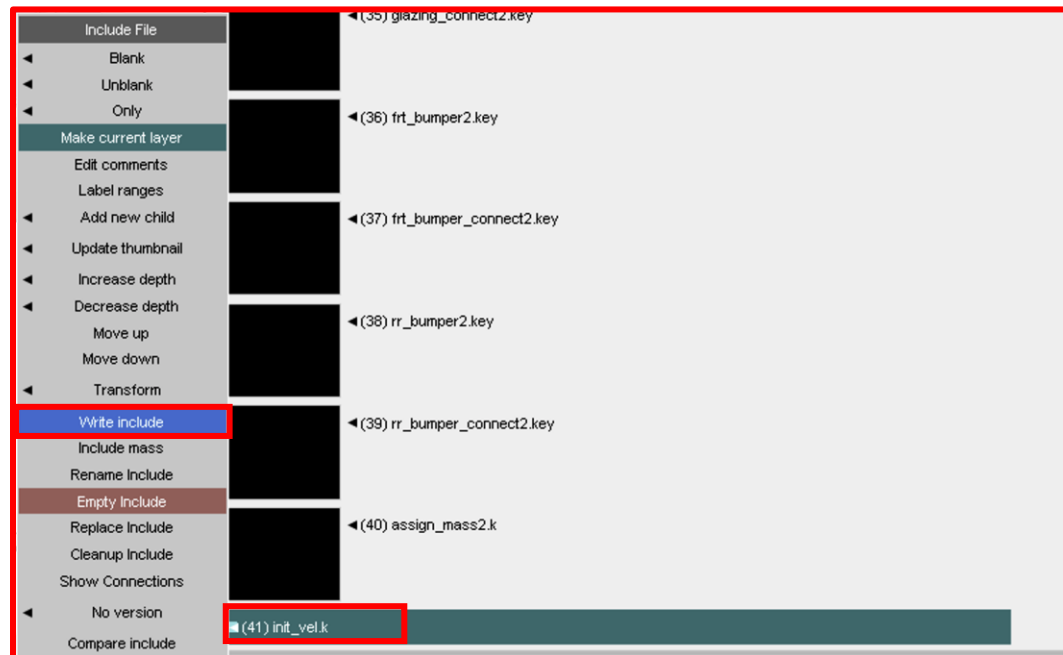


Adding a New INCLUDE File

Use Keyword=>INITIAL_VELOCITY to create an initial velocity for the whole model, VX=13500 as shown.

Press CREATE and UPDATE. This data will be put in the new INCLUDE file init_vel.k, because this is the current layer.

Using the Include menu, right-click off init_vel.k, WRITE INCLUDE. Use a text editor to check the contents of the file written out.



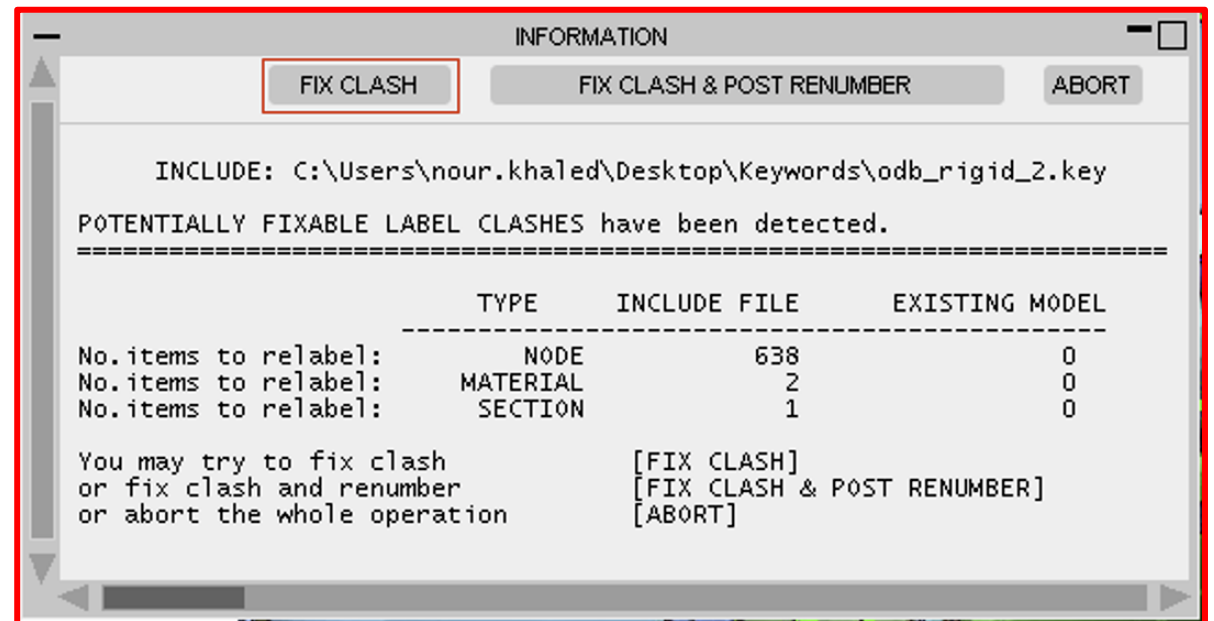
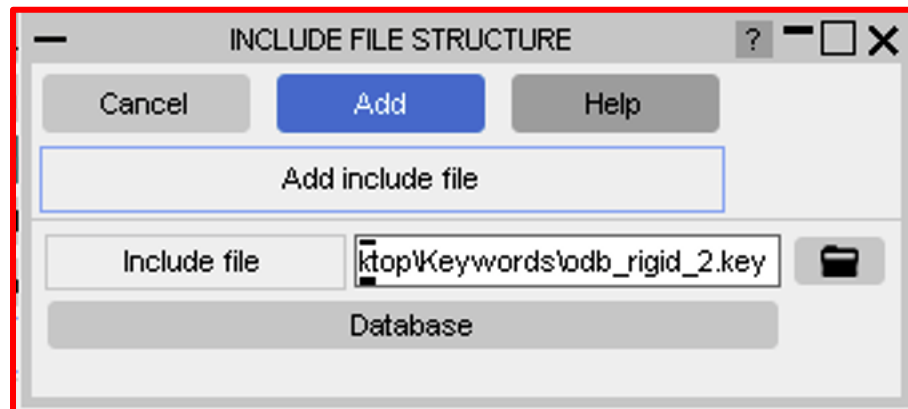
Adding Existing INCLUDE

Next we will add a barrier into the model. We have already seen how to do this by merging models; that method puts the barrier data into the master model. Instead, we will use the Include menu to add the barrier as an INCLUDE file.

Again, we use ADD NEW CHILD to add an INCLUDE to the master model. This time, instead of typing a new filename, browse for INCLUDE_FILES/LOADCASE/odb_rigid_2.key, press ADD.

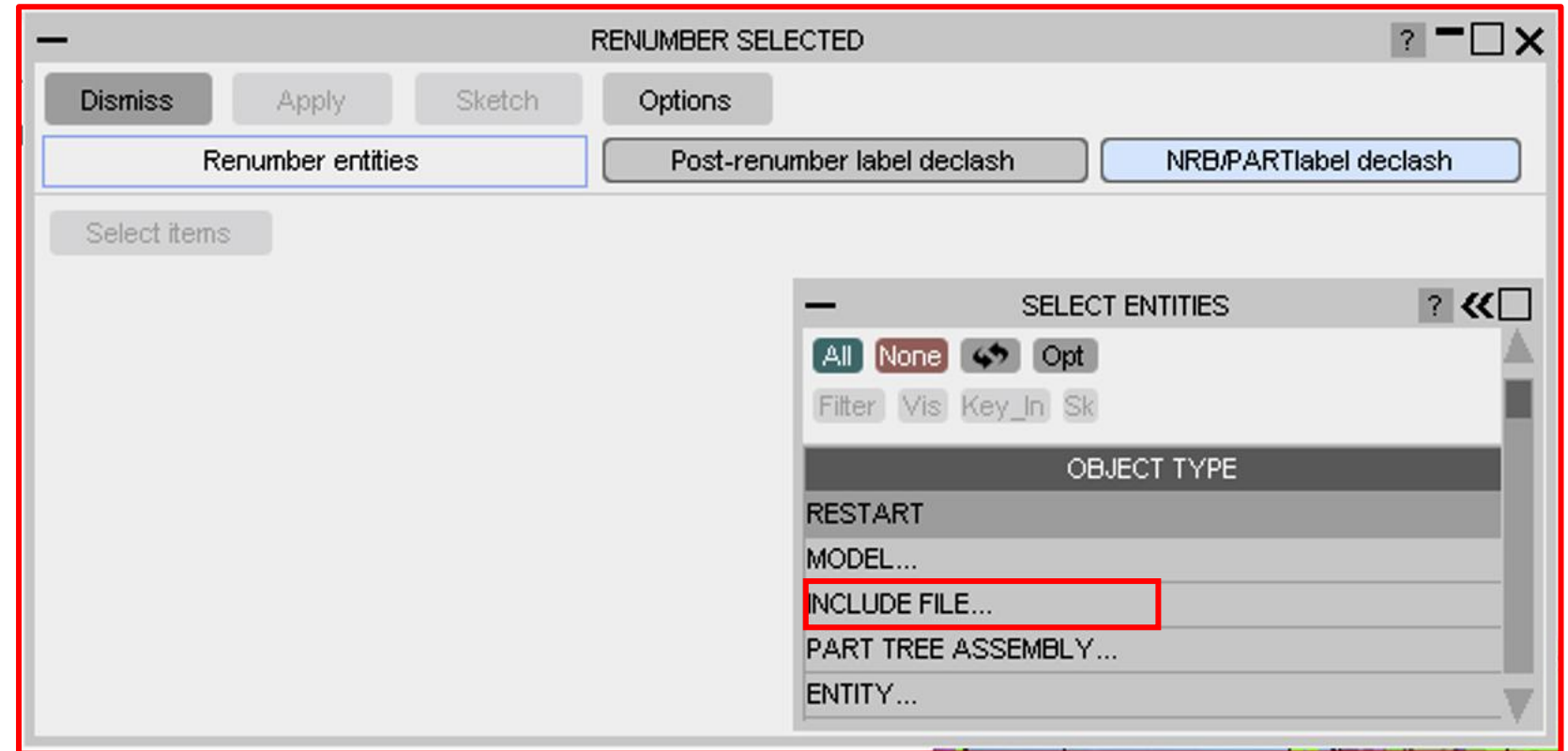
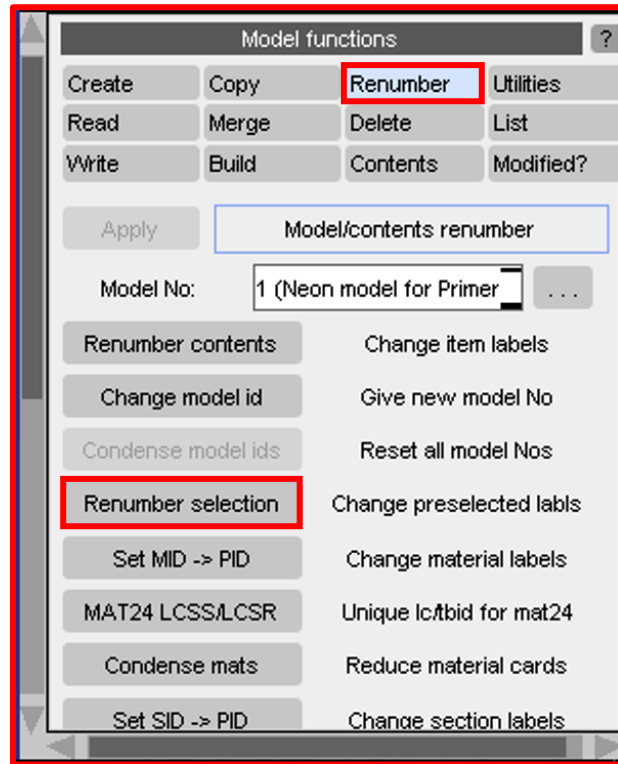
When adding INCLUDE files by this method, PRIMER is able to resolve label clashes. Press FIX_CLASH – we can adjust the numbering later.

The barrier appears as a new INCLUDE in the model.



Numbering an INCLUDE File

- To Renumber an Include File, use Renumber Selection
- Select Object Type INCLUDE FILE
- Select the include file odb_rigid_2



Numbering an INCLUDE File

- Set Renumber to **Yes** for all entity types.
- Type in 1000000 for “start at”. PRIMER detects that a clash will occur with existing entities. Change “start-at” to 3000000 – now there are no clashes. (other ways are available to resolve numbering clashes, using the drop-down Action menu).
- Press **APPLY**.

RENUMBER SELECTED

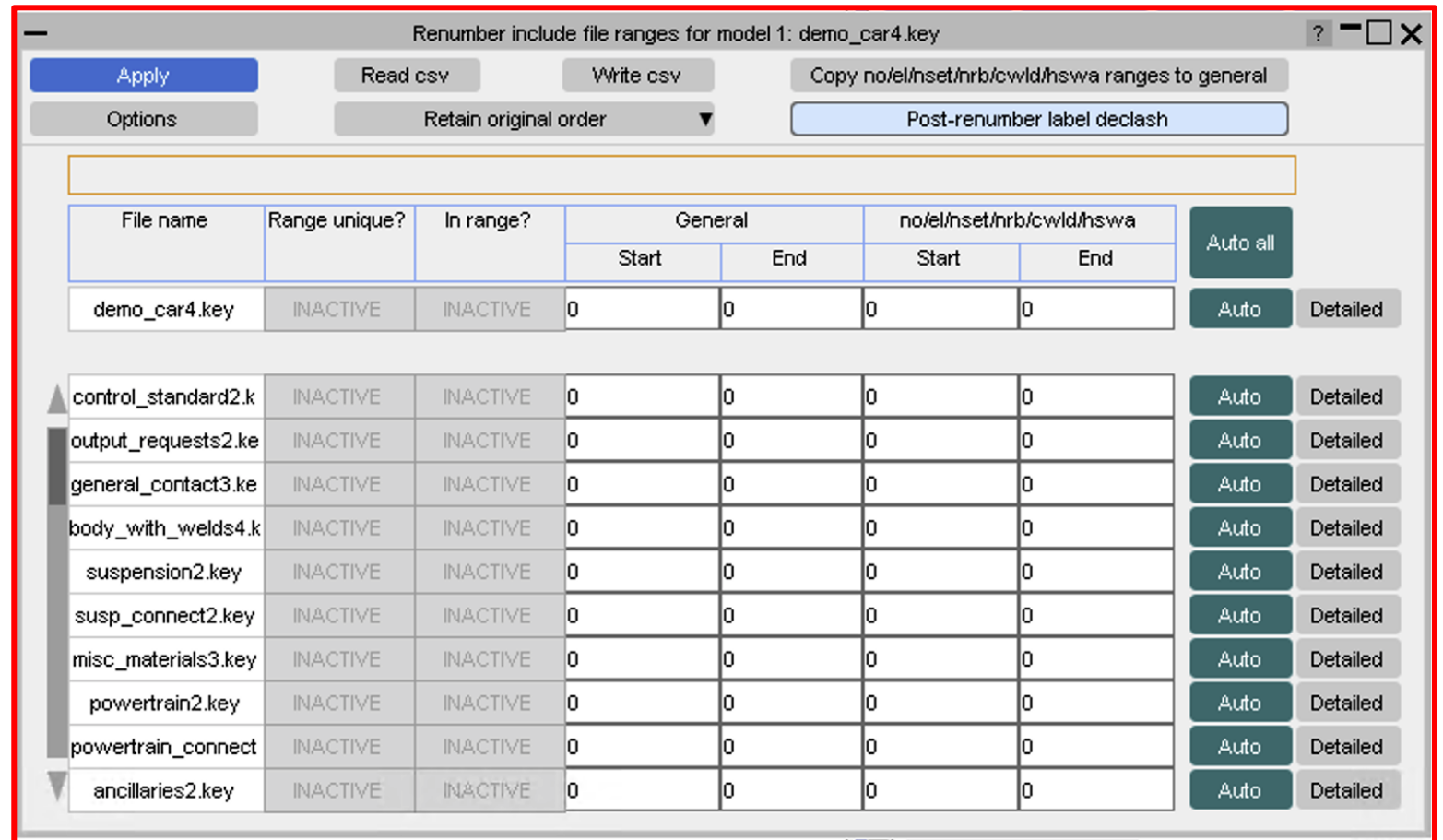
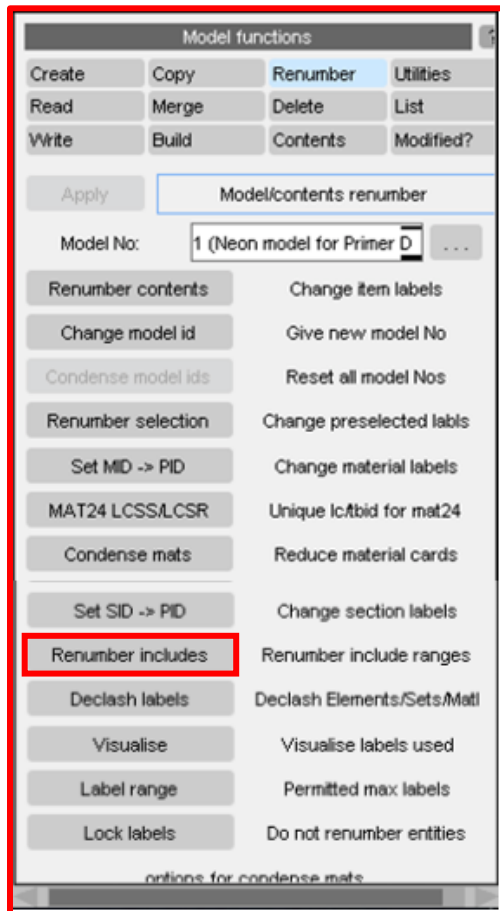
Dismiss Apply Sketch Options

Renumber entities Post-renumber label declash NRB/PARTlabel declash

Type	Renumber	low:high	start at	offset	Information	Action
ALL TYPES	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	id low:high (num sel)	1		LIST STATU	GLOBAL ACTION ▼
NODE	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:638 (638/297057)	1		CLASH	Do not renumber ▼
SOLID	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:360 (360/3184)	1		NO CLASH	No action needed ▼
SET_NODE	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:1 (1/122)	1		NO CLASH	No action needed ▼
SET_PART	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:1 (1/8)	1		NO CLASH	No action needed ▼
PART	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:2 (2/377)	1		NO CLASH	No action needed ▼
MATERIAL	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:2 (2/323)	1		CLASH	Do not renumber ▼
SECTION	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:1 (1/325)	1		CLASH	Do not renumber ▼
CONTACT	<input checked="" type="checkbox"/> YES <input type="checkbox"/> NO	1:1 (1/4)	1		NO CLASH	No action needed ▼

Numbering an INCLUDE File

- An alternative method is to use “Renumber includes”
- This allows you to specify ID ranges for each include file



Numbering an INCLUDE File

- Two ranges can be set for each include
 - One range for nodes/elements + some other types
 - One range for all other labelled entities.
 - Both ranges can be the same.
- Label ranges can typed in, or imported from a CSV file.
- Label ranges are saved as comments to each include file.
- The panel will tell you if you have entities within an include file which is outside the specified range.
- Entities can be renumbered into range through this panel.
- Ranges are also used during creation of entities when assigning ID's.

Renumber include file ranges for model 1: demo_car4.key

Apply Read csv Write csv Copy no/el/nset/nrb/cwld/hswa ranges to general

Options Retain original order Post-renumber label declash

Renumber include file ranges

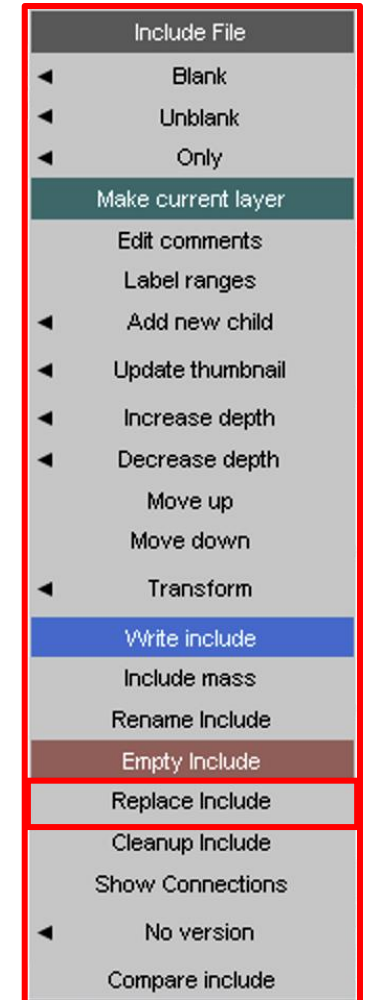
File name	Range unique?	In range?	General		no/el/nset/nrb/cwld/hswa		Auto all
			Start	End	Start	End	
demo_car4.key	INACTIVE	INACTIVE	0	0	0	0	Auto Detailed
control_standard2.k	INACTIVE	INACTIVE	0	0	0	0	Auto Detailed
output_requests2.ke	OK	NO	21000	21499	2000000	2000099	Auto Detailed
general_contact3.ke	OK	NO	20100	20199	2581000	3080999	Auto Detailed
body_with_welds4.k	OK	NO	20200	20299	2001000	2100999	Auto Detailed
suspension2.key	OK	NO	20300	20399	2101000	2200999	Auto Detailed
susp_connect2.key	OK	NO	20400	20499	2201000	2250999	Auto Detailed
misc_materials3.key	OK	NO	20500	20699	2251000	2350999	Auto Detailed
powertrain2.key	OK	NO	12000	12999	2351000	2550999	Auto Detailed
powertrain_connect	OK	YES	11000	11499	1200000	1399999	Auto Detailed
ancillaries2.key	OK	NO	20000	0	1100000	1149999	Auto Detailed

Replacing an INCLUDE File

As an example, we will now replace this barrier model with a different one that has already been renumbered and positioned.

Use the Include menu to remove the barrier (right click, REPLACE INCLUDE).

Browse for INCLUDE_FILES/LOADCASE/odb_rigid_moved.key



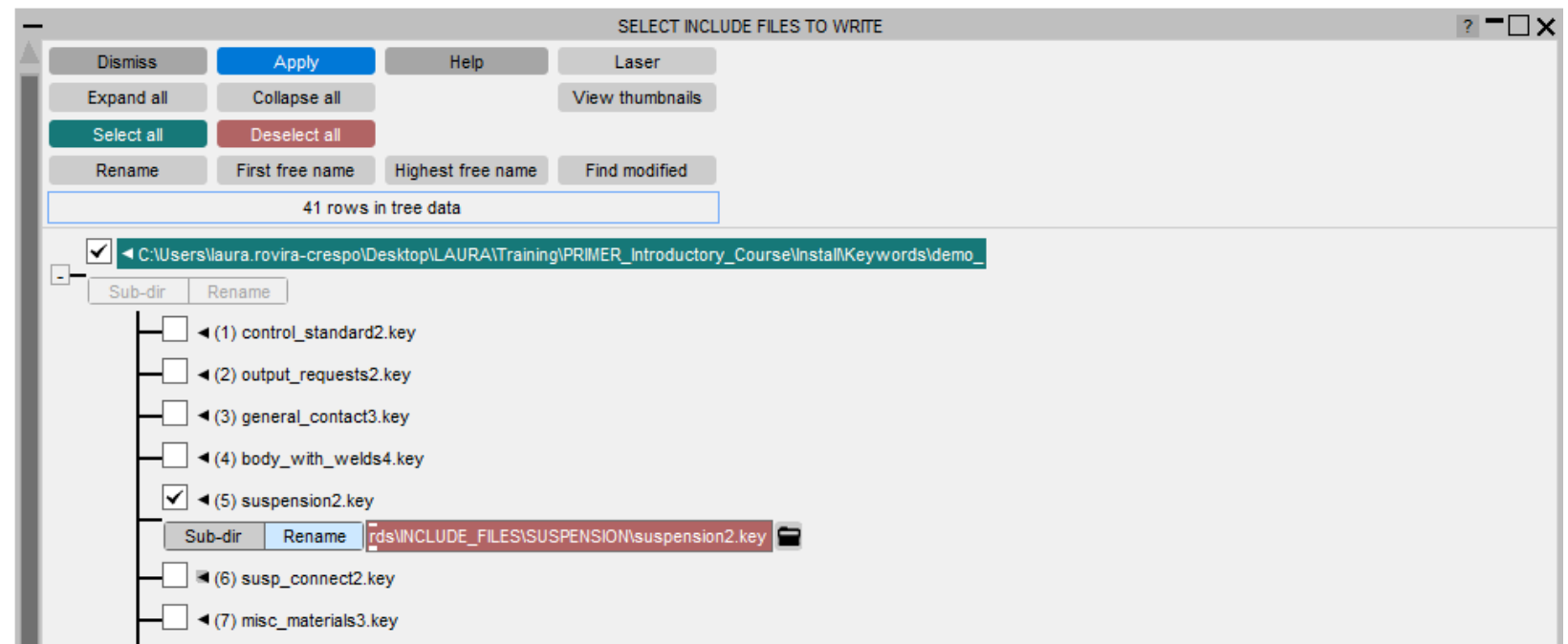
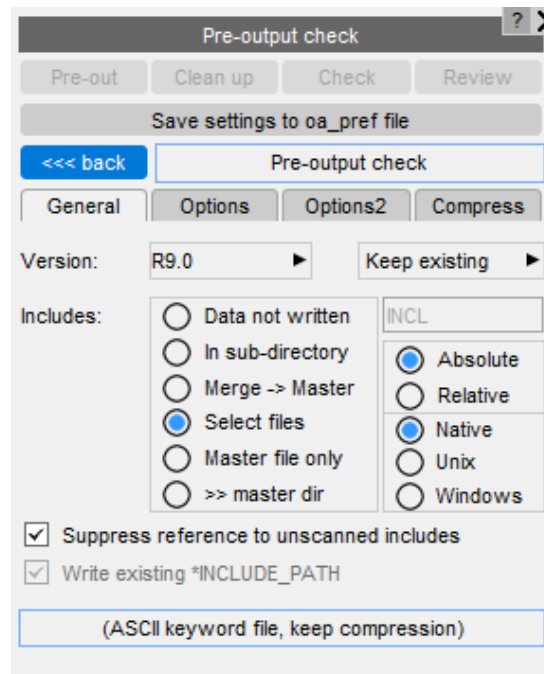
Other Functions in the INCLUDE Menu

- The Include menu is used to create and edit INCLUDE_TRANSFORM data.
- The INCLUDE structure of the model can be modified, e.g. flatten a multi-layer include structure
- Label Ranges may be defined for each INCLUDE – an error check and autofix is available to ensure that the contents of the INCLUDE obey the given ranges; you can apply the given ranges from this menu
- There are facilities to associate a design-level (“version”) with an INCLUDE file



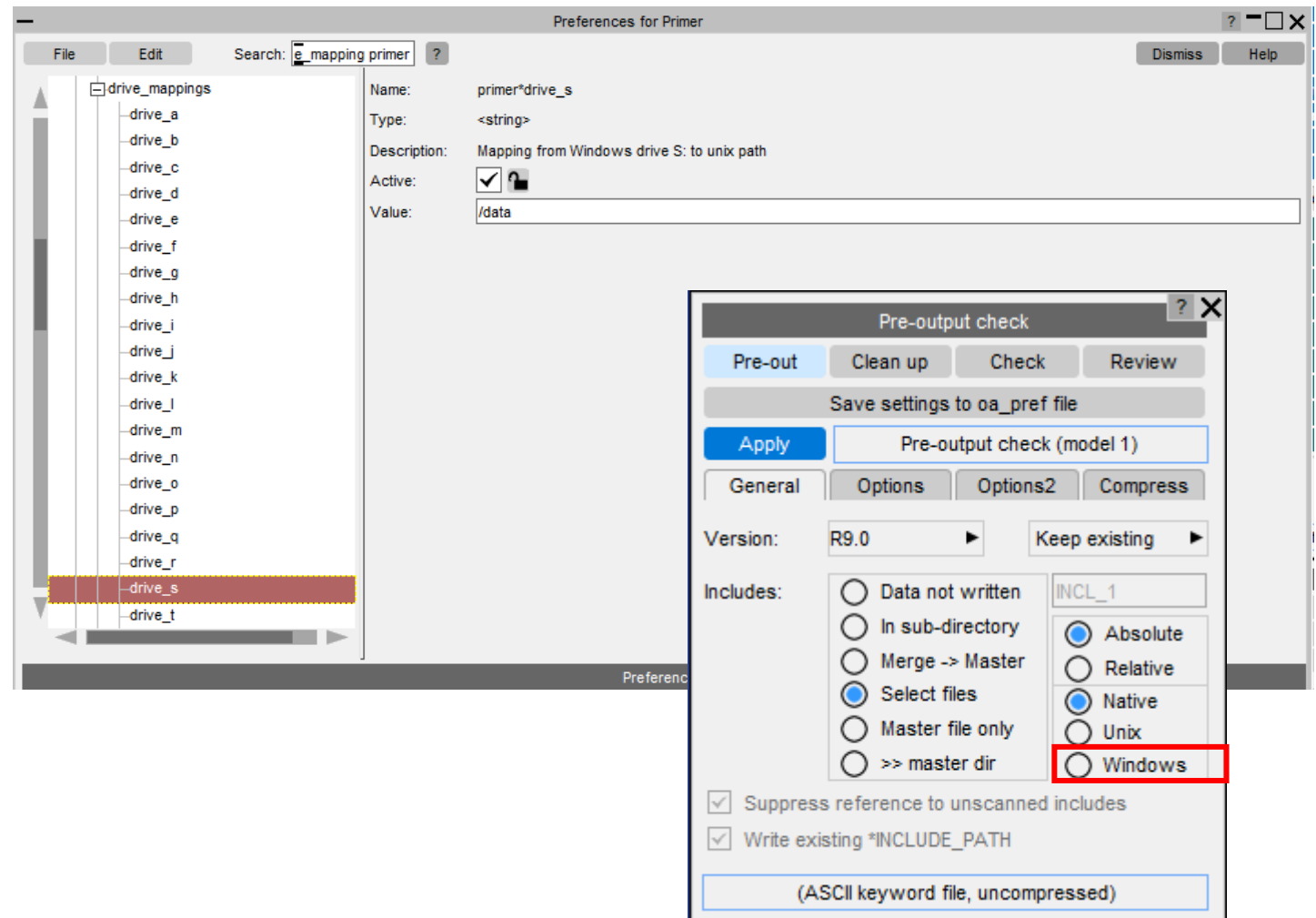
Writing INCLUDE Files

- We have already seen how to write an INCLUDE file from the Include menu.
- Now try Model=>Write, opt for Select Files.
- Check the box for suspension2.key (you can check as many INCLUDE files as you wish).
- Option SUB-DIR means that PRIMER will create a new subdirectory, INCL, INCL_1, INCL_2 etc in which to put the requested INCLUDE files.
- Option RENAME means that PRIMER will write the INCLUDE file to the same directory that it came from. A red text box for the file name means that the original file will be overwritten – change the name before writing out.



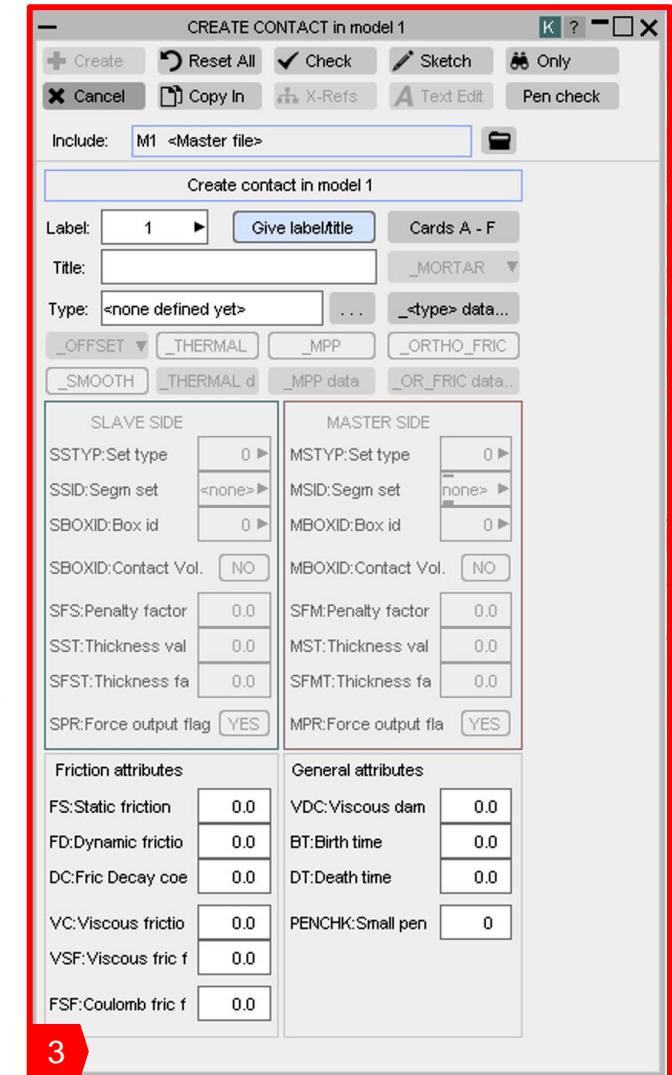
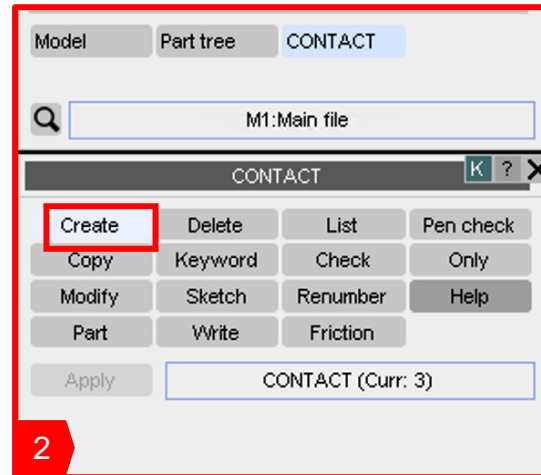
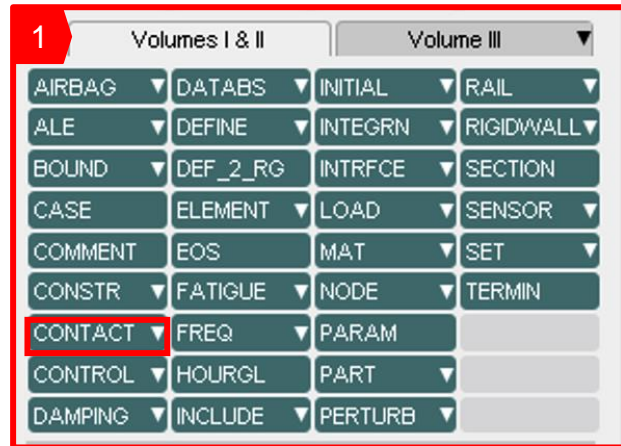
Note about Windows /Unix Systems

- It is quite common for pre/post to be performed on Windows, but to run LS-DYNA on linux. PRIMER allows for this combination:
 - Windows: `S:\models\model1.key`
 - Unix: `/data/models/model1.key`
- “Drive mappings” can be set up as preferences (e.g. S: = /data)
- Unix-style paths (forward-slash) can be written by setting Unix style on the Write menu – this can also be set as a preference



Contacts, Contact Penetration Checking and Fixing

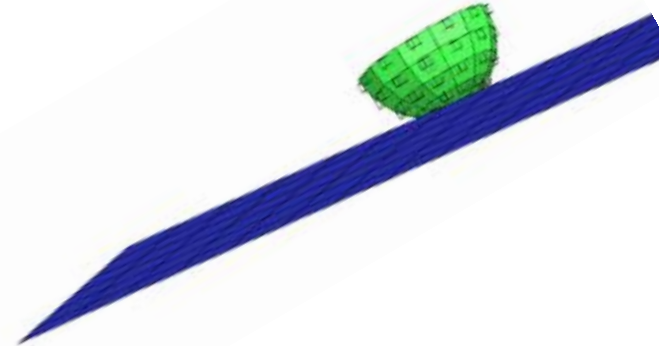
Exercise: Create a Contact



Create a new contact

1. Click **CONTACT**.
2. Click **Create**.
3. The create contact window pops up.

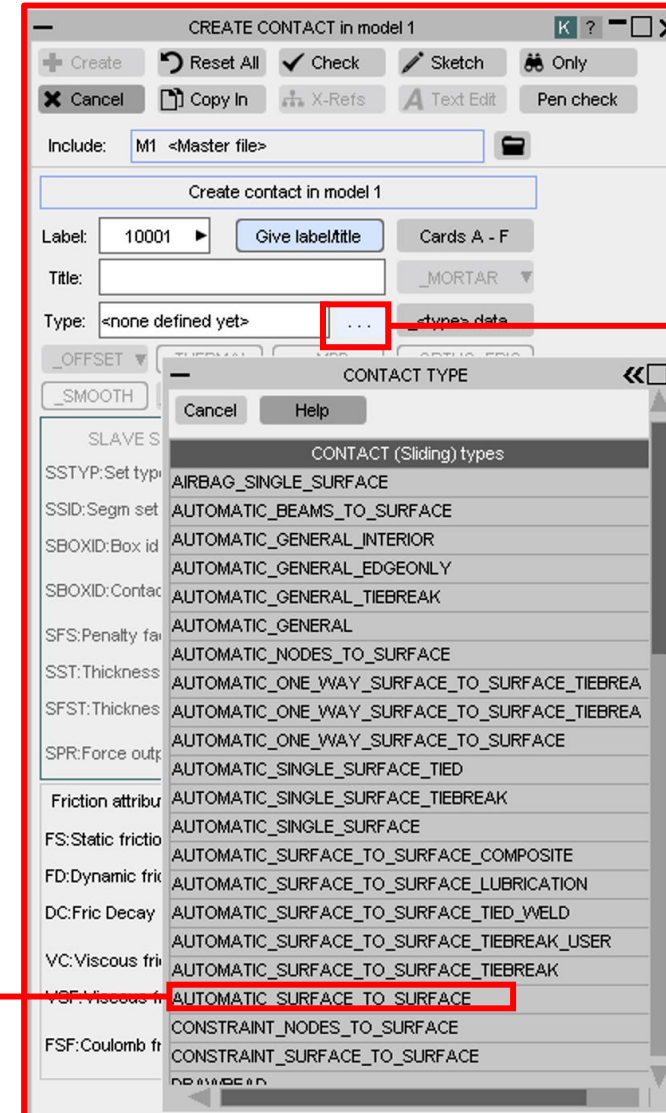
Primer's create contact window makes it easier to create contacts. Note the keyword are written out in the correct formatting.



Exercise: Create a Contact

Create a new contact

- Next, define the type of contact you want.
- There are many possibilities (discussed further in Introduction to LS-DYNA course). For this contact, select **AUTOMATIC_SURFACE_TO_SURFACE**



Exercise: Create a Contact

CREATE CONTACT in model 1

Include: M1 <Master file>

Create contact in model 1

Label: 10001 Give label/title Cards A - F

Title: No _MORTAR

Type: IC_SURFACE_TO_SURFACE <type> data...

_OFFSET _THERMAL _MPP _ORTHO_FRIC

_SMOOTH _THERMAL d _MPP data _OR_FRIC data...

SLAVE SIDE MASTER SIDE

SSTYP: Set type 0 Slave set type (SSTYP) 0

SSID: Segm set <none> 0 (Segment SET)

SBOXID: Box id 0 1 (Shell SET) 0

SBOXID: Contact Vol. NO 3 (PART id) 0

SFS: Penalty factor 0.0 4 (Node SET) 0

SST: Thickness val 0.0 5 (All) 0

SFST: Thickness fa 0.0 6 (Exempted Part SET) 0

SPR: Force output flag YES MPR: Force output fla YES

Friction attributes General attributes

FS: Static friction 0.0 VDC: Viscous dam 0.0

FD: Dynamic frictio 0.0 BT: Birth time 0.0

DC: Fric Decay coe 0.0 DT: Death time 0.0

VC: Viscous frictio 0.0 PENCHK: Small pen 0

VSF: Viscous fric f 0.0

FSF: Coulomb fric f 0.0

Create a new contact

- Click on the **SSTYP** (slave set type)
- There are many possibilities (discussed further in Introduction to LS-DYNA course).
- For the first part of this exercise, select **Part id**

Exercise: Create a Contact

CREATE CONTACT in model 1

Include: M1 <Master file>

Create contact in model 1

Label: 10001 Give label/title Cards A - F

Title: No _MORTAR

Type: IC_SURFACE_TO_SURFACE

SLAVE SIDE

SSTYP:Set type 3

SSID:Part id 2

SBOXID:Box id 0

SBOXID:Contact Vol. NO

MASTER SIDE

MSTYP:Set type 3

MSID:Part id 3

MBOXID:Box id 0

MBOXID:Contact Vol. NO

SFS:Penalty factor 0.0

SST:Thickness val 0.0

SFST:Thickness fa 0.0

SPR:Force output flag YES

SFM:Penalty factor 0.0

MST:Thickness val 0.0

SFMT:Thickness fa 0.0

MPR:Force output fla YES

Friction attributes

FS:Static friction 0.2

FD:Dynamic frictio 0.2

DC:Fric Decay coe 0.0

VC:Viscous frictio 0.0

VSF:Viscous fric f 0.0

FSF:Coulomb fric f 0.0

General attributes

VDC:Viscous dam 0.0

BT:Birth time 0.0

DT:Death time 0.0

PENCHK:Small pen 0

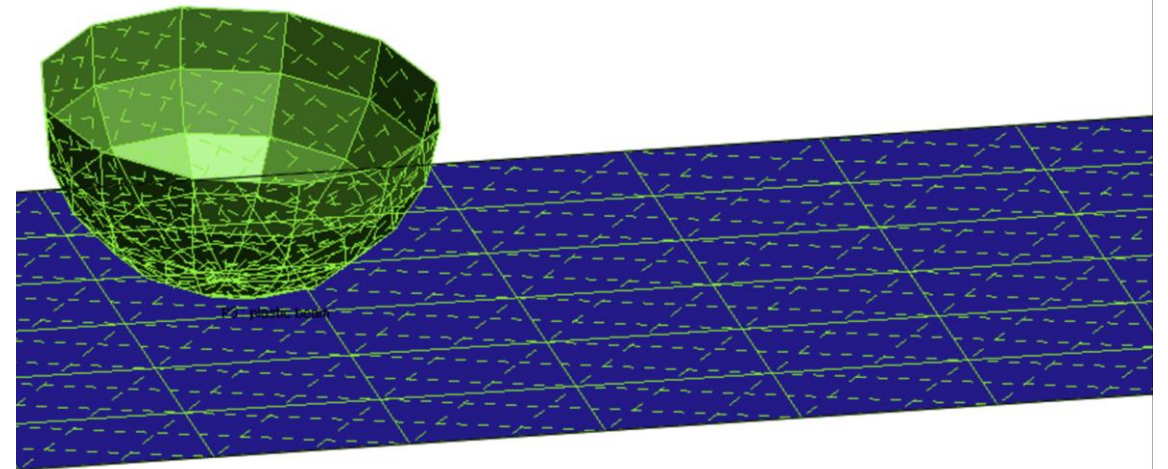
9

10

11

Create a new contact

9. Fill out the remainder of the input variables for master and slave side.
10. Enter any penalty stiffness factors, thickness values or thickness factors.
11. Enter the coefficient of friction values for static and dynamic friction.
12. Click on Create contact.
13. Write out your model.



Exercise: Create a Contact

CREATE CONTACT in model 1

Include: M1 <Master file>

Create contact in model 1

Label: 10001 Give label/title Cards A - F

Title: No _MORTAR

Type: JC_SURFACE_TO_SURFACE <type> data...

_OFFSET _THERMAL _MPP _ORTHO_FRIC

_SMOOTH _THERMAL d _MPP data _OR_FRIC data...

SLAVE SIDE

SSTYP: Set type 0

SSID: Segm set <none>

SBOXID: Box id 0

SBOXID: Contact Vol. NO

SFS: Penalty factor 0.0

SST: Thickness val 0.0

SFST: Thickness fa 0.0

SPR: Force output flag YES

Friction attributes

FS: Static friction 0.0

FD: Dynamic friction 0.0

DC: Fric Decay coe 0.0

VC: Viscous frictio 0.0

VSF: Viscous fric f 0.0

FSF: Coulomb fric f 0.0

MASTER SIDE

MSTYP: Set type 0 Master set type (MSTYP)

MSID: Segm set <none> 0 (Segment SET)

MBOXID: Box id 0 1 (Shell SET)

MBOXID: Contact Vol. NO 2 (Part SET)

SFM: Penalty factor 0.0 3 (PART id)

MST: Thickness val 0.0 5 (All)

SFMT: Thickness fa 0.0 6 (Exempted Part SET)

MPR: Force output fla YES 7 (Part tree SET)

General attributes

VDC: Viscous dam 0.0

BT: Birth time 0.0

DT: Death time 0.0

PENCHK: Small pen 0

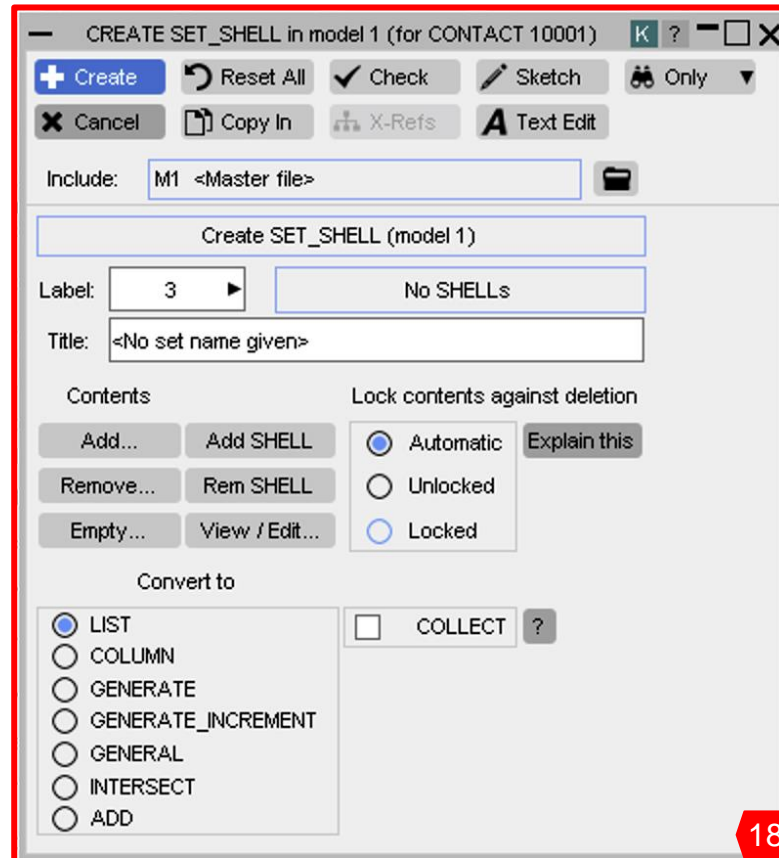
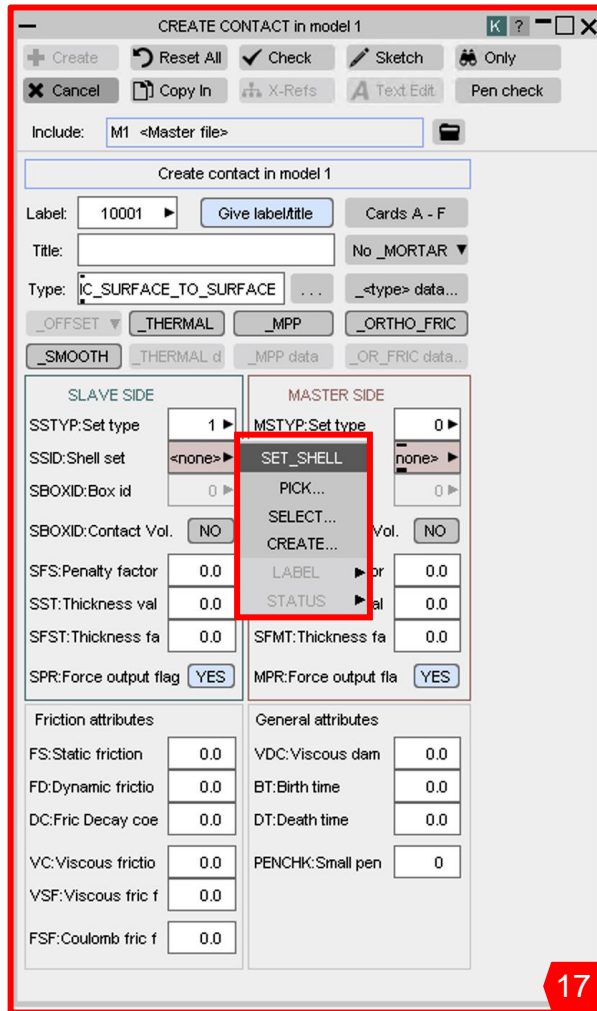
Y

X

Create a new contact

14. Repeat the exercise by selecting a different slave set type.
15. Try Shell Set, Node Set and Segment Set

Exercise: Create a Contact



Create a new contact

16. Sets can be created directly from the SSID menu.
17. Right click and click Create.
18. This takes you to the set creation Primer menu.

Exercise: Create a Contact

Create a new contact

Sometimes you will need use a segment set to define your contact. Using segment sets is usually more robust. To create a segment set, you need to coat parts in segments. Primer helps you do this.

19. Change SSTYP to 0.
20. Right click on SSID and create segment set.
21. In the create set_segment menu, select **Coat elements**.
22. Choose **External faces** and Coat **Face**.
23. Click on the face of interest.
24. Click **Apply**.
25. Back in the create set menu, click **Create**.

19. Change SSTYP to 0.

20. Right click on SSID and create segment set.

21. In the create set_segment menu, select **Coat elements**.

22. Choose **External faces** and Coat **Face**.

23. Click on the face of interest.

24. Click **Apply**.

25. Back in the create set menu, click **Create**.

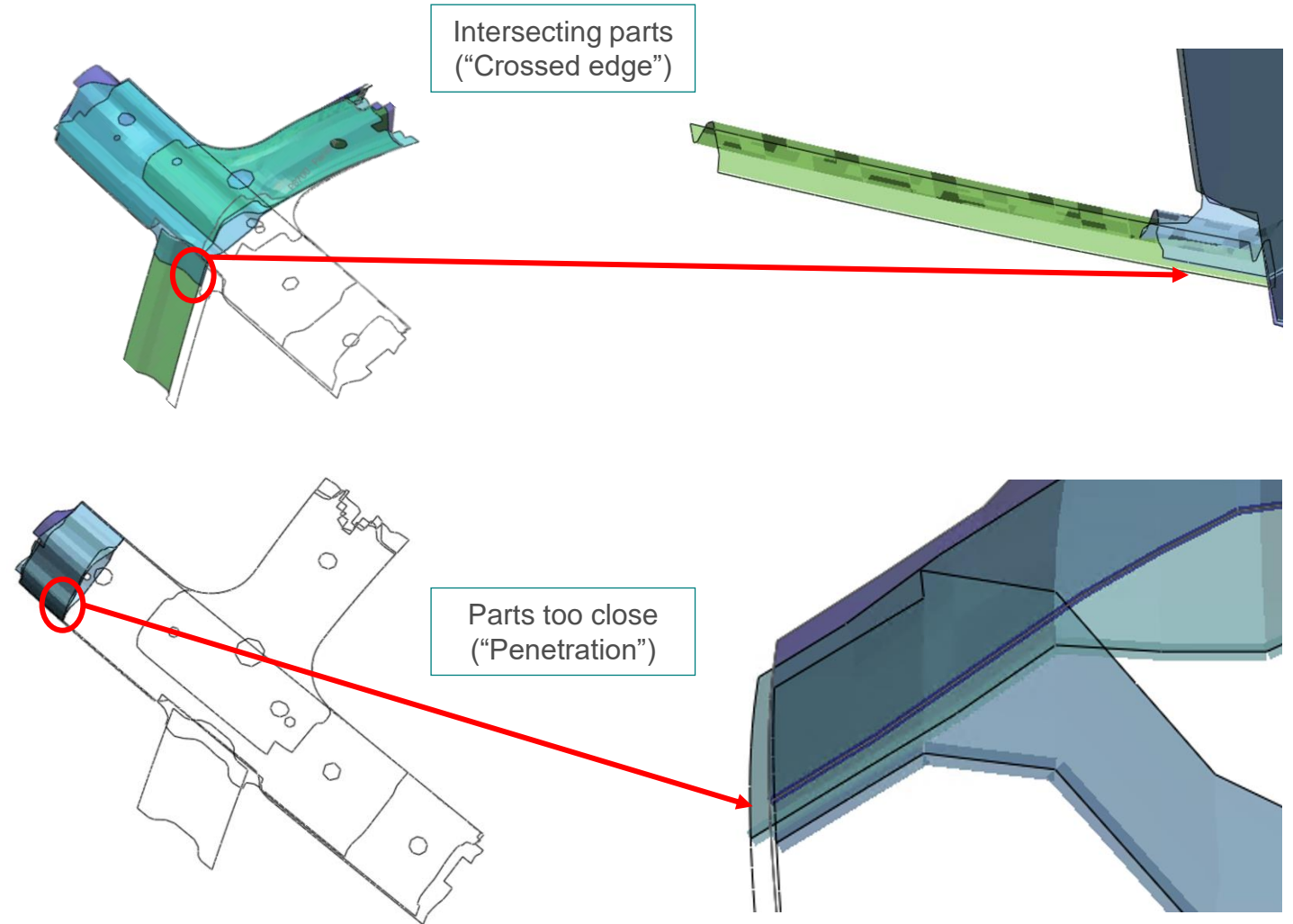
Penetration Checking

Delete the models from PRIMER.

Read joint_bpost.key.

Begin by looking at a cut-section. Use shortcut X then N, pick a node, switch “negative action” to “normal”. Drag the section through the model (shortcut D, then use mouse). PRIMER draws cut sections through shell elements using the true thickness. Observe where the panels are penetrating and where they are too close.

When you have finished dragging, shortcut Q returns to Quickpick mode.



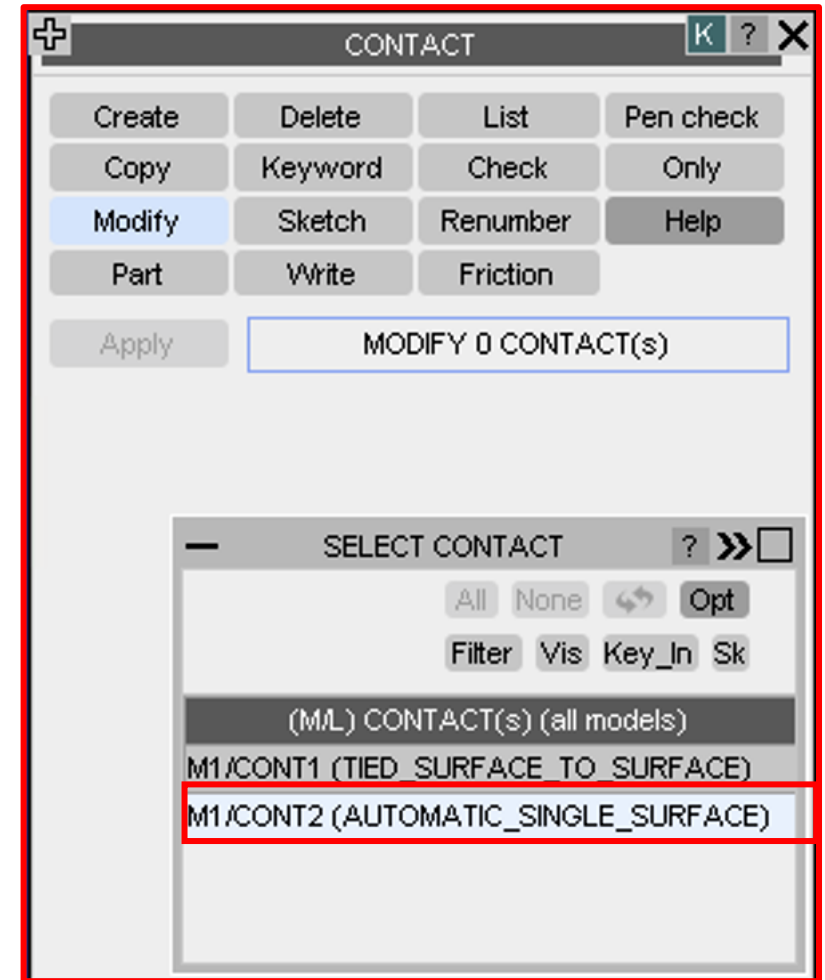
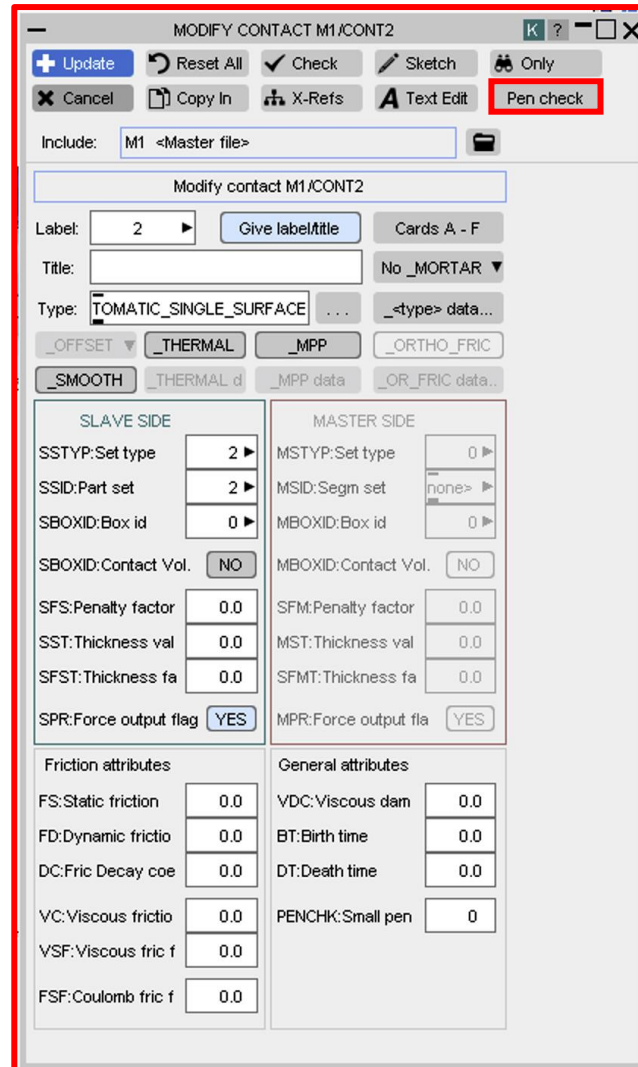
Penetration Checking

Turn off the cut section.

Start the Contact Penetration checker from

Keyword=>Contact=>Sliding&Tied=>

Modify (select the Automatic Single Surface contact) =>Pen Check



Penetration Checking

Contact Penetrations:

1. "Penetration" means that a node is too close to an element, i.e. within the contact thickness defined by $0.5(t1+t2)$, where $t1$ and $t2$ are the contact thicknesses of the contacting elements
2. "Crossed edge" means an intersection of one element plane with another
3. PRIMER understands the different methods of defining contact thickness, e.g. SST on *CONTACT, or on *PART_CONTACT.
4. PRIMER tries to use the same equations as LS-DYNA to determine penetrations.

PEN CHECK M1/CONT2 (for CONTACT 2) ? [X]

Dismiss Check all Options...

List Info Check visible Only

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE

<No title defined>

select parts sel none sel all sel xedge ?

List crossed ☒ List penetrating: ☒

P8231:P8710 xedge=5 pen=6 (1.11482)

P8710:P82151 xedge=0 pen=108 (0.299279)

P8712:P82151 xedge=0 pen=78 (0.199826)

penetration magnitude >= 0.0

sketch unblank recursive

<- Only Pen elements Only X'd elements

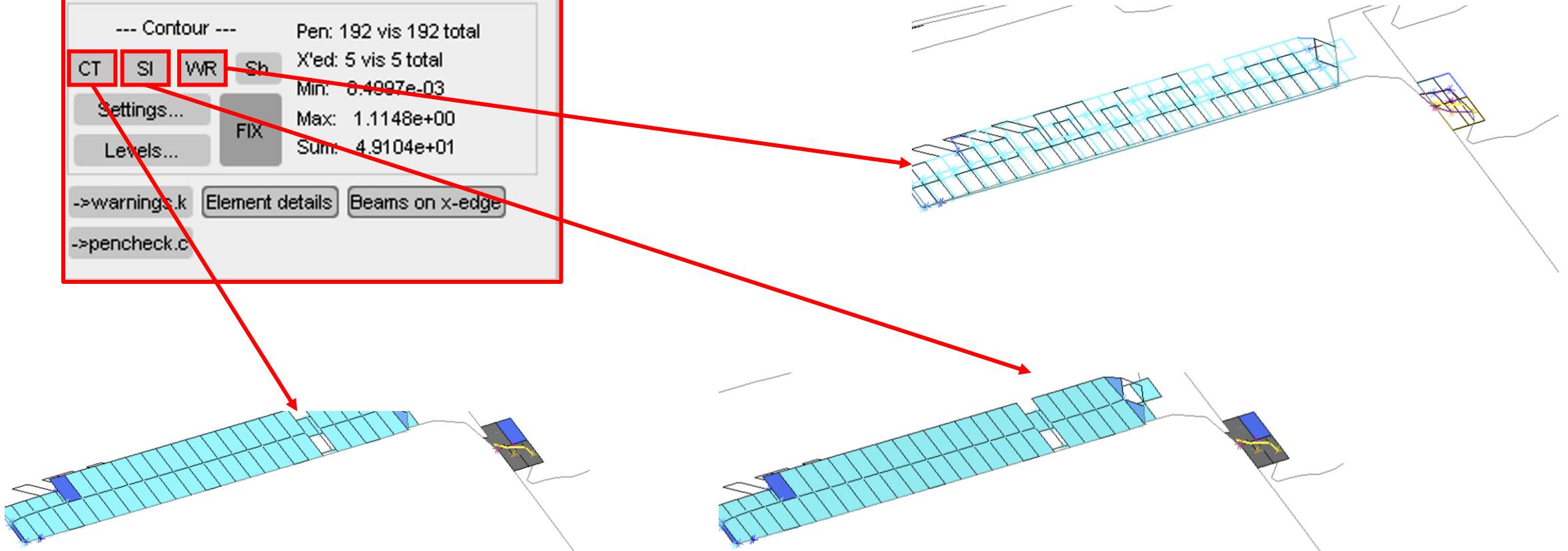
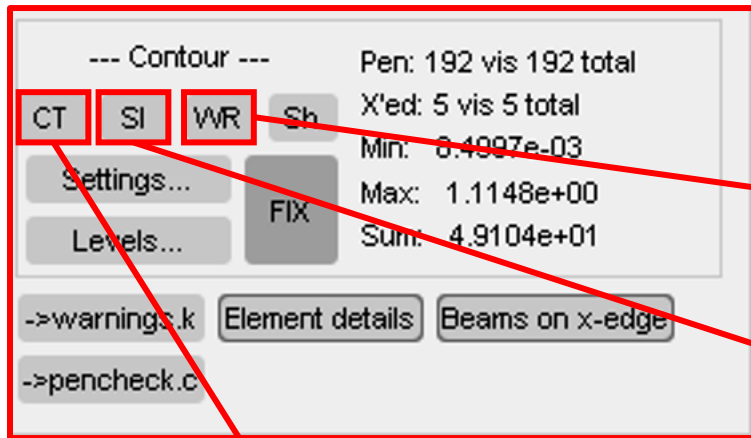
--- Contour --- Pen: 192 vis 192 total
X'ed: 5 vis 5 total
Min: 8.4997e-03
Max: 1.1148e+00
Sum: 4.9104e+01

Settings... Levels... FIX

->warnings.k Element details Beams on x-edge

->pencheck.c

Penetration Checking

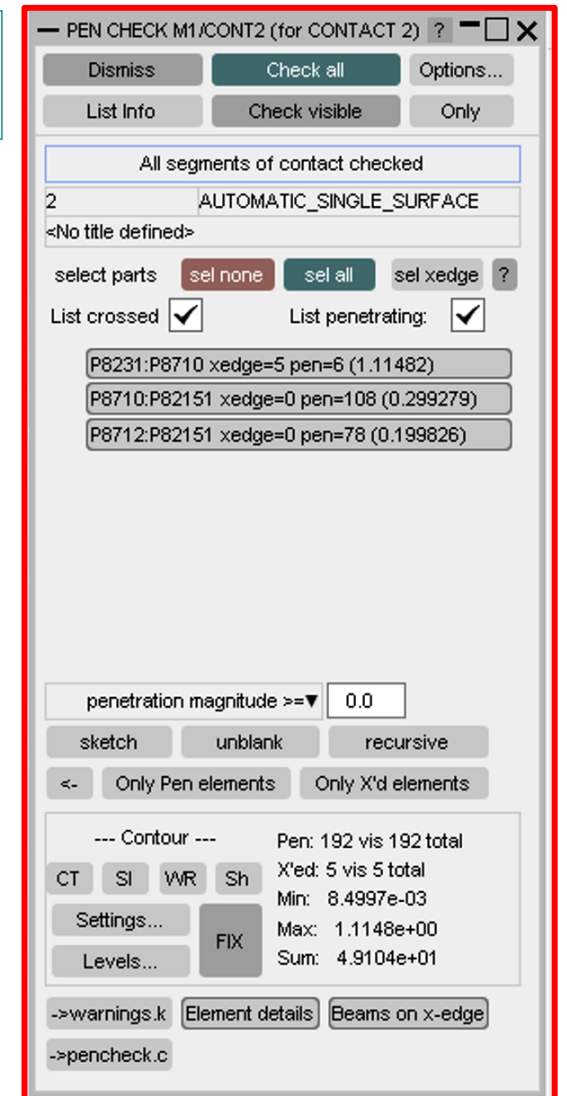


Try the different plotting modes.

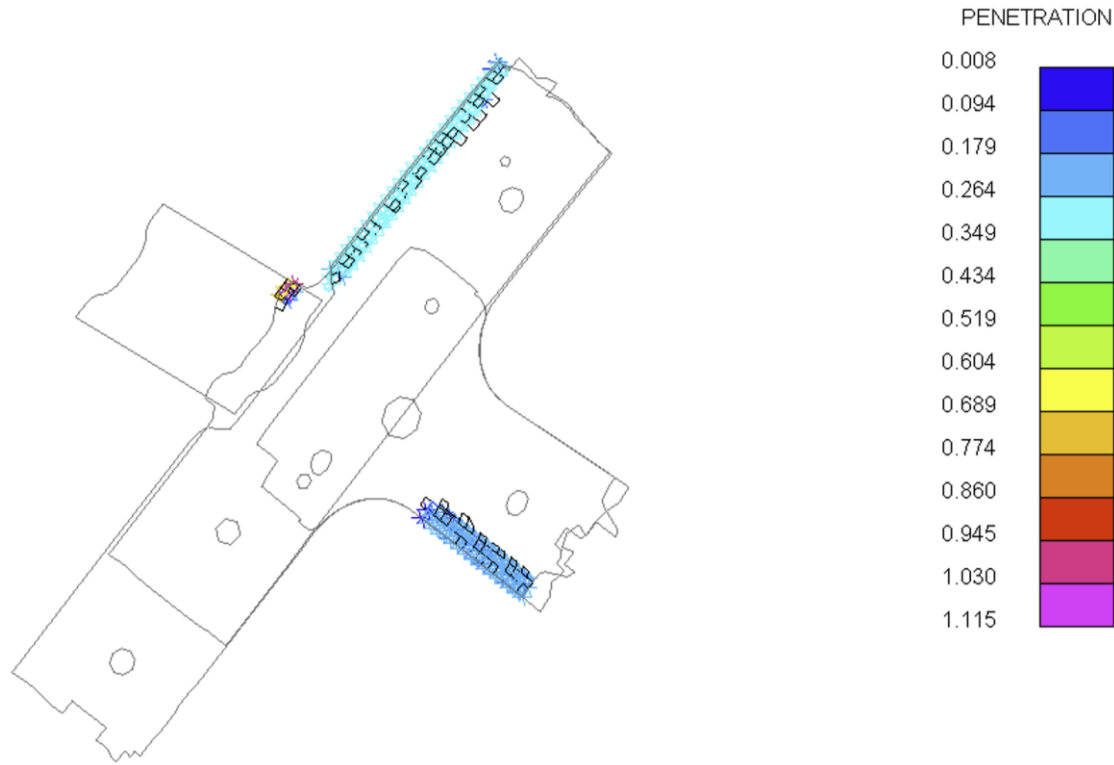
Crossed edges are shown by yellow lines on grey elements.

Penetrated elements are coloured according to penetration depth. The penetrating nodes are shown by coloured crosses, with coloured arrows showing the “escape vector” – where the node would have to move to, in order to eliminate the penetration.

The pairs of parts that interact (penetrations or crossed edges) are shown. To display just one of those pairs or parts, click on it in the menu. Restore display of all parts using “Sel All”.



Contact Penetrations - Settings



PEN CHECK M1/CONT2 (for CONTACT 2) ? [] [X]

Dismiss Check all Options...
List Info Check visible Only

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE
<No title defined>

select parts sel none sel all sel xedge ?

List crossed ☒ List penetrating: ☒

P8231:P8710 xedge=5 pen=6 (1.11482)
P8710:P82151 xedge=0 pen=108 (0.299279)
P8712:P82151 xedge=0 pen=78 (0.199826)

penetration magnitude >= 0.0

sketch unblank recursive

<- Only Pen elements Only X'd elements

--- Contour --- Pen: 192 vis 192 total
X'ed: 5 vis 5 total
Min: 8.4997e-03
Max: 1.1148e+00
Sum: 4.9104e+01

CT SI WR Sh
Settings... FIX
Levels...

->warnings.k Element details Beams on x-edge
->pencheck.c

Contact Penetrations - Settings

Penetration vectors	DRAWN		
Pen elem	Drawn	Labels	As thick
Pen node	Drawn	Labels	Value
Crossed edges	DRAWN		
X'd elems	Drawn	Labels	As thick



In the Settings panel, the different displays (penetrated elements, penetrating nodes, crossed edges) may be turned on or off, or labelled. Drawing the elements with their Contact Thickness can sometimes help to visualise why there is a penetration.

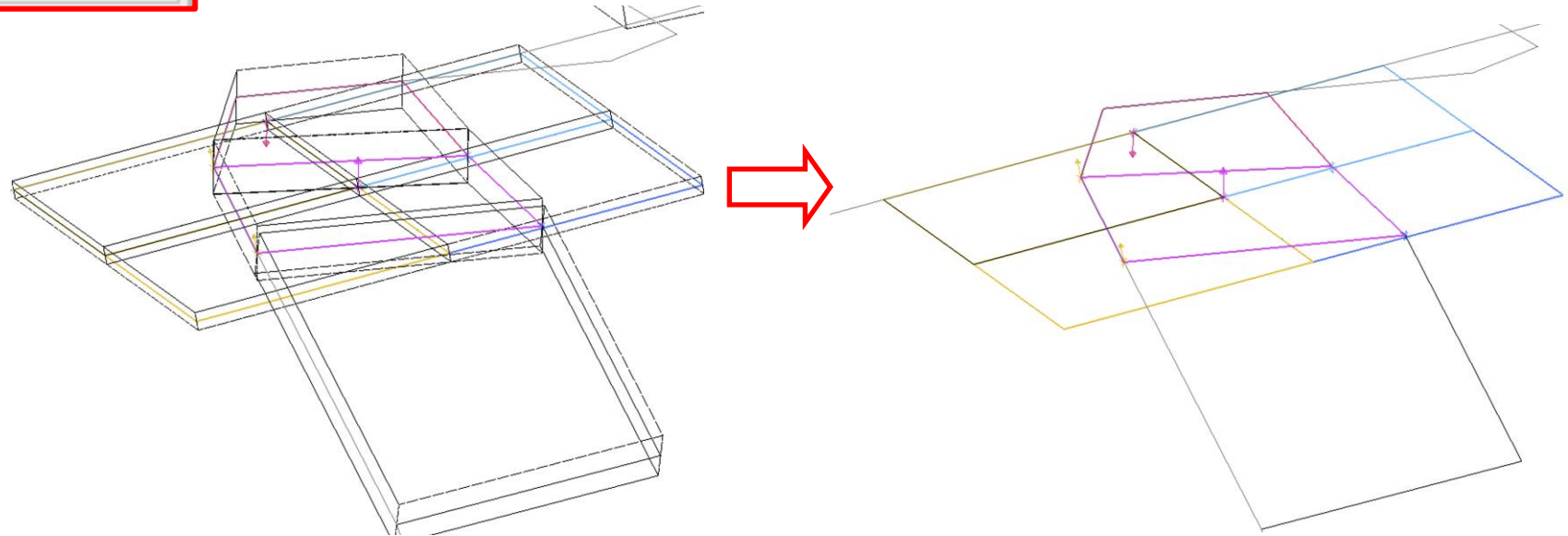
The coloured arrows show where the node would have to move to, in order to eliminate the penetration.

Contact Penetrations - Settings

Penetration vectors	DRAWN		
Pen elem	Drawn	Labels	As thick
Pen node	Drawn	Labels	Value
Crossed edges	NOT_DRA		
X'd elems	Drawn	Labels	As thick

Try turning off Crossed edges, press UPDATE.

Before proceeding to the next step, turn the crossed edges back on, and turn off "As Thick".



Contact Penetrations - Settings

penetration magnitude >=▼

0.2

sketch

unblank

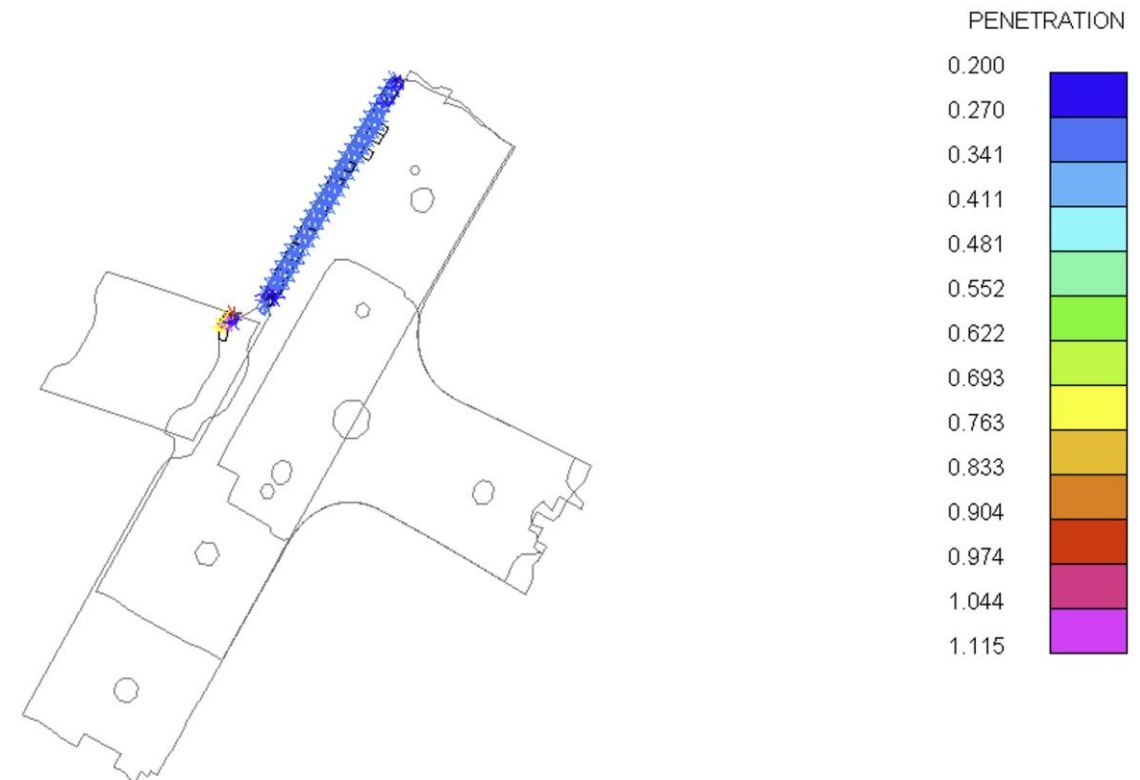
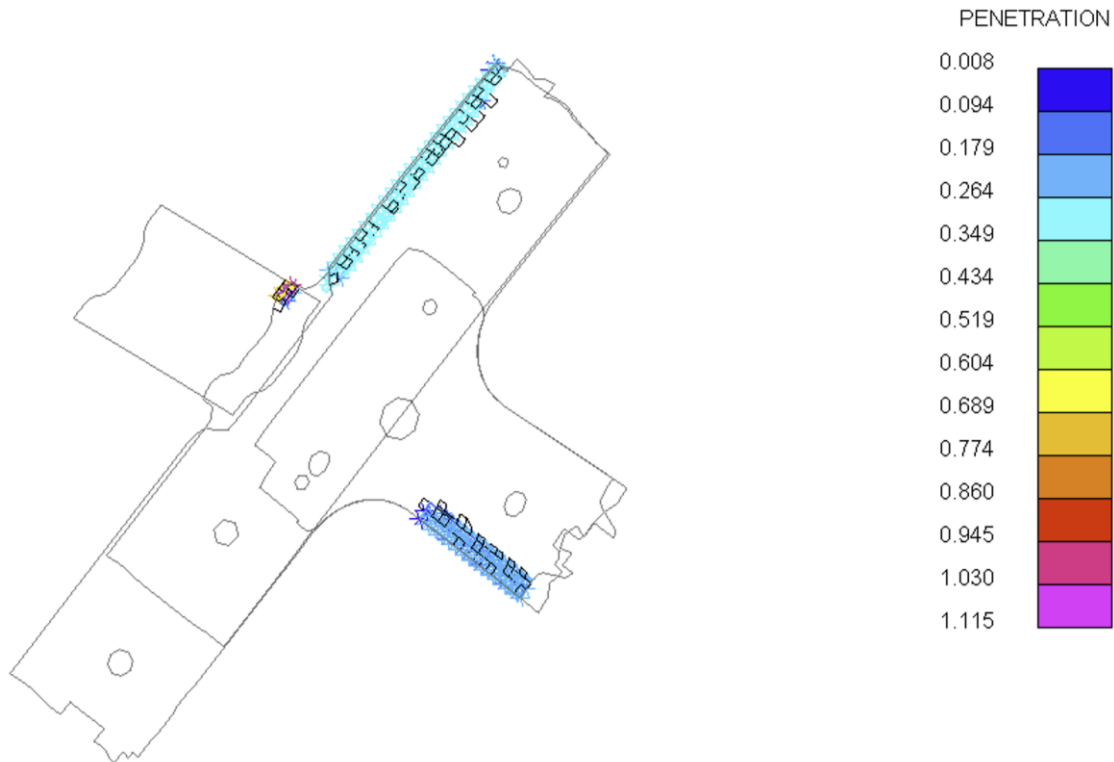
recursive

<-

Only Pen elements

Only X'd elements

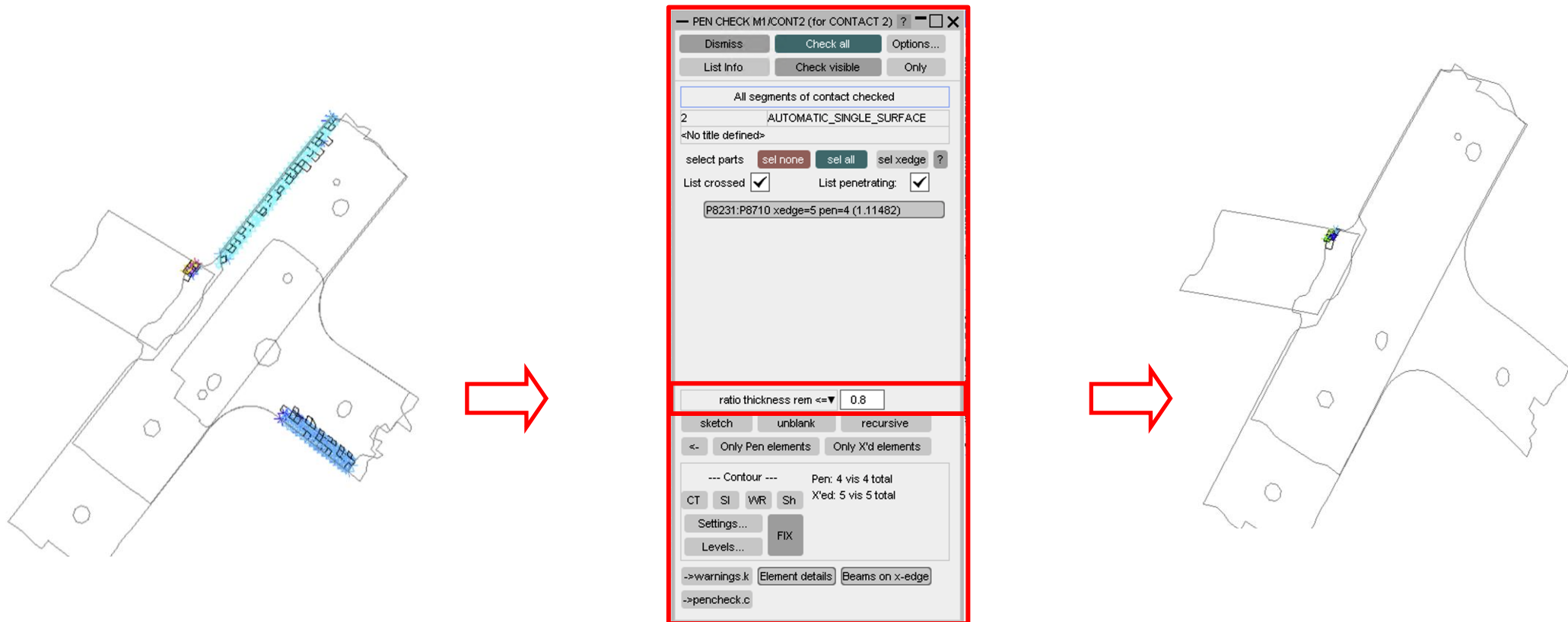
The LS-DYNA control flag IGNORE is recommended – this makes the contact algorithm tolerant to small penetrations. To find out whether any penetrations exceed a certain tolerance – say 0.2mm – input a minimum value in the Pen Check Panel. Update Plot. PRIMER now displays only penetrations greater than 0.2mm



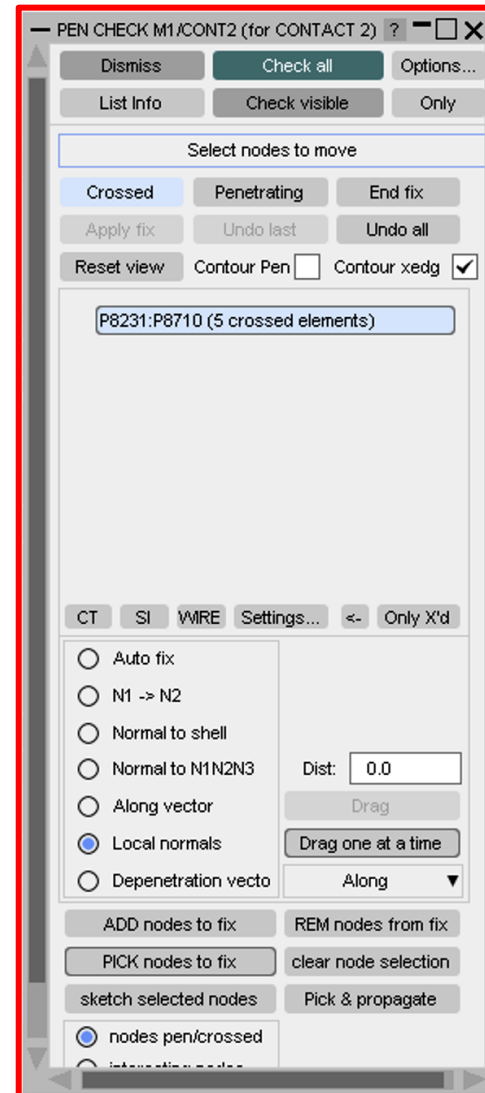
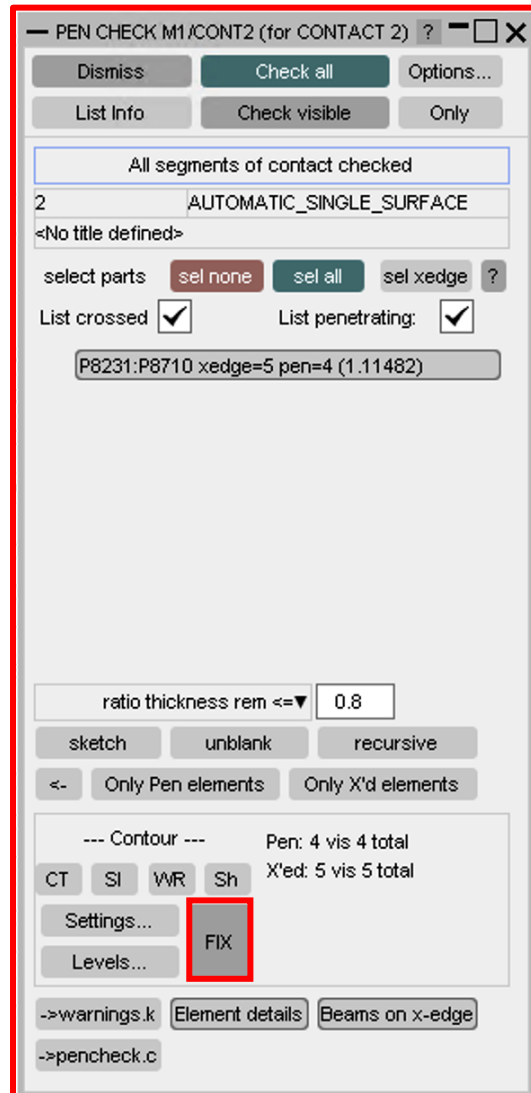
Contact Penetration Ratio

To find out whether any penetrations exceed, say, 0.2 times the shell thickness, set the data component to “ratio thickness remaining”, and the value to 0.8 (i.e. PRIMER will display colours only where the remaining thickness, after the penetration is subtracted, is 0.8 or less times the thickness. Press SI again to display.

Before proceeding to the next step, set the data component back to Penetration, and the limit value to 0.2.



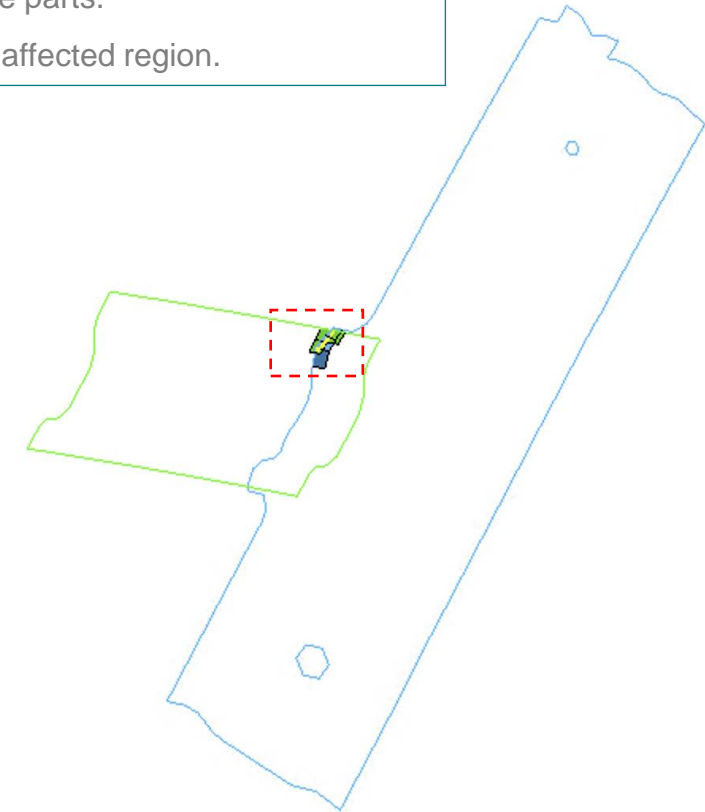
Cross-edges Fixing



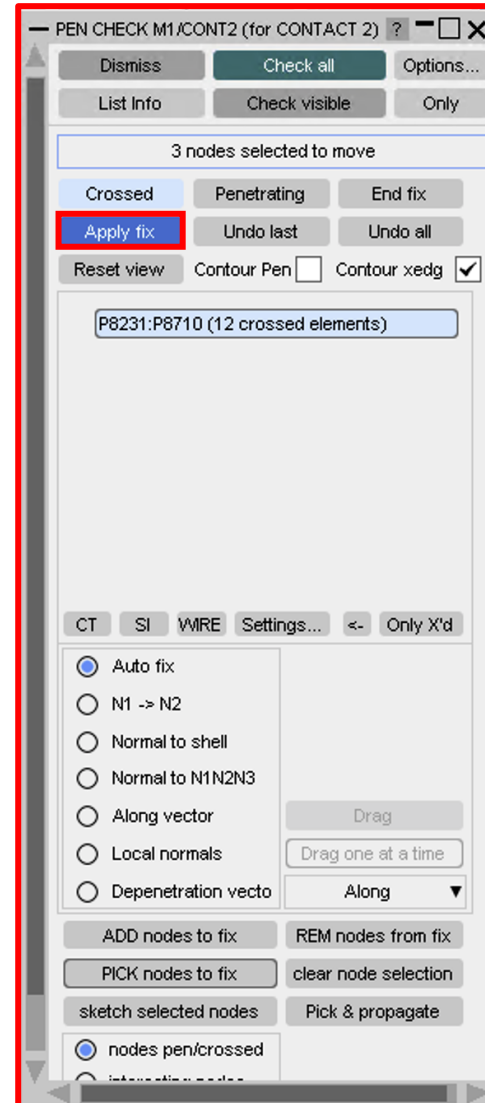
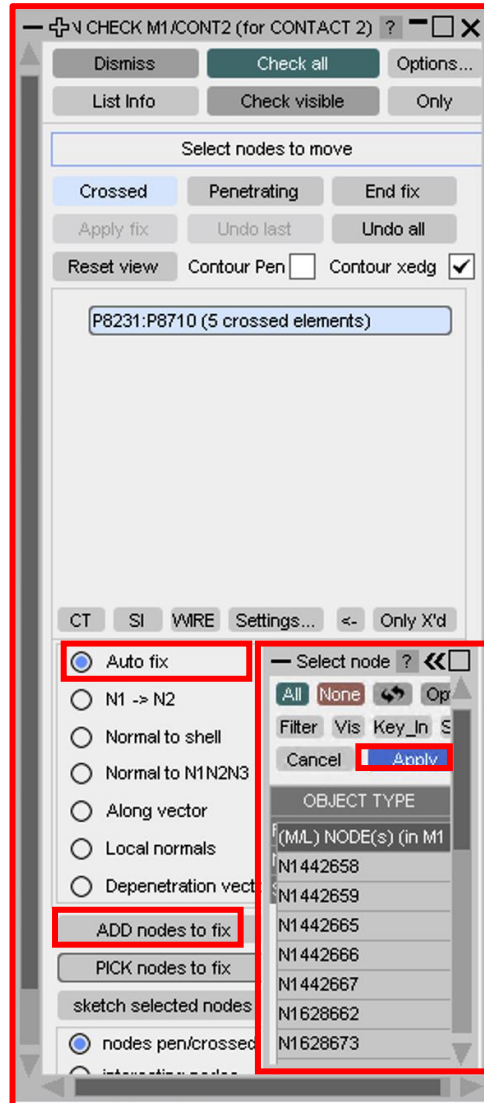
Press FIX. We recommend to fix the crossed edges first.

PRIMER offers the pairs of parts that are intersecting. Click on the only pair. PRIMER shows just these parts.

Zoom in on the affected region.

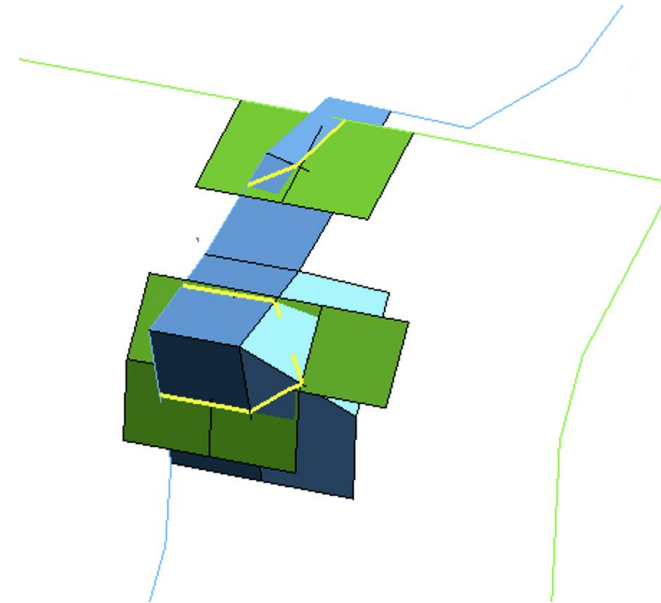


Cross-edges Fixing

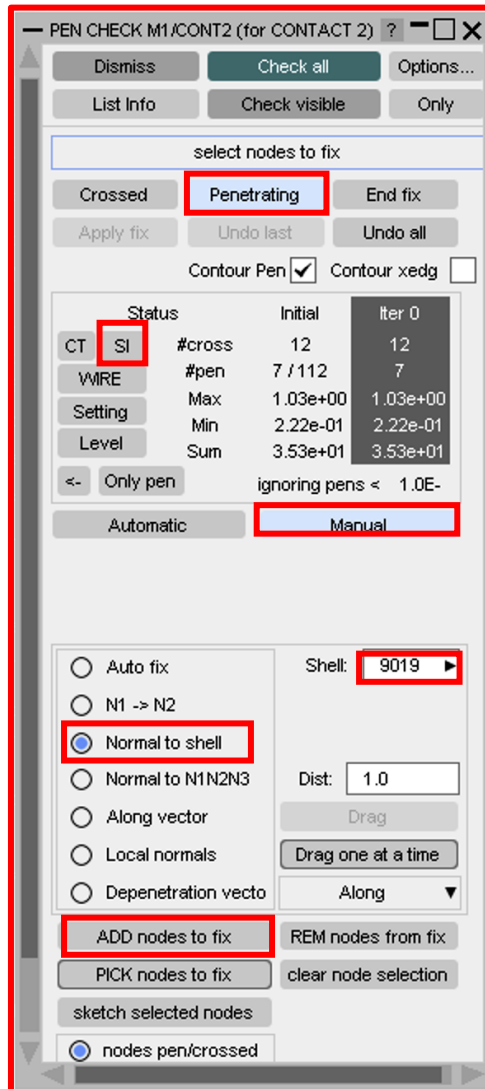


We must tell PRIMER which nodes are on the “wrong” side. Use “ADD nodes to fix” to select the three nodes shown, APPLY to confirm the node selection.

There is a choice of manual (e.g. dragging) or automatic methods to calculate the movement of each node. Try the Auto fix method. Press “Apply fix”.



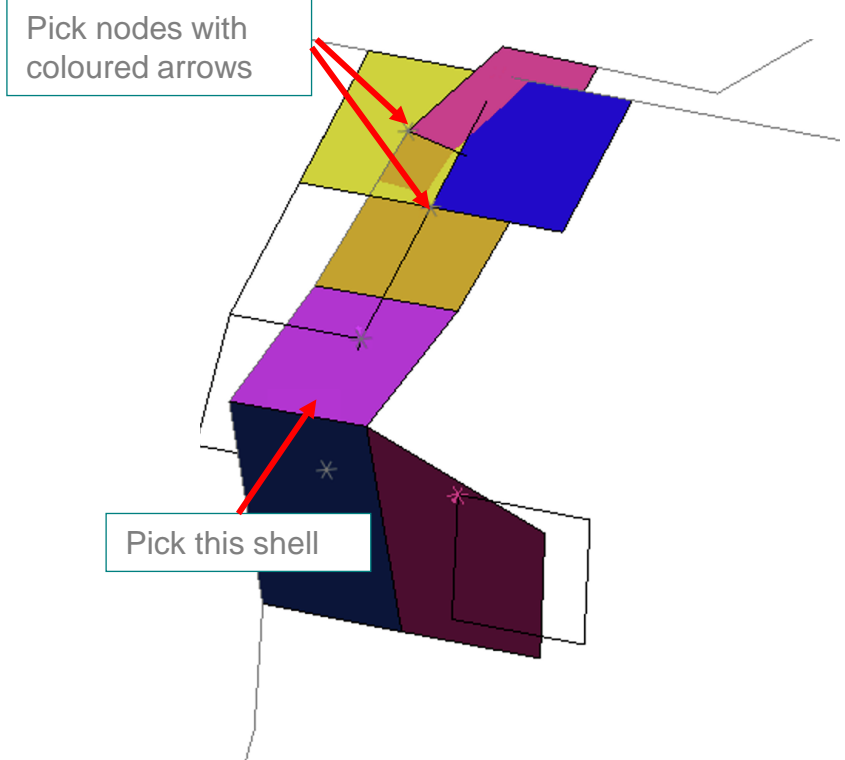
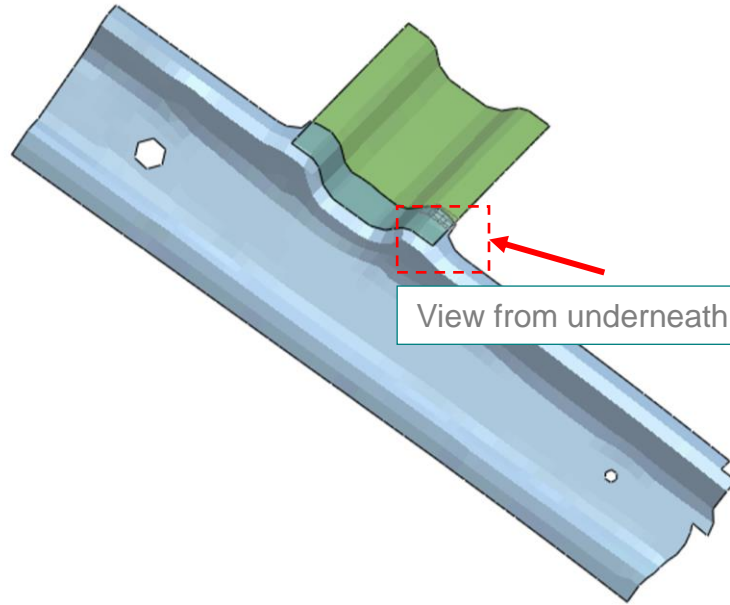
Penetrations – Manual Fixing



Switch to Penetrating, ensure the menu is switched to Manual, press SI.

Although we have fixed the crossed edges, the same two parts are still penetrating. We will now fix these penetrations manually.

Use “ADD nodes to fix” to select the nodes to be dragged. Try dragging using “normal to shell”, until these penetrations disappear.



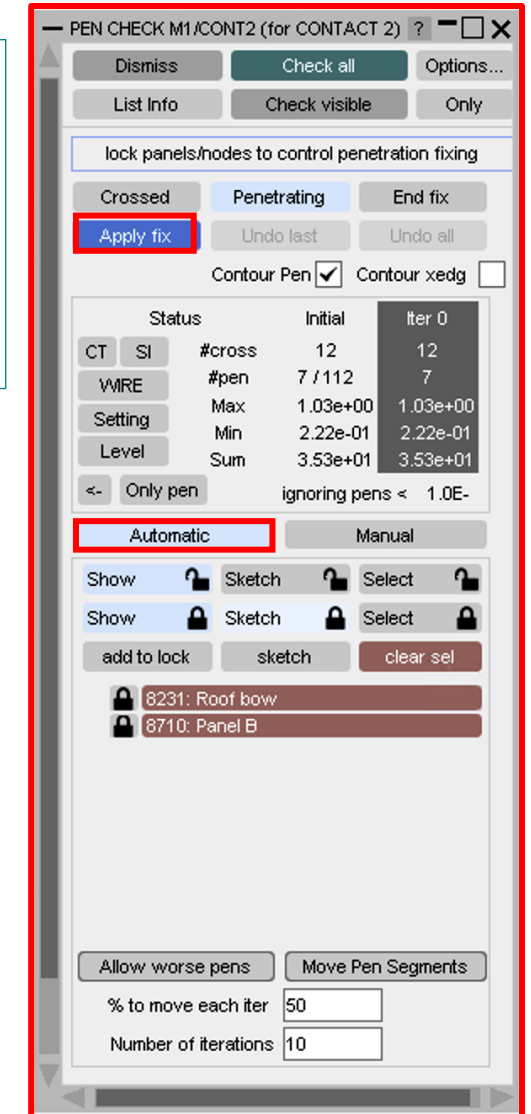
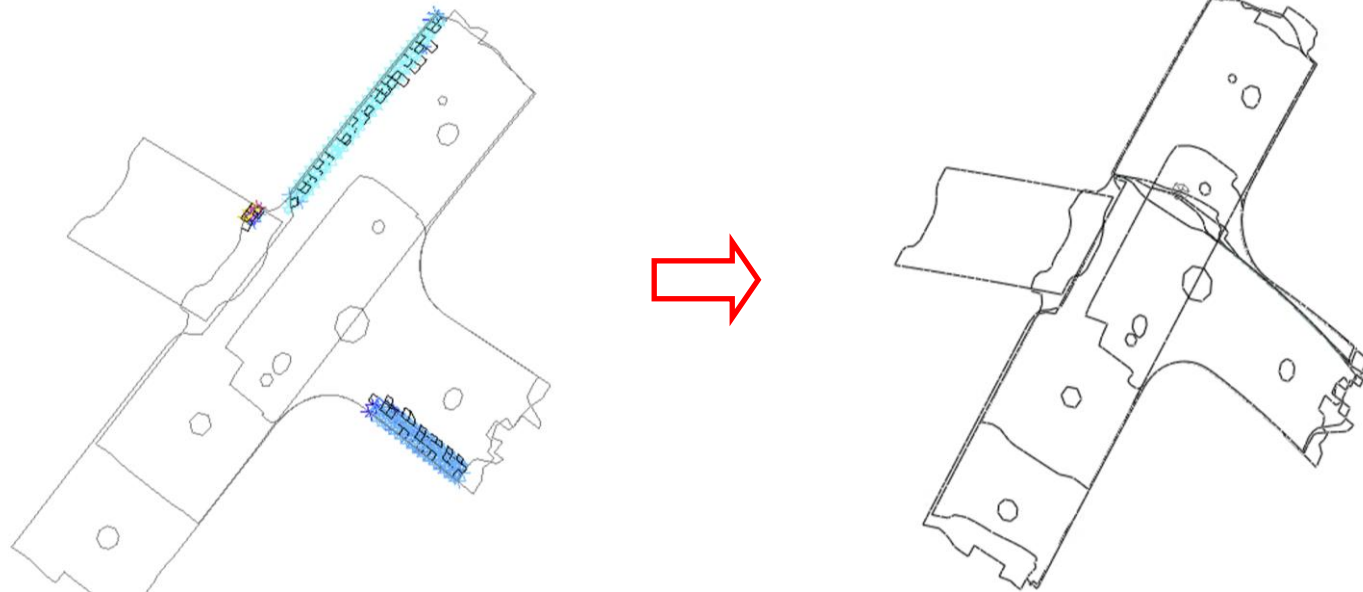
Penetrations – Automatic Fixings

Unblank the whole model (shortcut U). We will fix the remaining penetrations using the Automatic method.

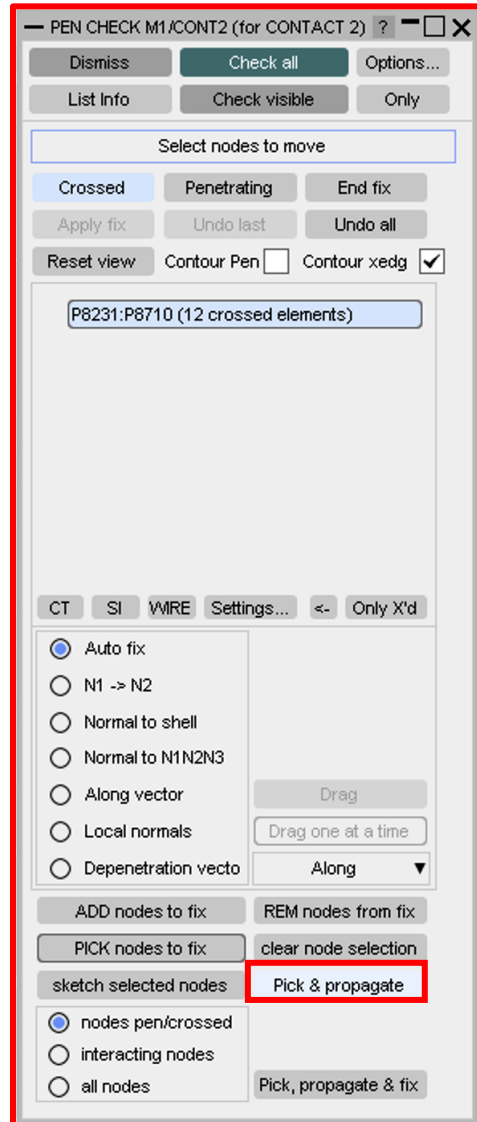
Switch to Automatic Fixing. Press Apply.

PRIMER moves the penetrating nodes iteratively to reduce the penetrations. In this model, all the penetrations are fixed.

In more complex models, some penetrations may remain; these need to be fixed manually. Usually the reasons are either (a) crossed edges have not been fixed or (b) since PRIMER moves only the penetrating nodes (shown with coloured arrows), not the penetrated elements, this is not always enough to resolve complex multi-layer situations.

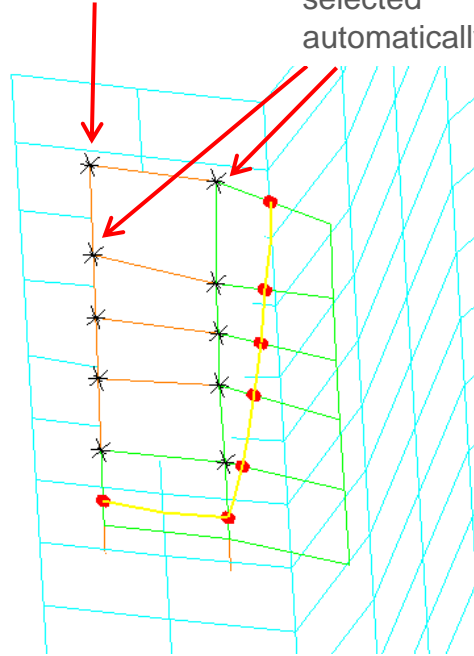


Penetrations: Crossed Edge Fixings

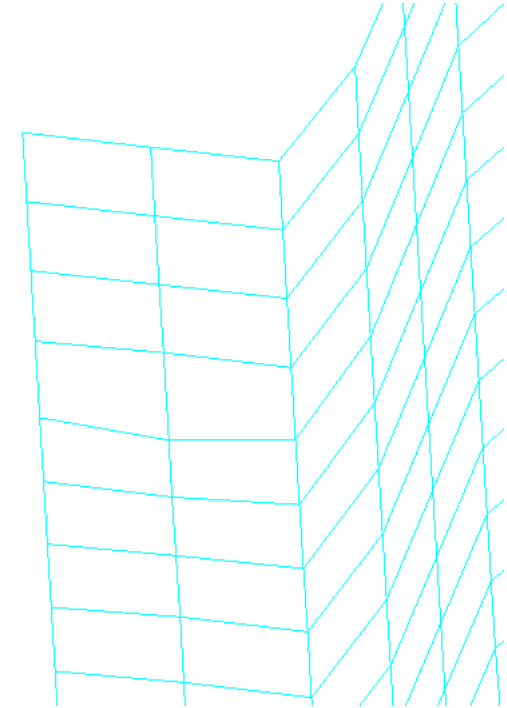


Pick one node

Other nodes are
selected
automatically

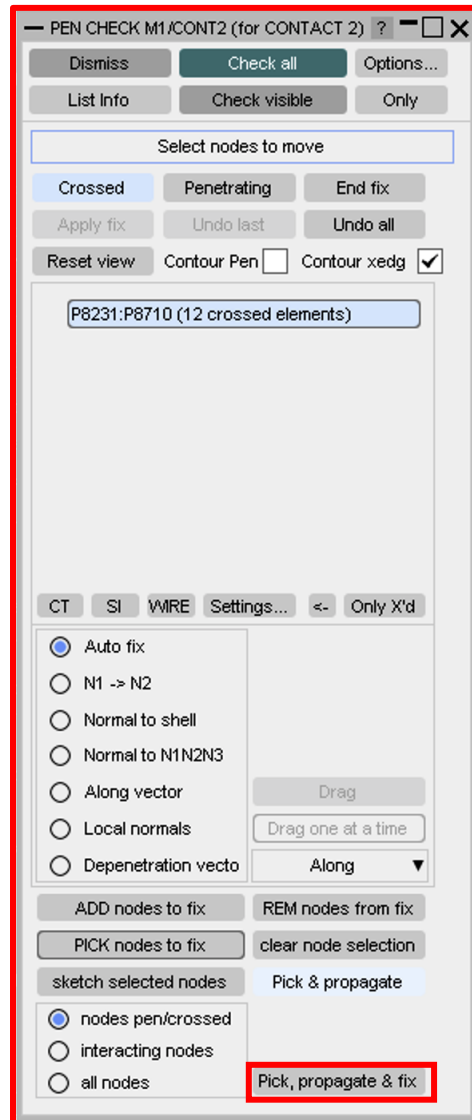


Try this on crossed_edge.key

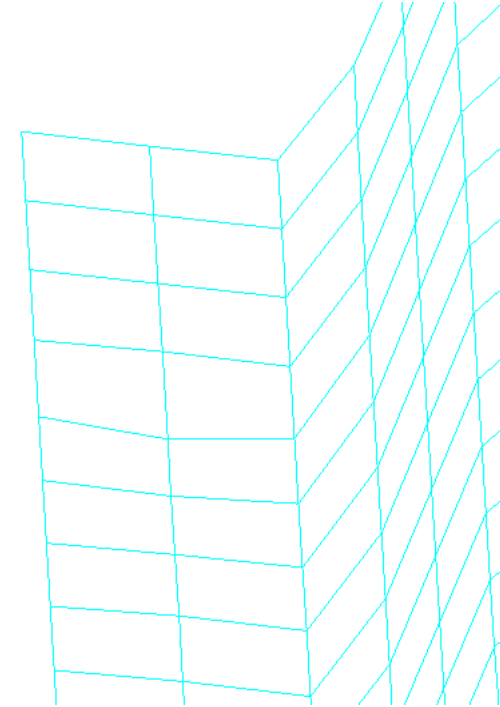
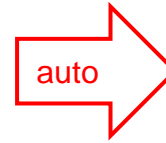
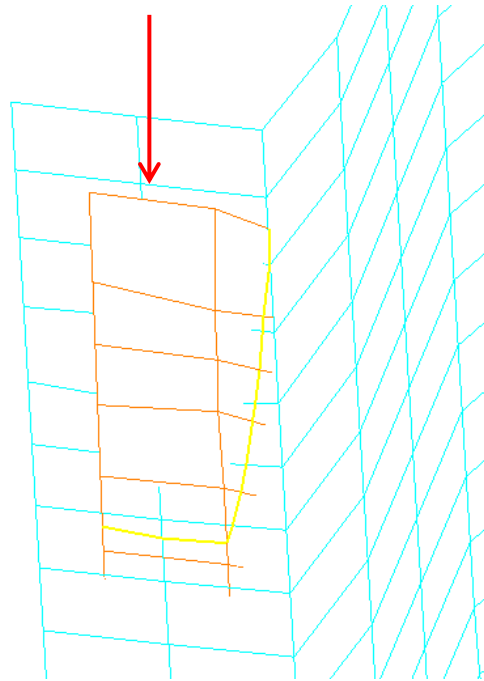


“Pick & propagate” allows user to pick one node; PRIMER then automatically selects all the other nodes that are on the same side of the Crossed Edge. Selection can be adjusted using ADD, REM
Fixing the selected nodes is a separate operation. The usual methods are available, e.g. “Auto fix”, “Normal to shell”, etc

Penetrations: Crossed Edge Fixings



Pick one node



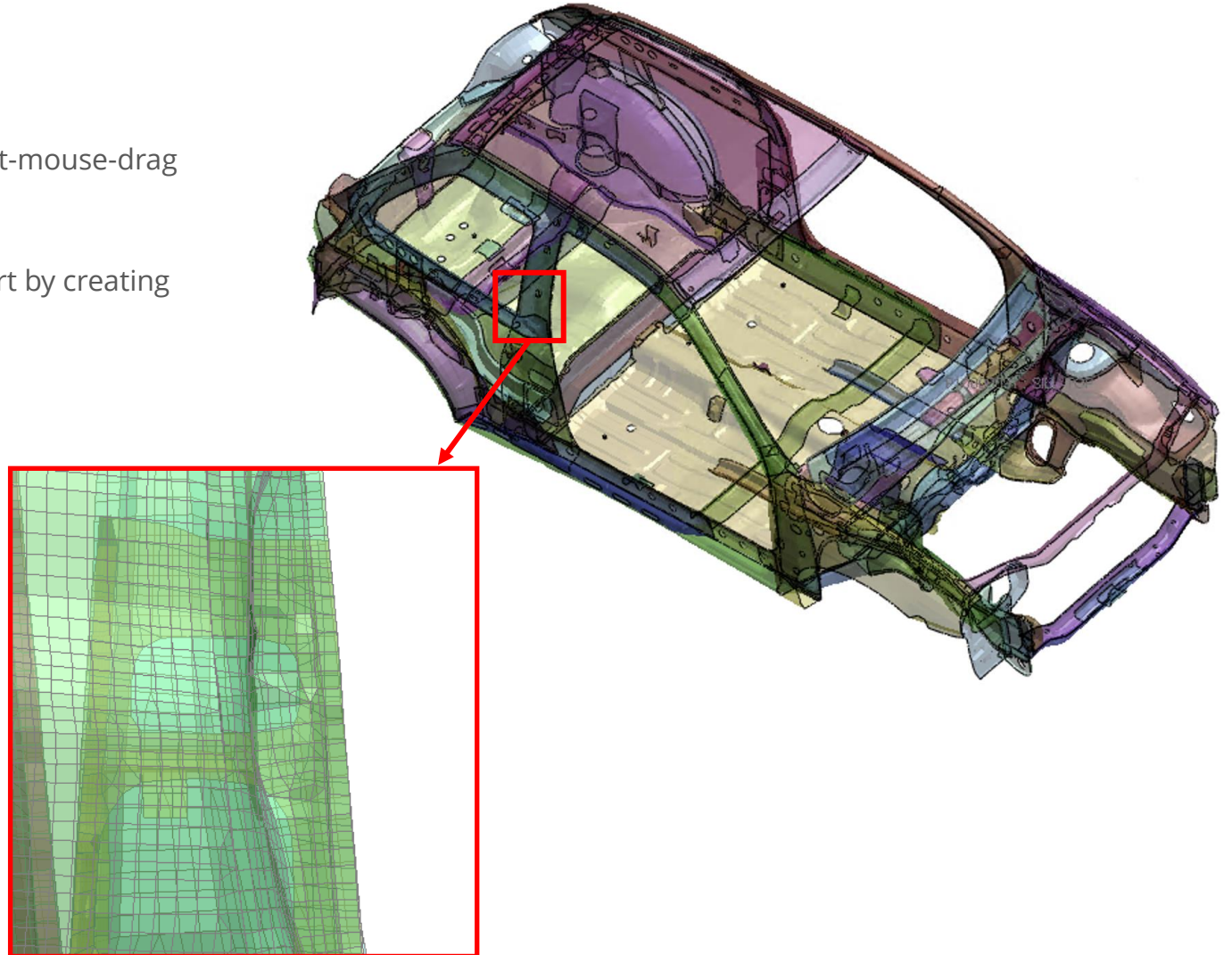
“Pick, propagate & fix” allows user to pick one node; the selection of other nodes that are on the same side of the Crossed Edge, and the fixing, are both performed automatically in a single operation

Spotwelding

This section of the course is a reduced version of our separate Oasys PRIMER training course, “Spotwelds and Connections”

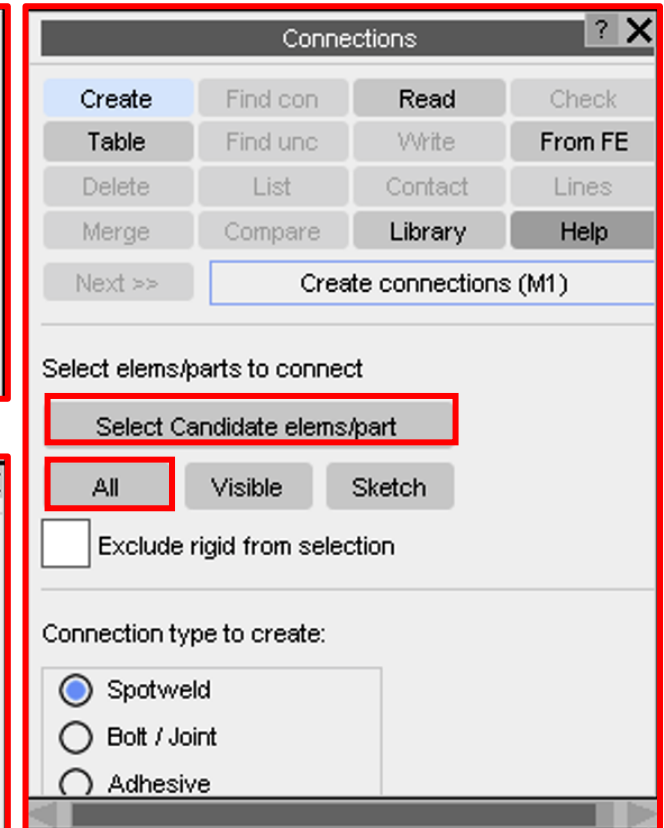
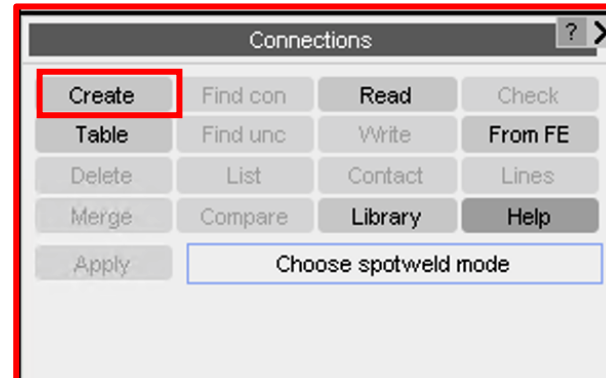
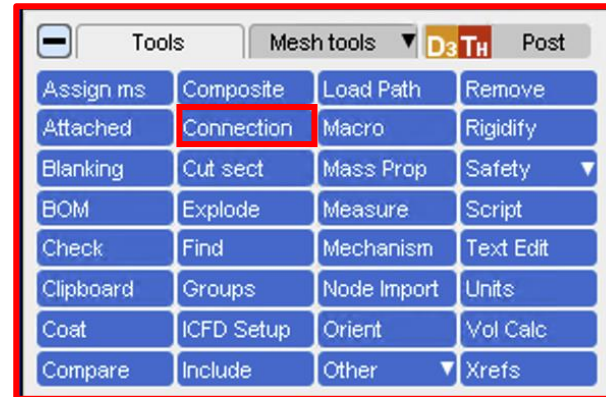
Creating Spotwelds

- Delete all models from PRIMER
- Read in biw_for_welding.key
- Make all the body panels 50% transparent (e.g. right-mouse-drag across=>transparency=>50%)
- Zoom in on the right-hand lower B-post: we will start by creating spotwelds on the forward flange.
- Switch on the element borders (shortcut Y)



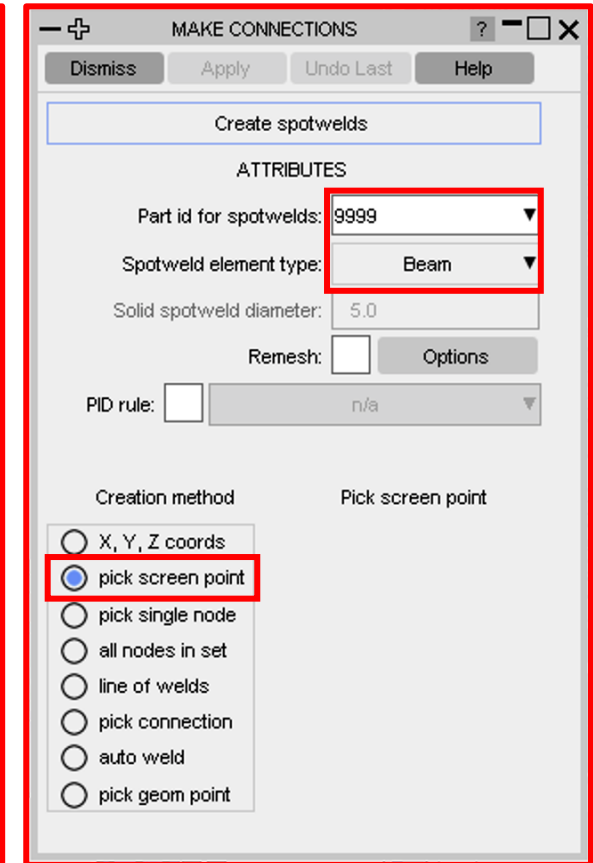
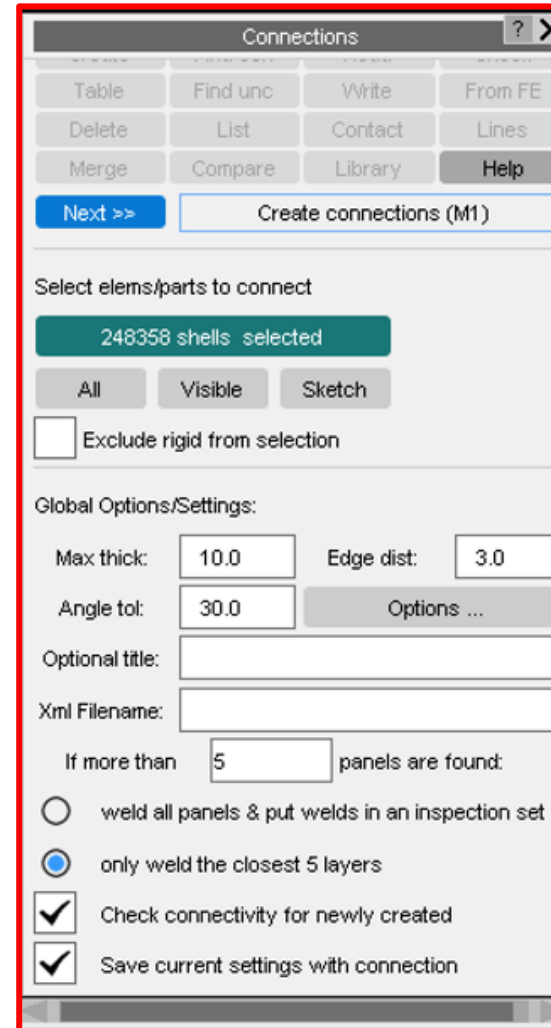
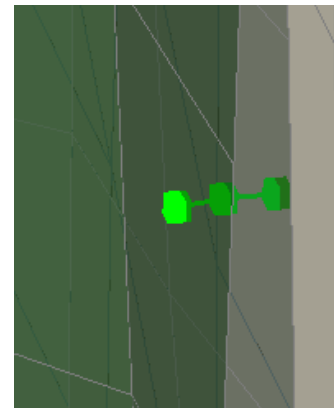
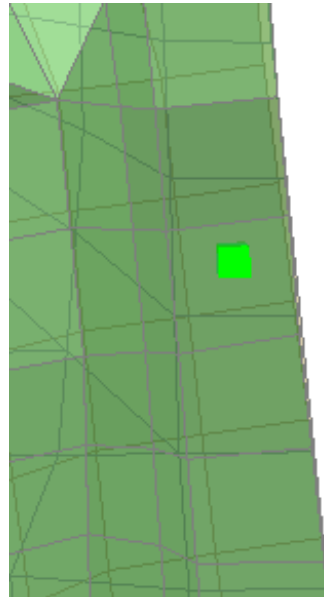
Creating Spotwelds

- Tools=>Connection=>Create
- The Connections menu is used for spotwelds and other connection types
- First we have to tell PRIMER which panels can be considered for welding. This model consists only of body panels so we can just select all the shell-element parts in the model.
- Press “Select Candidate elems/part” , All, Apply Selection. Or, just press “All”.



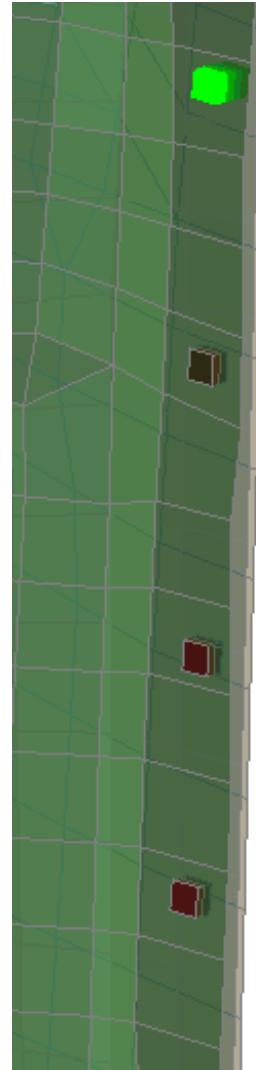
Creating Spotwelds

- Leave the spotweld type set to Beam.
- This model already contains an empty Part ID for the beam spotwelds (ELFORM=9, *MAT_SPOTWELD). PRIMER automatically finds this and uses it. If this part were not present in the model, you would have to create one first before creating the spotwelds.
- Try the “pick screen point” option
- Click on a point on the forward flange of the lower B-post.
- A green spotweld appears.
- Rotate the view: PRIMER found 3 layers to be welded, so it created a chain of 2 beams.



Creating Spotwelds

- Switch the element type to Hexa.
- Again, a suitable part is already present in the model so PRIMER uses it.
- Click on the flange in several places: a solid spotweld appears in each place.
- Also try 4-Hexa.



MAKE CONNECTIONS

Dismiss Apply Undo Last Help

Create spotwelds

ATTRIBUTES

Part id for spotwelds: 9998

Spotweld element type: Hexa

Solid spotweld diameter: 5.0

Remesh: ☐ Options

PID rule: ☐ n/a

Creation method

Pick screen point

☐ X, Y, Z coords

☒ pick screen point

☐ pick single node

☐ all nodes in set

☐ line of welds

☐ pick connection

☐ auto weld

☐ pick geom point

Autoweld

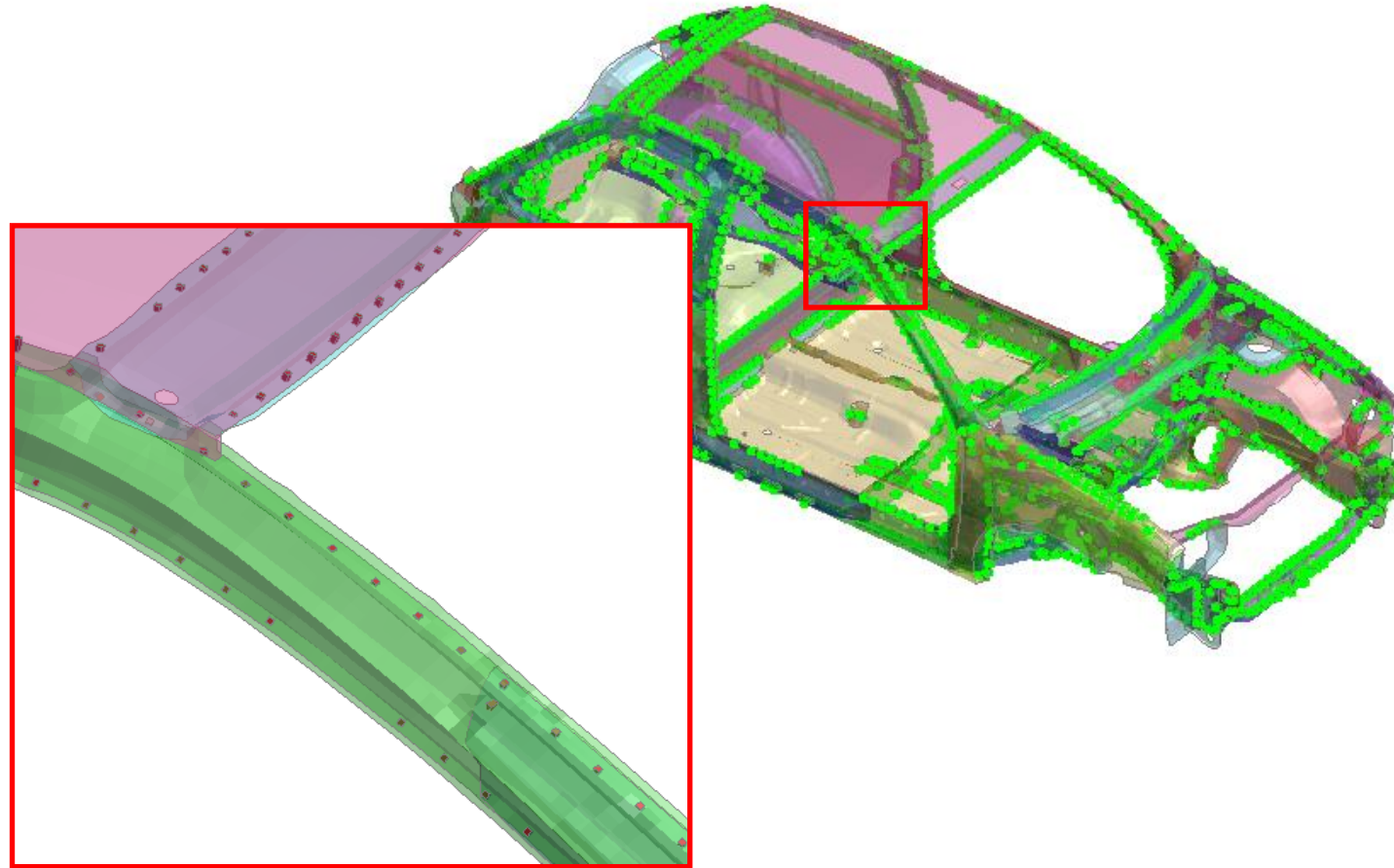
- Switch the creation method to Autoweld. With this option, PRIMER tries to create welds at a user-defined pitch, using the free edges of the panels as a guide.
- This option is used when no information is available about the actual locations of the spotwelds.
- Set the “min run length” to 50, i.e. PRIMER will try to spotweld any flange longer than 50mm.
- Check that the weld pitch is 40mm.
- Press Apply.

The screenshot shows the 'MAKE CONNECTIONS' dialog box with the following settings:

- Create spotwelds** (button)
- ATTRIBUTES**
 - Part id for spotwelds: 9998
 - Spotweld element type: Hexa
 - Solid spotweld diameter: 5.0
 - Remesh: ☐ Options
 - PID rule: ☐ n/a
- Creation method**
 - ☒ auto weld (highlighted with a red box)
 - ☐ X, Y, Z coords
 - ☐ pick screen point
 - ☐ pick single node
 - ☐ all nodes in set
 - ☐ line of welds
 - ☐ pick connection
 - ☐ pick geom point
- Weld parameters**
 - Sketch weld positions**
 - Min run length: 50.0 (highlighted with a red box)
 - Min sub length: 50.0 (highlighted with a red box)
 - Sub break angle: 45.0
 - Pitch: 40.0 (highlighted with a red box)
 - Weld edge dist: 10.0
 - ☒ Master Part
 - ☐ Master Part S
 - <none>

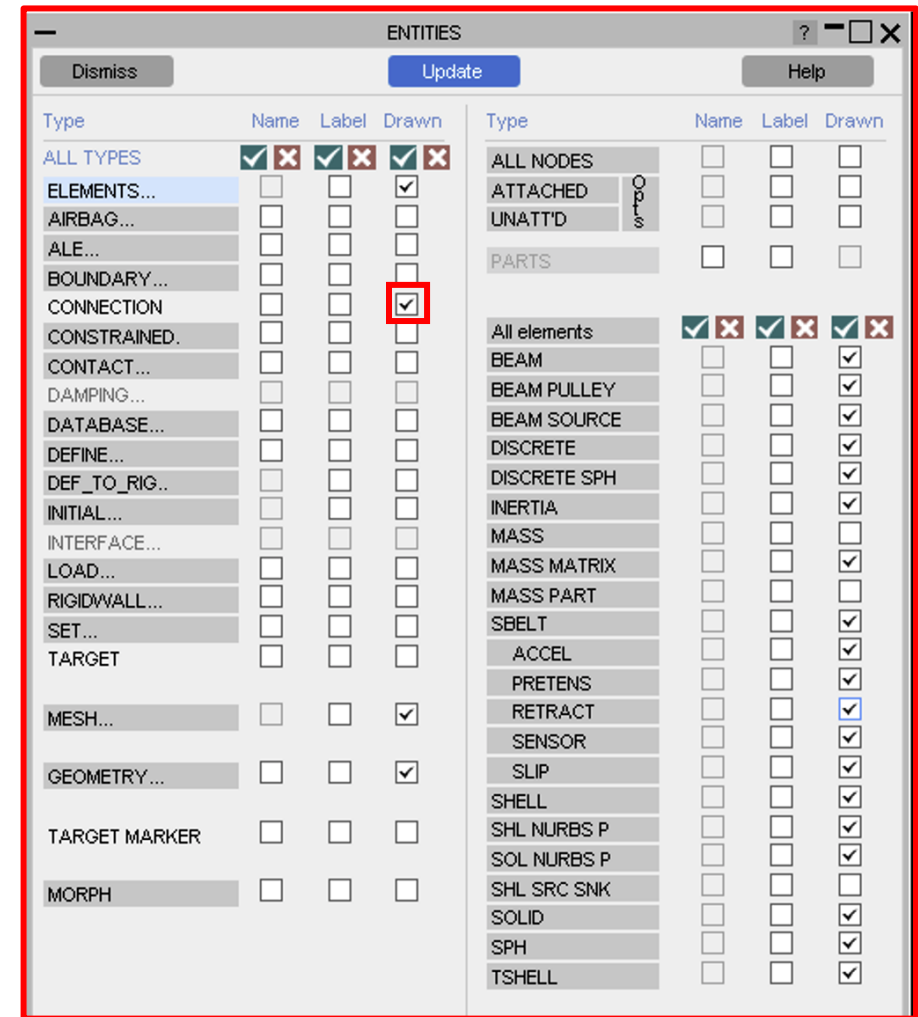
Autoweld

- There will be places where PRIMER has not created welds (e.g. where the panel gaps are too large), so the model still needs to be checked carefully when using this creation method!



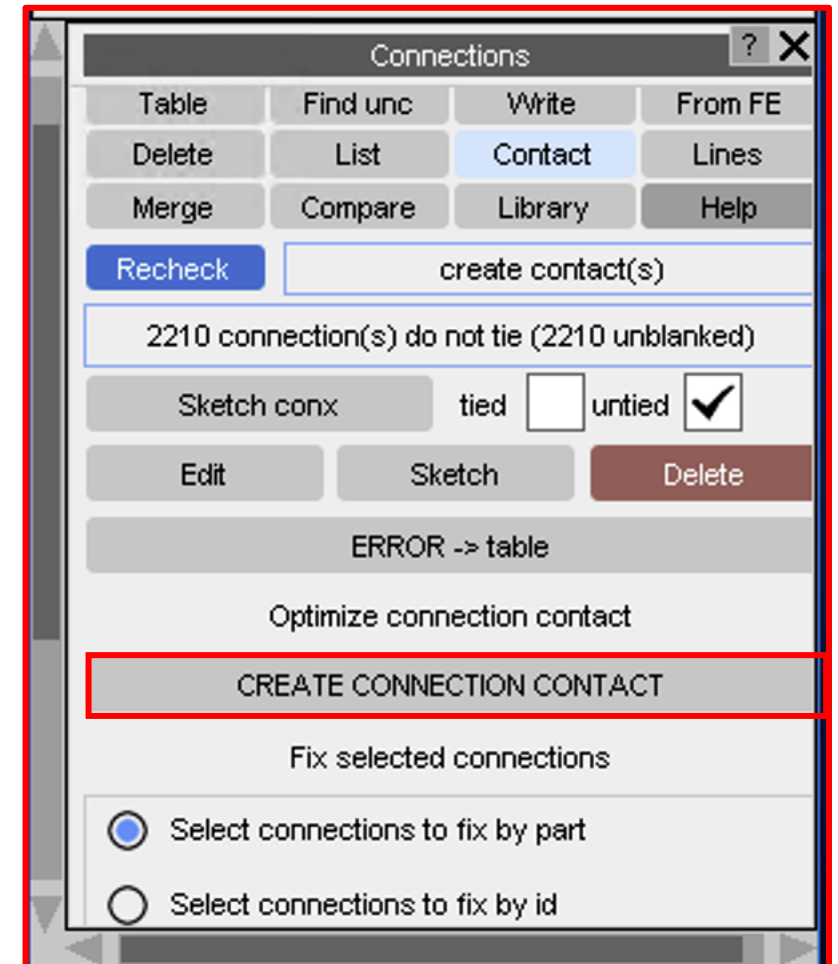
Creating Spotwelds

- When creating spotwelds, PRIMER also creates connection entities. These are the green or red circles, coloured according to status.
- If you wish, visibility of these can be switched off using the Entities panel.
- This entity visibility switch will automatically be switched on whenever new spotwelds are created.



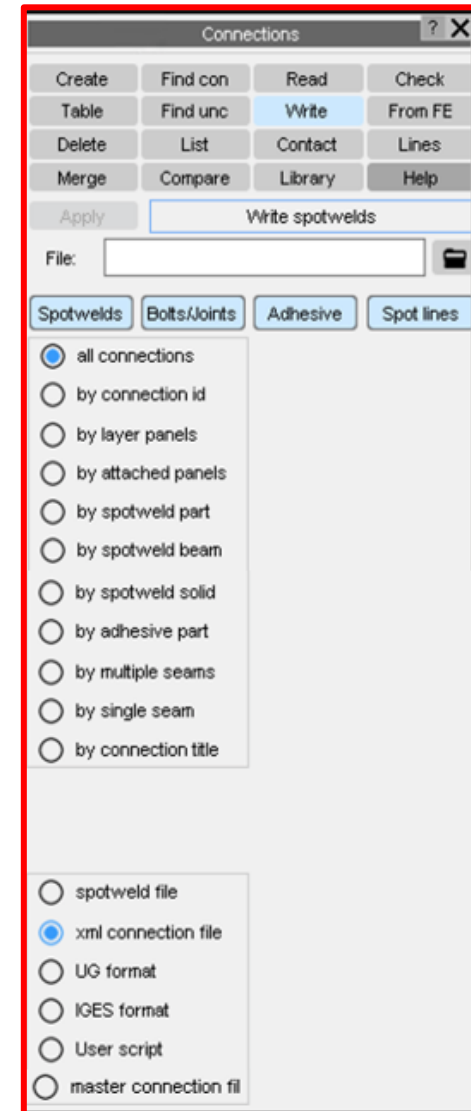
Adding a Tied Contact

- After creating spotwelds, we need to create a tied contact to stick them to the panels.
- Press Contact, Create connection contact, Two contacts.
- PRIMER creates a *CONTACT_TIED_SURFACE_TO_SURFACE for solid elements, or a *CONTACT_SPOTWELD for beam elements. It will also create parallel penalty based contacts if any rigid constraint clash is detected.
- In this model, the body structure Part IDs are in the range 1,000,000 to 1,000,999. We recommend to change the *SET_PART of the master side of the tied contact to *SET_PART_GENERATE such that any body panels added later will automatically be included in the tied contact.



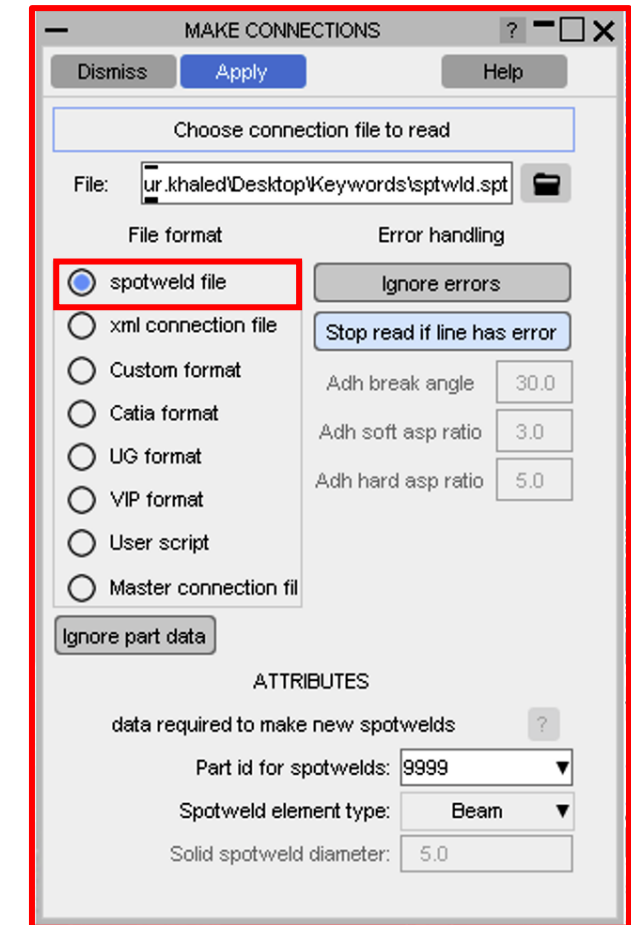
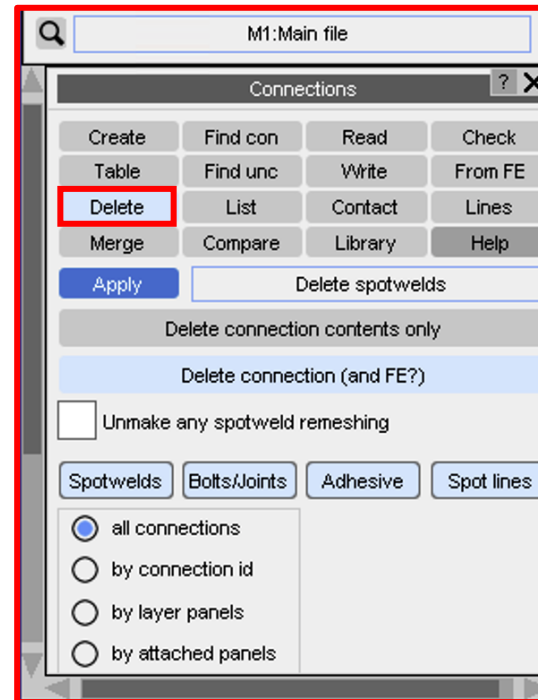
Writing a Spotweld File

- Dismiss the Spotweld Create menu (e.g. press ESC while the mouse is in that menu)
- To save the newly created spotwelds, we should either write out a keyword file, or write a spotweld file. The spotweld file contains the coordinates of each spotweld, and which panels are connected. The same spotwelds could be re-created in another similar model from the spotweld file.
- “spotweld file” = simple csv format, one line per weld.
- “PRIMER connection file” = xml-format, contains more information e.g. whether the welds are beam or hexa.
- Unigraphics (UG) format weld file is also available.
- We would recommend to choose a PRIMER Connection file, to avoid losing any information.
- Write a connections file named example1.xml.



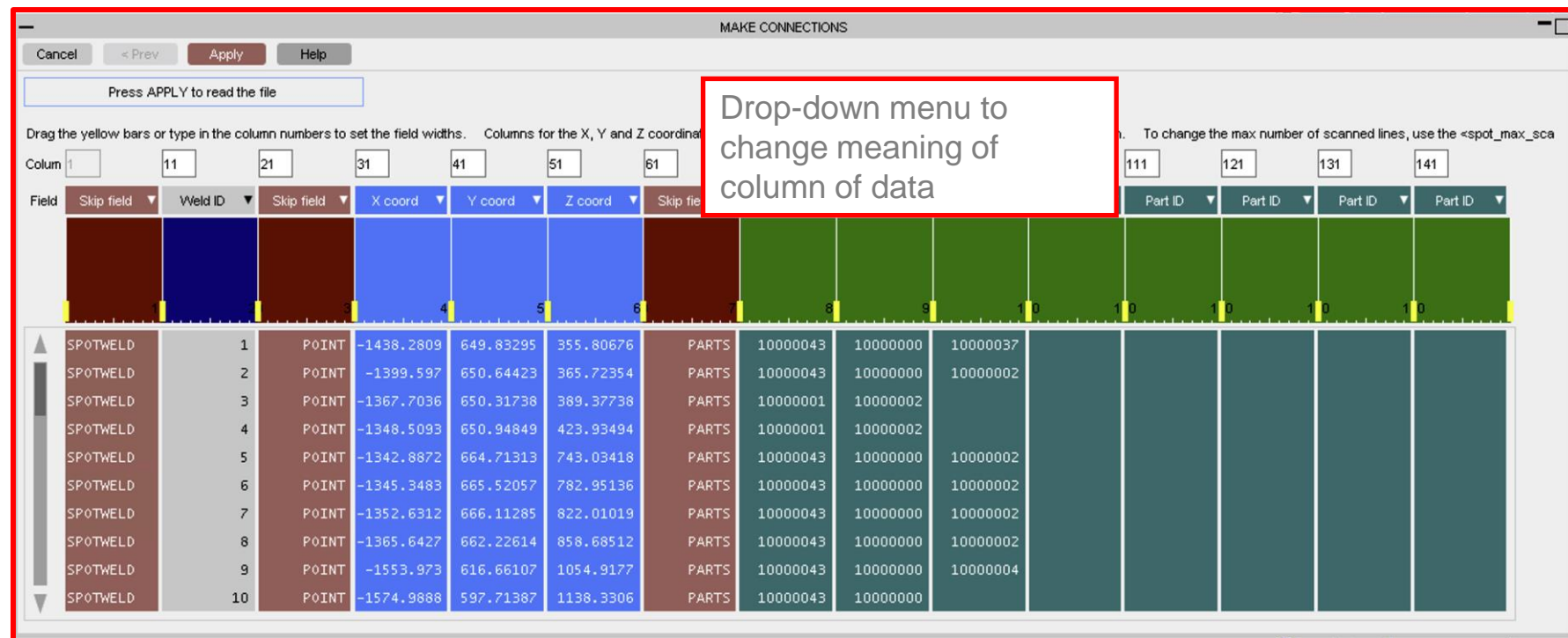
Reading a Spotweld File

- Before reading the spotweld file, we will delete all the existing spotwelds.
- Press Delete, leave the selector on “All connections”, Apply, “Delete conx and FE”.
- Press Read, select file format as “spotweld file” and browse to find the file sptwld.spt
- The file format is PRIMER Spotweld File.
- Press Apply



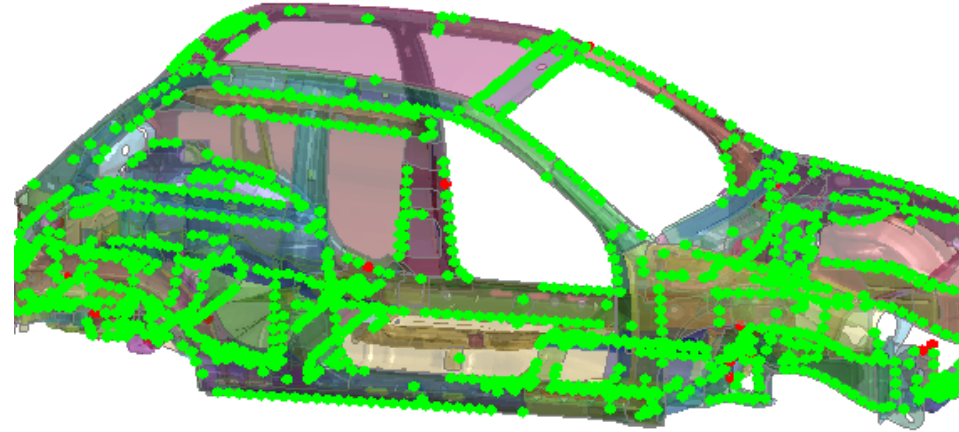
Reading a Spotweld File

- An image of the file contents appears
- “PRIMER spotweld file” format can be used for files written by other software – the meaning of each column may be controlled from the drop-down menus. The limitation is that the file must have only one line per spotweld.
- Move the menu so you can see the vehicle.
- Press Apply.



Reading a Spotweld File

- PRIMER creates the spotwelds (green symbols) where possible. Bad welds are shown in red.
- A message appears stating how many spotwelds were successfully created, and how many were not. Press Continue.
- Most spotweld files contain some errors. Fixing the errors can be a time-consuming task. PRIMER automatically puts the bad welds into the Connection Table.
- We will now use the table to fix some of the errors.



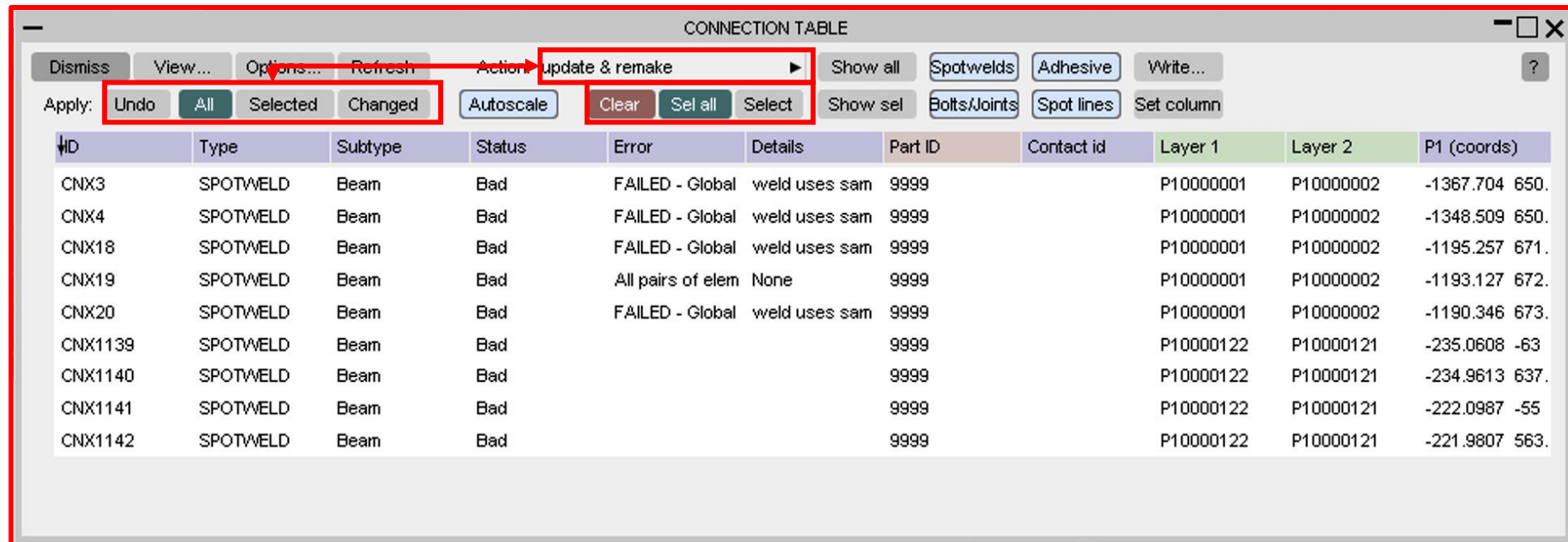
CONNECTION TABLE										
Dismiss		View...	Options...	Refresh	Action: update & remake		Show all	Spotwelds	Adhesive	Write...
Apply: Undo		All	Selected	Changed	Autoscale	Clear	Sel all	Select	Show sel	Boils/Joints
									Spot lines	Set column
ID	Type	Subtype	Status	Error	Details	Part ID	Contact id	Layer 1	Layer 2	P1 (coords)
CNX3	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1367.704 650.
CNX4	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1348.509 650.
CNX18	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1195.257 671.
CNX19	SPOTWELD	Beam	Bad	All pairs of elem	None	9999		P10000001	P10000002	-1193.127 672.
CNX20	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1190.346 673.
CNX1139	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-235.0608 -63
CNX1140	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-234.9613 637.
CNX1141	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-222.0987 -55
CNX1142	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-221.9807 563.

The Connection Table

- Try hovering over the Error and Details columns.
- Try sorting the table by clicking on the column headers.
- Try using the View button to change the contents of the table.
- Look at the options under “Action” – the most commonly used of these are also available by right-click on selected welds.

Make the action happen using the Apply buttons, or
by right-click on welds selected in the table

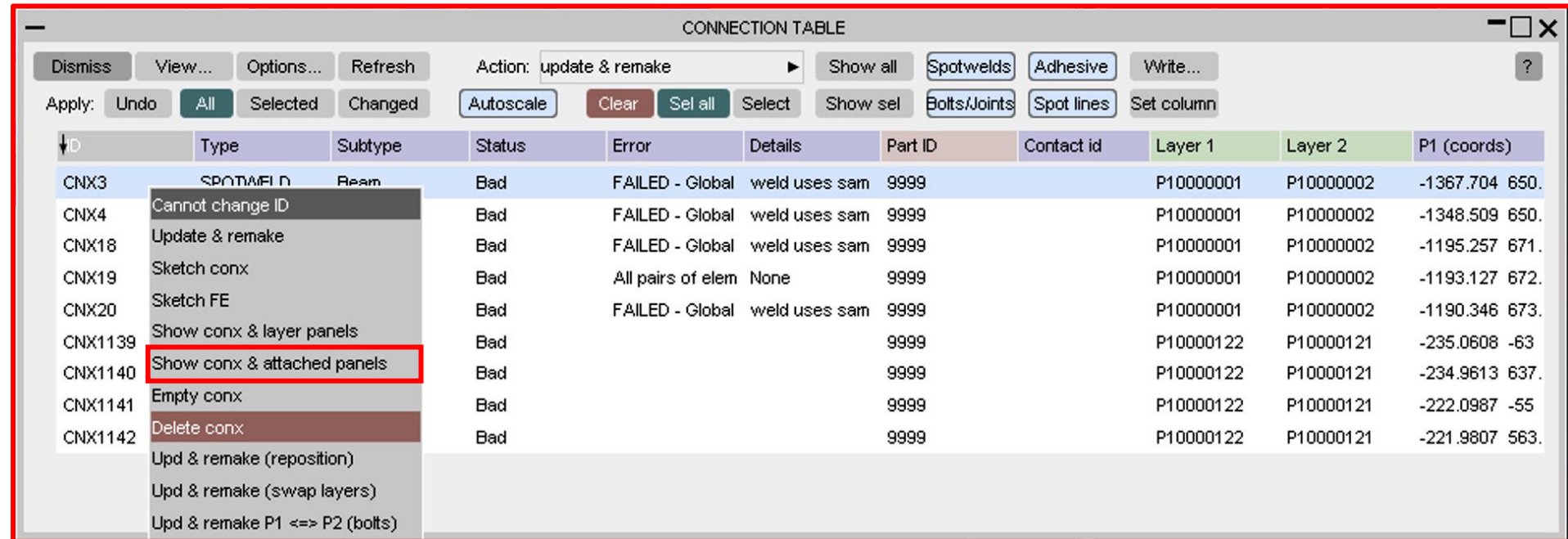
Select welds here, or by clicking
in the table



ID	Type	Subtype	Status	Error	Details	Part ID	Contact id	Layer 1	Layer 2	P1 (coords)
CNX3	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1367.704 650.
CNX4	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1348.509 650.
CNX18	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1195.257 671.
CNX19	SPOTWELD	Beam	Bad	All pairs of elem	None	9999		P10000001	P10000002	-1193.127 672.
CNX20	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1190.346 673.
CNX1139	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-235.0608 -63.
CNX1140	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-234.9613 637.
CNX1141	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-222.0987 -55.
CNX1142	SPOTWELD	Beam	Bad			9999		P10000122	P10000121	-221.9807 563.

Using the Connection Table to Fix Bad Welds

- Try sorting the table by clicking on the column headers.
- Go back to sorting by ID (left-hand column).
- Select welds 3,4,18,19 & 20.
- Right-click in the blue area, click on “show conx and attached panels”



The screenshot shows the 'CONNECTION TABLE' window. A right-click context menu is open over the 'ID' column header. The menu options are: Cannot change ID, Update & remake, Sketch conx, Sketch FE, Show conx & layer panels, Show conx & attached panels (highlighted with a red box), Empty conx, Delete conx, Upd & remake (reposition), Upd & remake (swap layers), and Upd & remake P1 <=> P2 (bolts).

ID	Type	Subtype	Status	Error	Details	Part ID	Contact id	Layer 1	Layer 2	P1 (coords)
CNX3	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1367.704 650.
CNX4			Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1348.509 650.
CNX18			Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1195.257 671.
CNX19			Bad	All pairs of elem	None	9999		P10000001	P10000002	-1193.127 672.
CNX20			Bad	FAILED - Global	weld uses sam	9999		P10000001	P10000002	-1190.346 673.
CNX1139			Bad			9999		P10000122	P10000121	-235.0608 -63
CNX1140			Bad			9999		P10000122	P10000121	-234.9613 637.
CNX1141			Bad			9999		P10000122	P10000121	-222.0987 -55
CNX1142			Bad			9999		P10000122	P10000121	-221.9807 563.

Fixing Errors

- The selected connections, and the panels referred to, are displayed.
- The error is clear – the right-hand A-post has been referenced instead of the left-hand one.
- Right-click on the data in Layer 1 (P10000001), drop down to “Select PID”, select part 10000000.

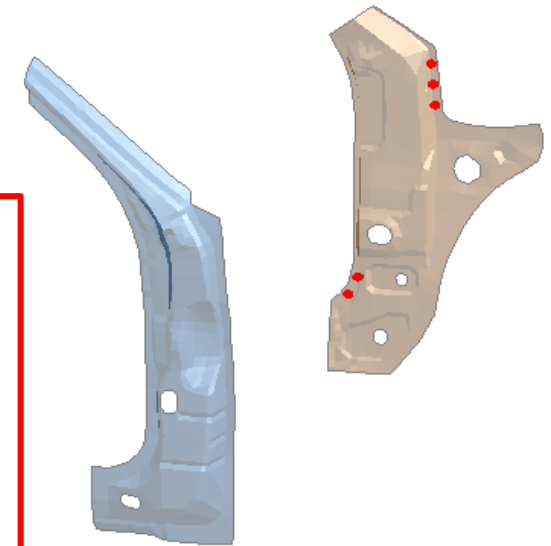
CONNECTION TABLE

Dismiss View... Options... Refresh Action: update & remake Show all Spotwelds Adhesive Write... ?

Apply: Undo All Selected Changed Autoscale Clear Sel all Select Show sel Bolts/Joints Spot lines Set column

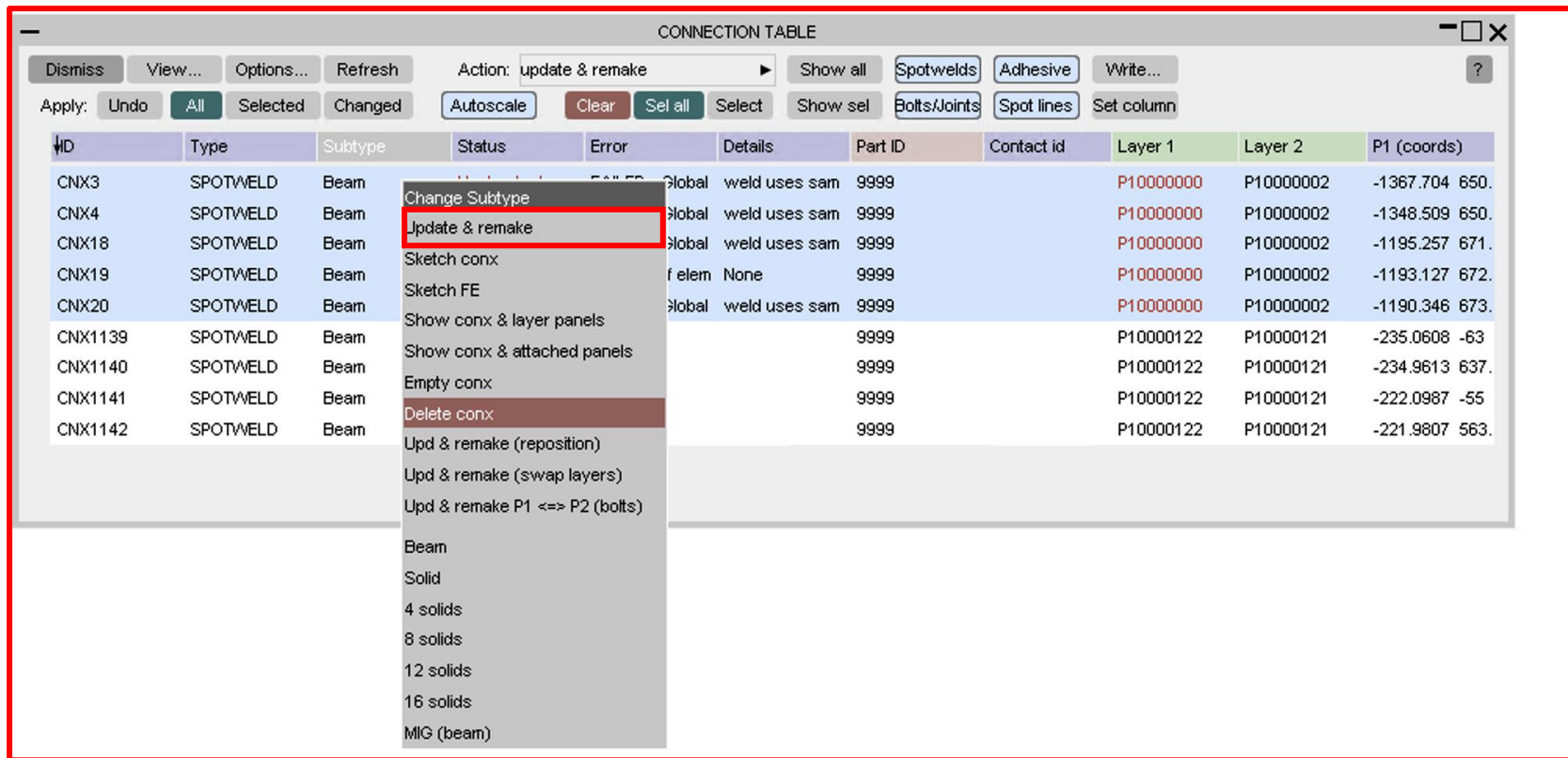
ID	Type	Subtype	Status	Error	Details	Part ID	Contact id	Layer 1	Layer 2	P1 (coords)
CNX3	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000		4 650.
CNX4	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000		9 650.
CNX18	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000		7 671.
CNX19	SPOTWELD	Beam	Bad	All pairs of elem	None	9999		P10000		7 672.
CNX20	SPOTWELD	Beam	Bad	FAILED - Global	weld uses sam	9999		P10000		3 673.
CNX1139	SPOTWELD	Beam	Bad			9999		P10000		3 -63
CNX1140	SPOTWELD	Beam	Bad			9999		P10000		3 637.
CNX1141	SPOTWELD	Beam	Bad			9999		P10000		7 -55
CNX1142	SPOTWELD	Beam	Bad			9999		P10000		7 563.

Change Layer 1
Update & remake
Sketch conx
Sketch FE
Show conx & layer panels
Show conx & attached panels
Empty conx
Delete conx
Upd & remake (reposition)
Upd & remake (swap layers)
Upd & remake P1 <=> P2 (bolts)
Select PID
More...
Change to...
Sketch layer
Delete layer

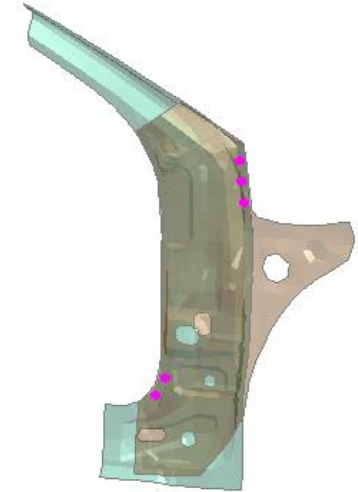


Fixing Errors

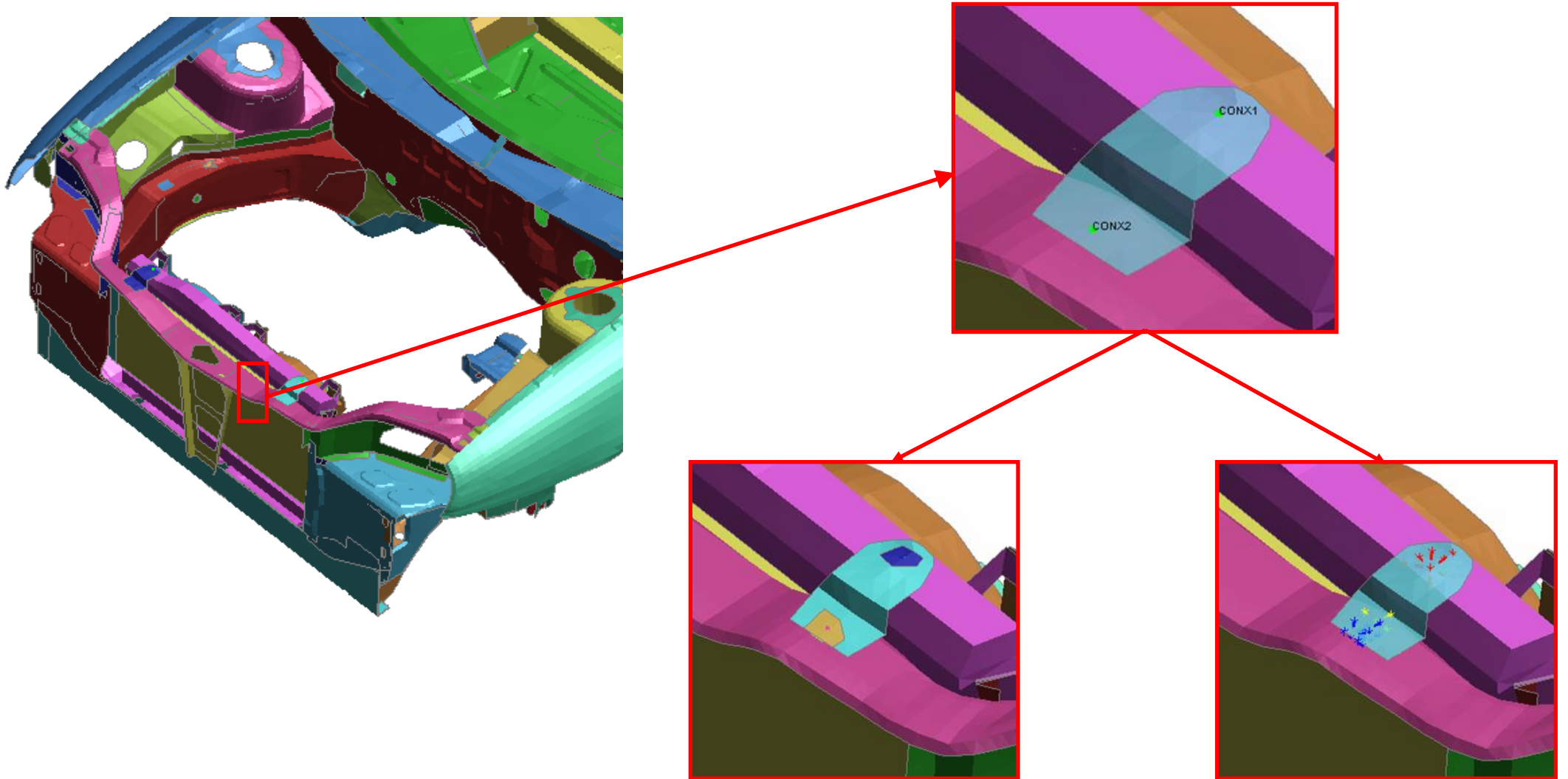
- The revised data is shown in red.
- To update the connection data and remake the weld, right-click in the blue area, click on “Update & remake”.
- Right-click again, “Show conx & panels” – to check that the correct panels are now shown.



ID	Type	Subtype	Status	Error	Details	Part ID	Contact id	Layer 1	Layer 2	P1 (coords)
CNX3	SPOTWELD	Beam			Global weld uses sam	9999		P10000000	P10000002	-1367.704 650.
CNX4	SPOTWELD	Beam			Global weld uses sam	9999		P10000000	P10000002	-1348.509 650.
CNX18	SPOTWELD	Beam			Global weld uses sam	9999		P10000000	P10000002	-1195.257 671.
CNX19	SPOTWELD	Beam			f elem None	9999		P10000000	P10000002	-1193.127 672.
CNX20	SPOTWELD	Beam			Global weld uses sam	9999		P10000000	P10000002	-1190.346 673.
CNX1139	SPOTWELD	Beam				9999		P10000122	P10000121	-235.0608 -63
CNX1140	SPOTWELD	Beam				9999		P10000122	P10000121	-234.9613 637.
CNX1141	SPOTWELD	Beam				9999		P10000122	P10000121	-222.0987 -55
CNX1142	SPOTWELD	Beam				9999		P10000122	P10000121	-221.9807 563.

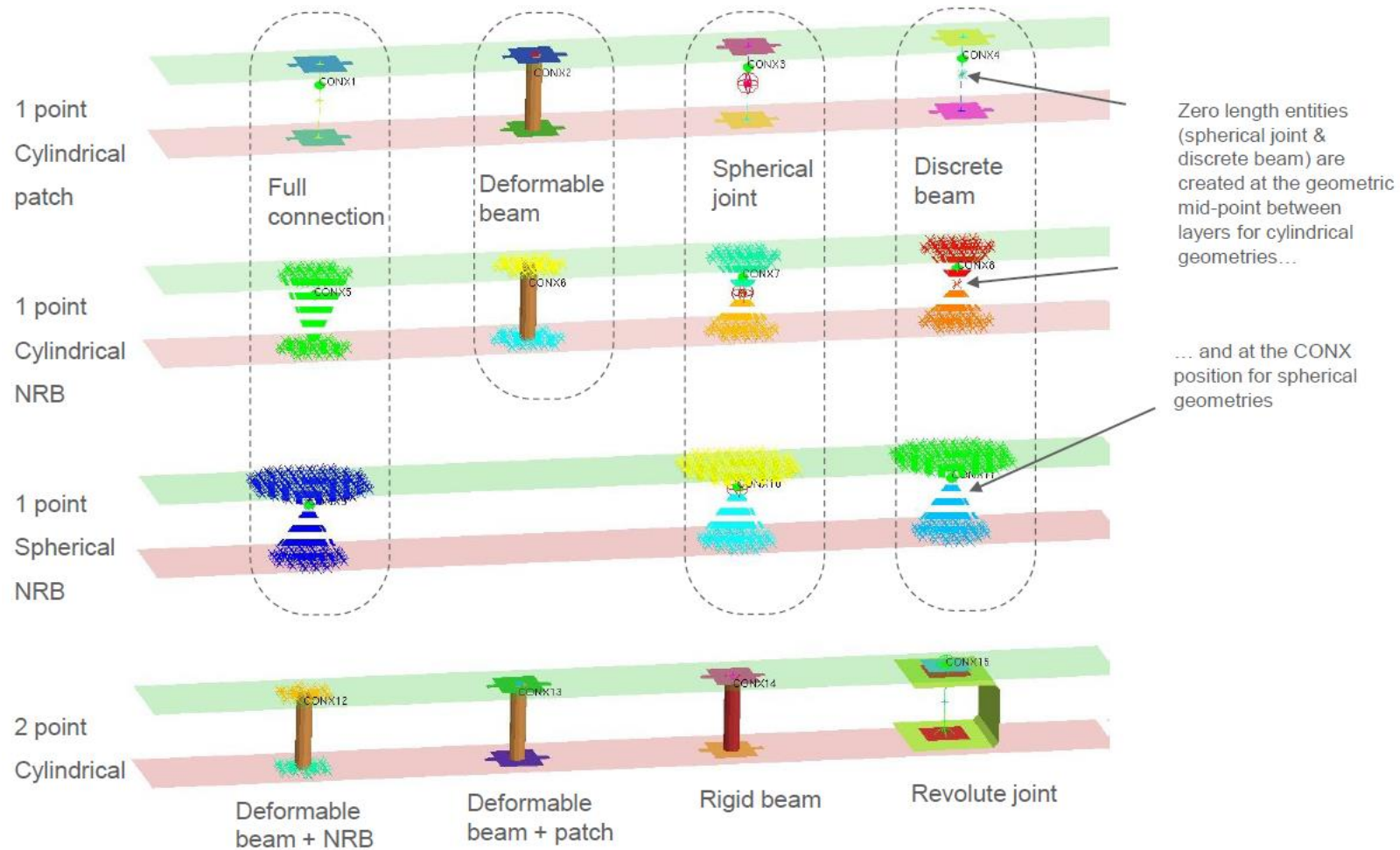


Bolt Connections



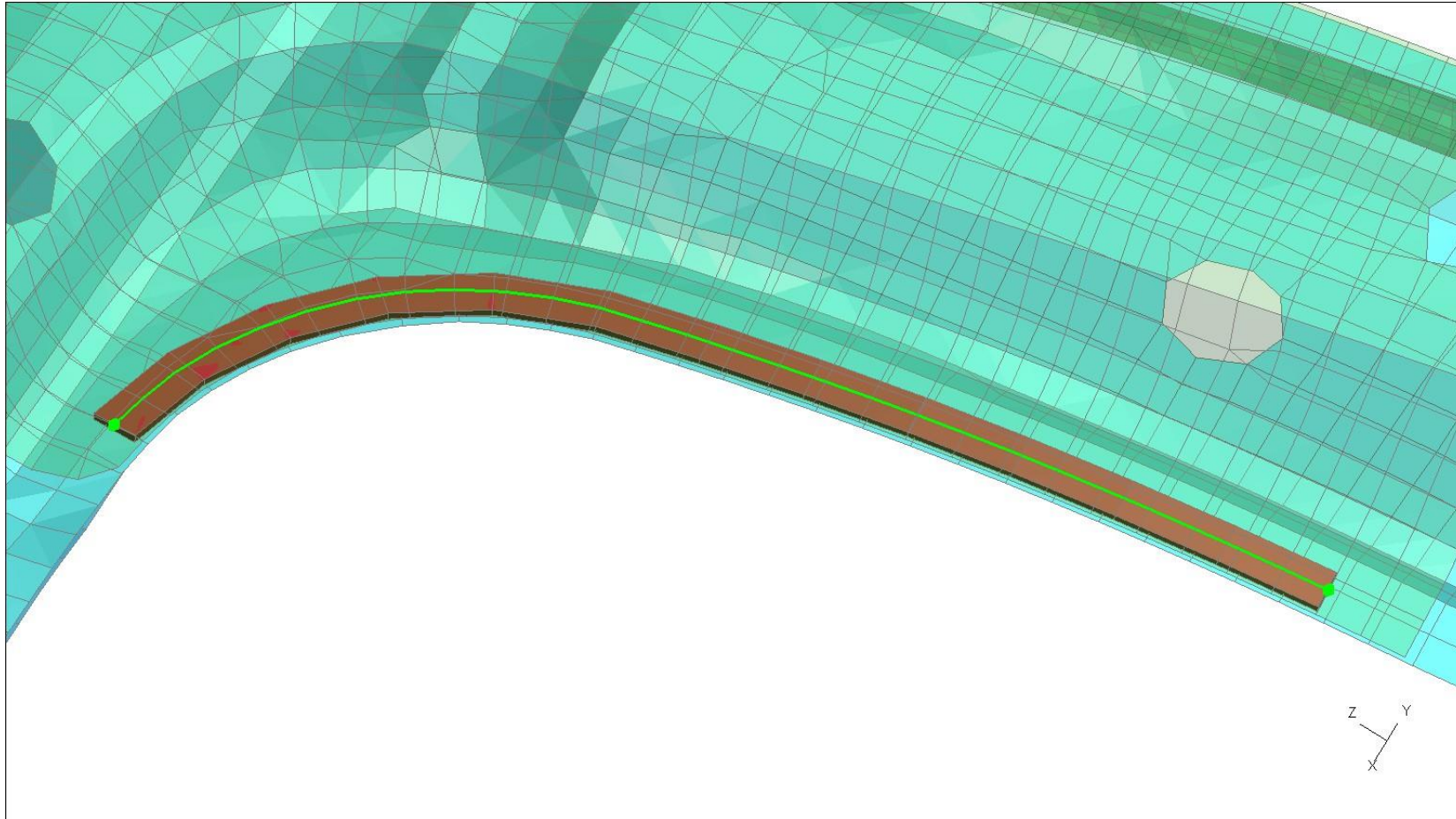
Bolt Connections

- Various bolt/joint connections can be created in PRIMER:



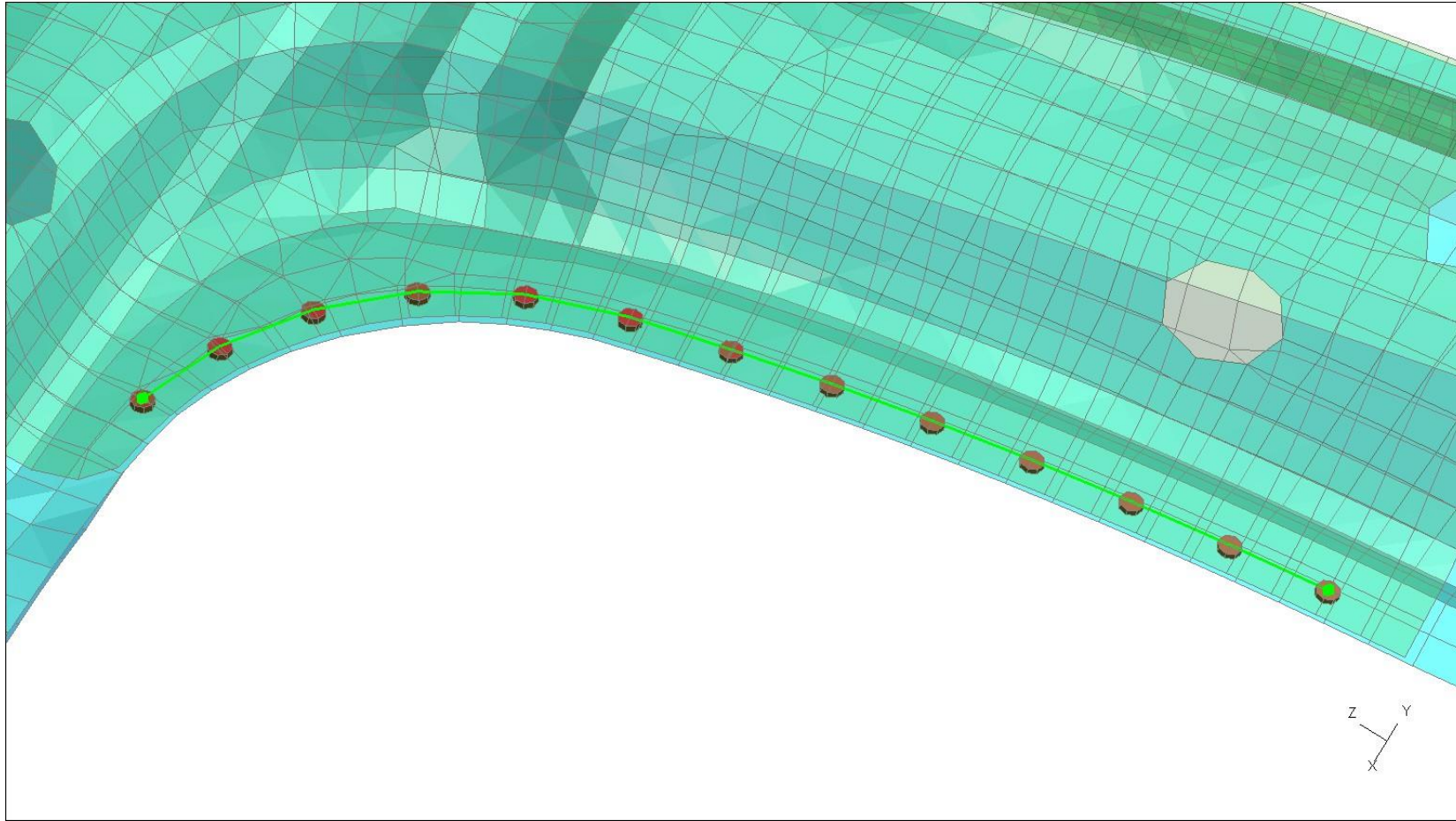
Other Connection Types - Adhesive

- Runs of solid adhesive can be created in PRIMER:



Other Connection Types - Adhesive

- Lines of spotwelds can be created in PRIMER:



More on Spotwelding

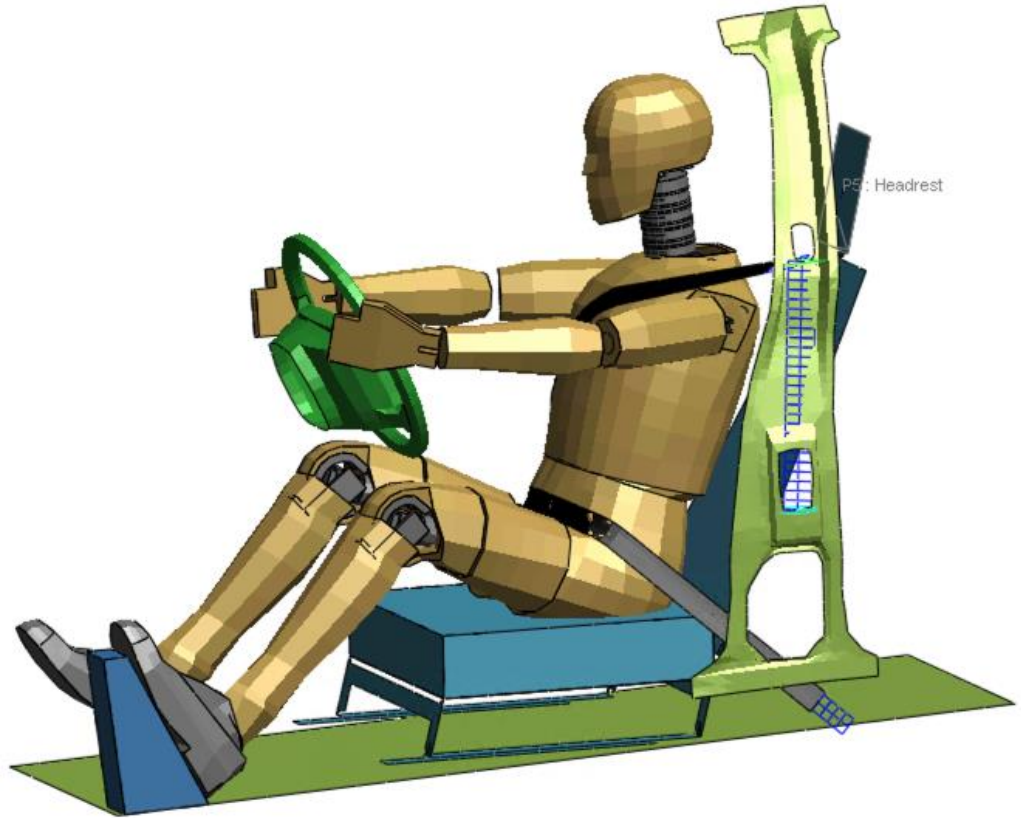
- In the separate course “spotwelds and connections”, we also cover:
 - Spotweld file formats.
 - More on checking and correcting errors in spotweld definitions.
 - Reproject to new panel geometry.
 - Creating and modifying bolts/adhesive.

Occupants

This section of the course is a reduced version of our separate Oasys PRIMER training course, “Seat and Dummy Positioning and Belt Fitting”

Occupants

- PRIMER has various tools available for creating/editing models containing occupants. Tools available include:
 - Dummy positioning
 - Seat squash
 - Mechanisms (for example, seats)
 - Seatbelt fitting
 - Airbag folding



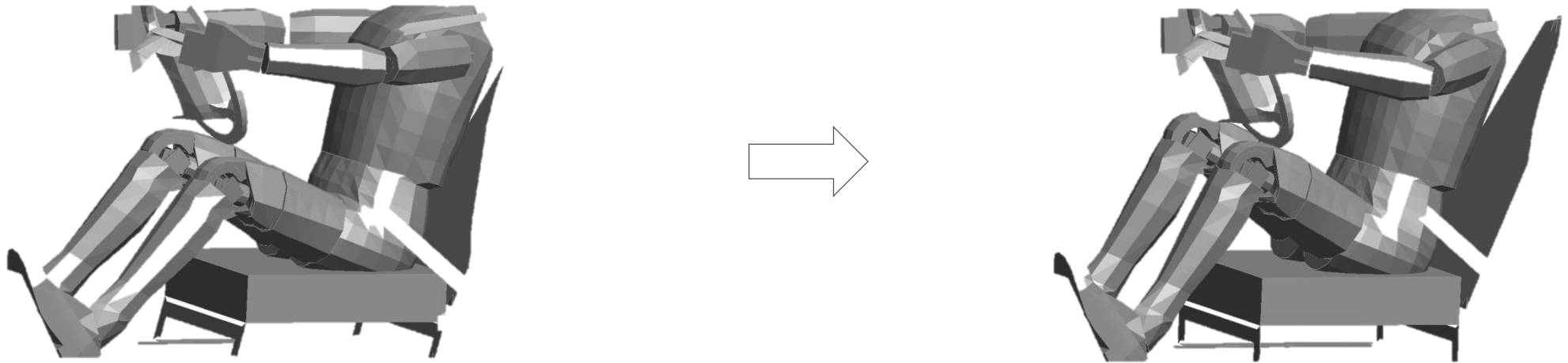
Occupants

- Seat mechanisms and dummies can be linked so that any seat movement will lead to the dummy moving as well.

Occupants

Delete all models and read in dummy_seat.key.

1. In the Tools menu, click **Mechanism->Position**.
2. Left mouse click on the seat and drag the mouse across the screen.
3. While moving the seat notice that the dummy also moves.
4. Notice the hands/feet are fixed.



Occupants

- After the occupant has been moved, the belt can easily be refitted over the dummy in the new position.

Occupants

From the last slide, leave the dummy in a position forward from the original.

1. In the Tools menu, click **Safety->Seatbelts**.
2. Click on **Auto-refit** then **Apply**.
3. Notice the belt will refit around the dummy. If you get a message about labels, click **SHIFT_OLD**.



Model Modified

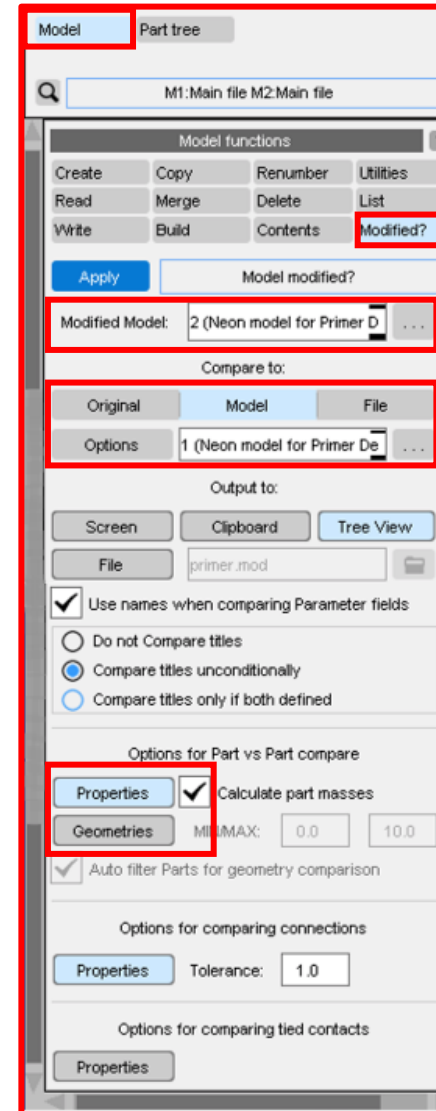
Model Modified

- Often, it is useful to see differences between models

Model Modified

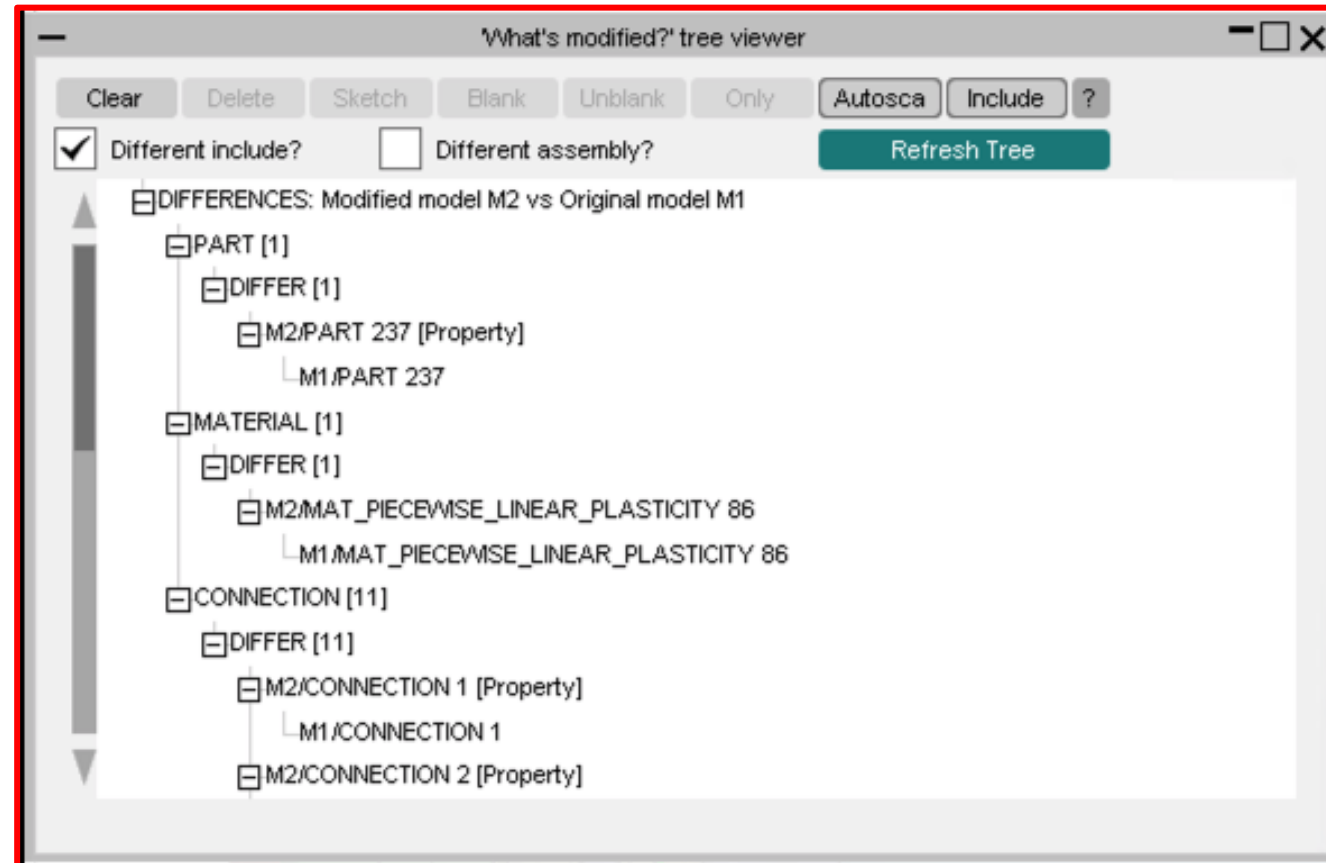
Delete all models and read in demo_car4.key. Also read in demo_car4_modified.key.

1. Go to **Model->Modified**
2. Select model 2 for the “Modified model”.
3. Select to compare to model 1.
4. Ensure that the **Properties** and **Geometries** switches are on.
5. Click **Apply**.



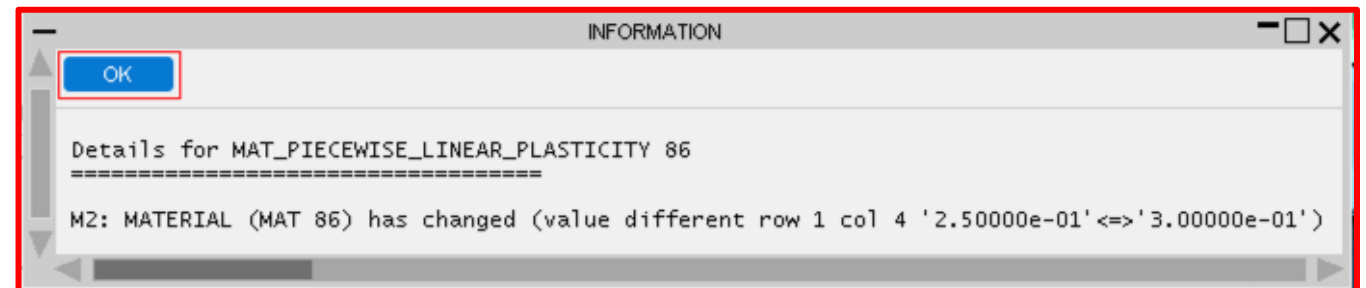
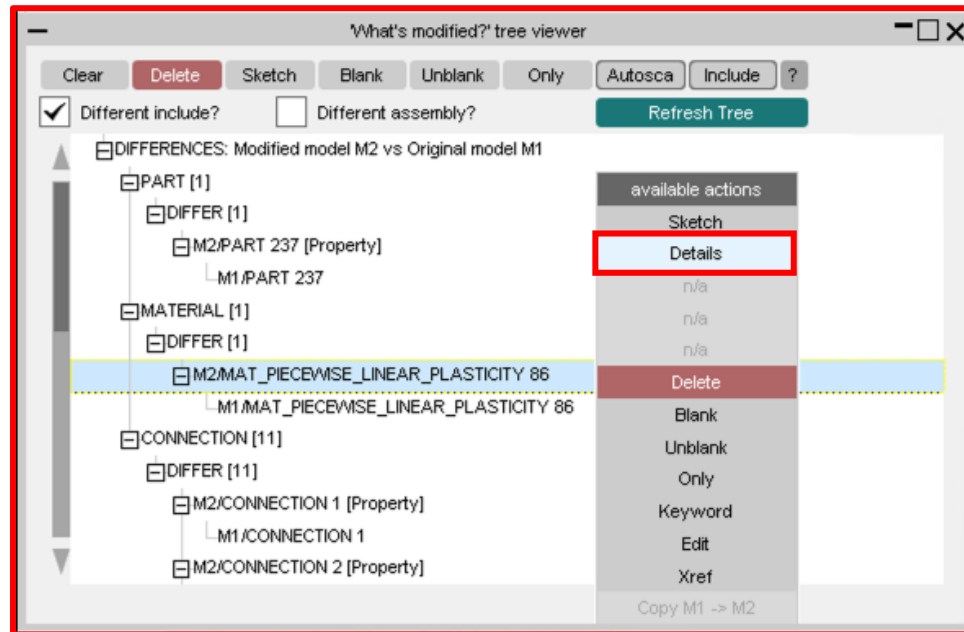
Model Modified

- Differences between the two models are shown in a tree view.
- PRIMER gives you information about differences in keywords, as well as additions/subtractions of entities between models.



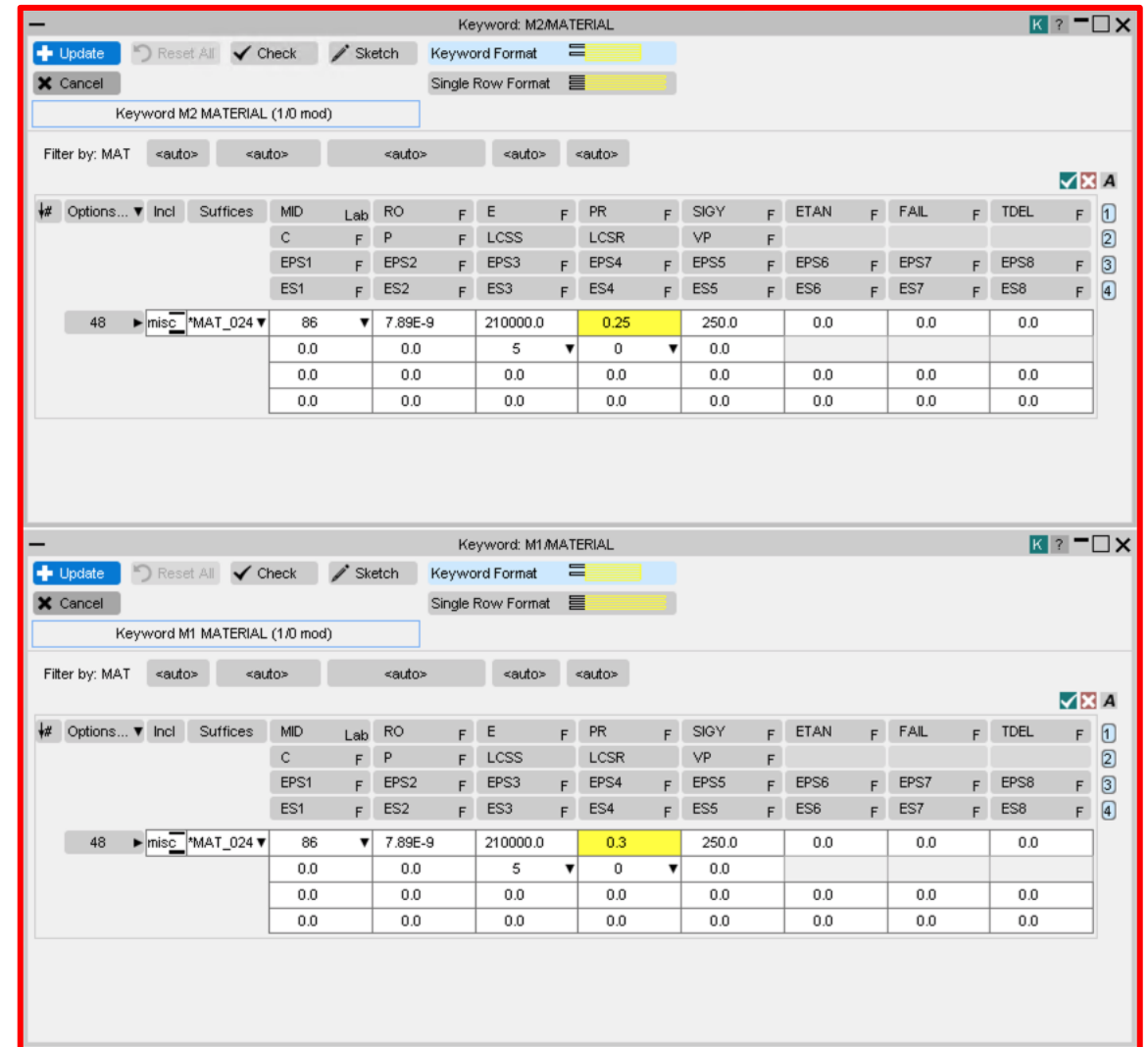
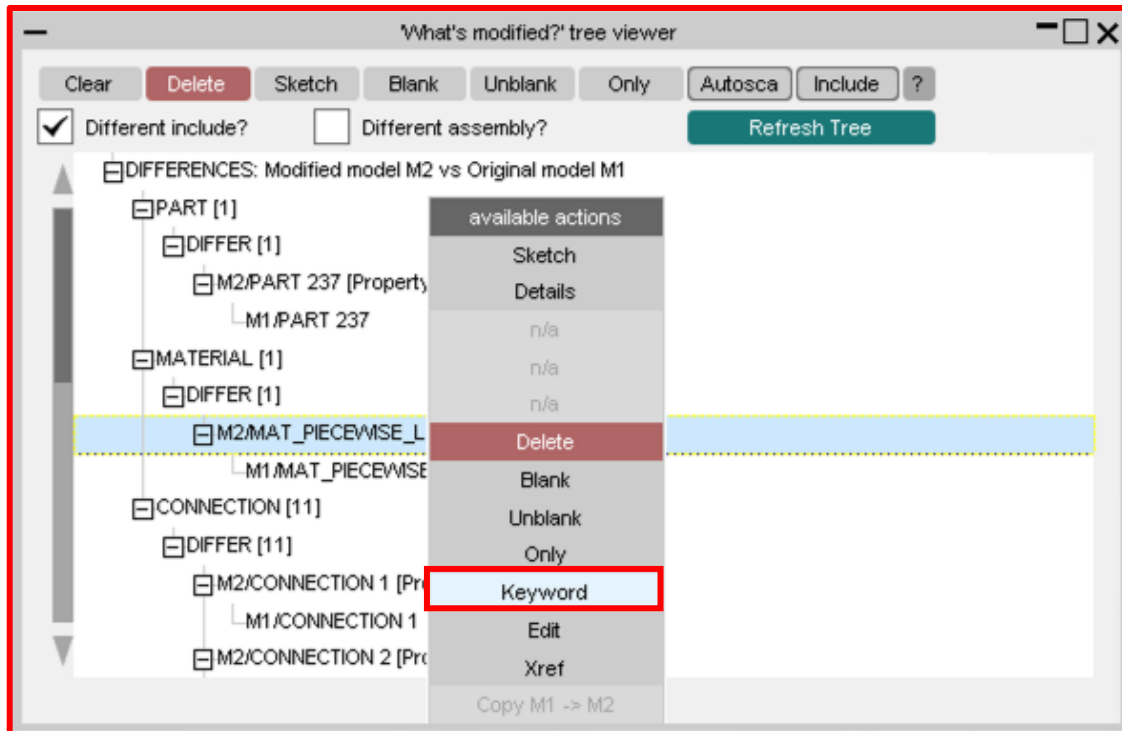
Model Modified

- Various options are available when right mouse clicking on one of the differences:
 - **Details** will print details of the differences found.



Modified Models

- Various options are available when right mouse clicking on one of the differences:
 - Keyword** will display keyword panels highlighting the differences between the two cards.



Find

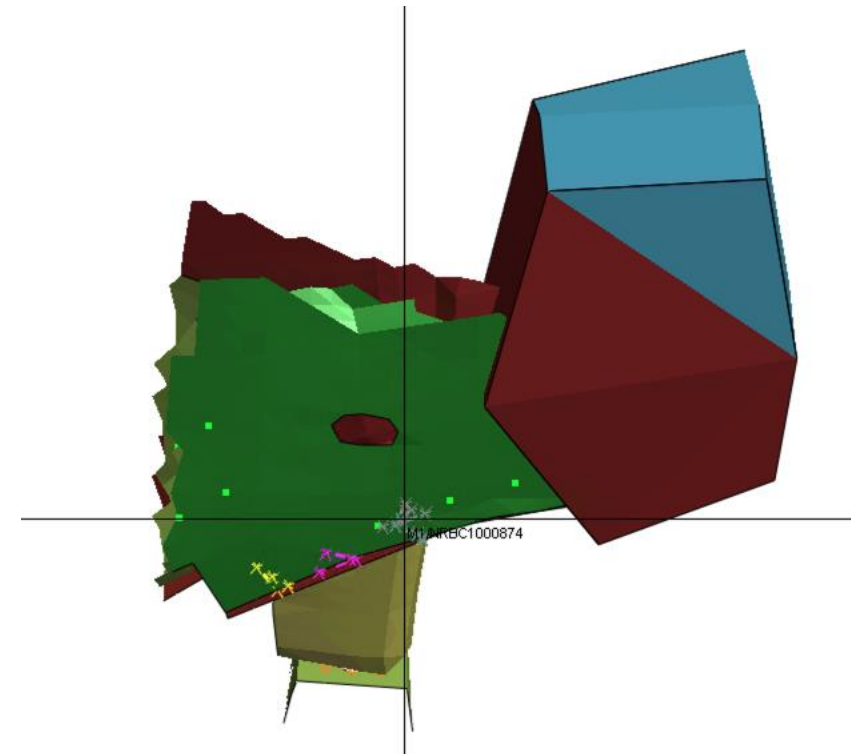
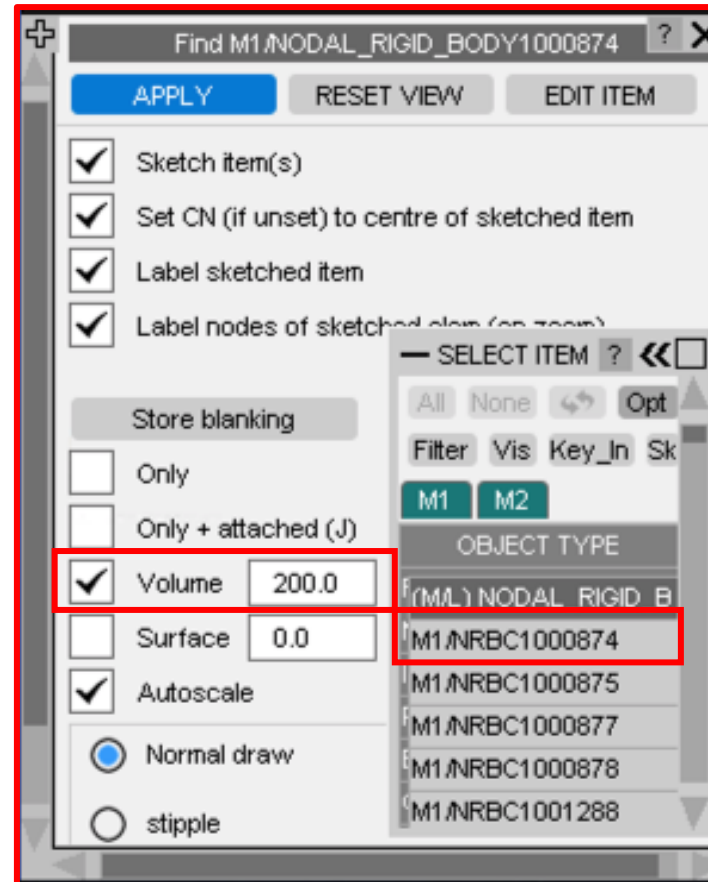
Find

- The “Find” function can be used to easily find entities in the model, and see their surroundings.

Find

Delete all models and read in demo_car4.key.

- In the Tools menu, click **Find**.
- Select **NODAL_RIGID_BODY** in the selection menu. Choose the first one in the list (**1000874**).
- PRIMER will sketch the location of the NRB with crosshairs.
- Turn on **Volume**. PRIMER will show entities within a specified volume around the NRB.



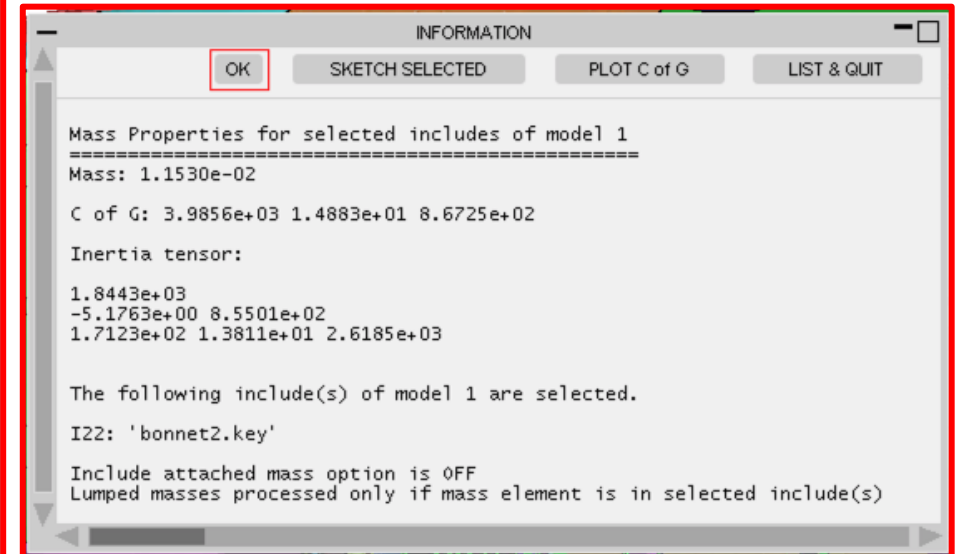
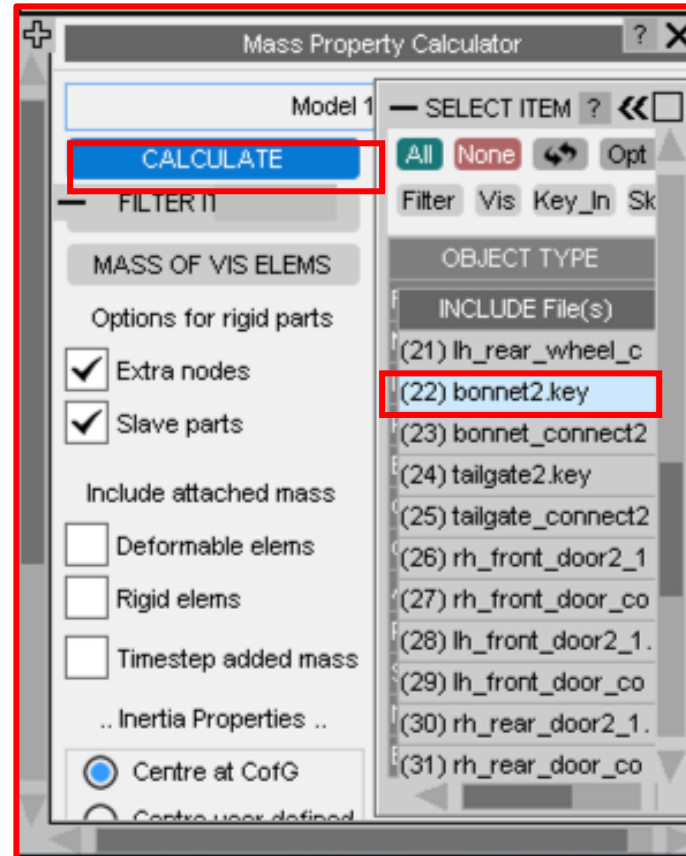
Mass Properties

Mass Properties

- The “Mass Properties” function can be used to calculate mass properties of selected entities.

Mass Properties

- In the Tools menu, click **Mass Prop.**
- Select **INCLUDE FILE** in the selection menu. Choose **(22) bonnet2.key**.
- Click **CALCULATE**. PRIMER will calculate mass properties of the include based on the settings on the panel, and report the result.
- Various options are available for how to handle attached mass, and inertia calculations.



Scripting

Why write scripts within PRIMER?

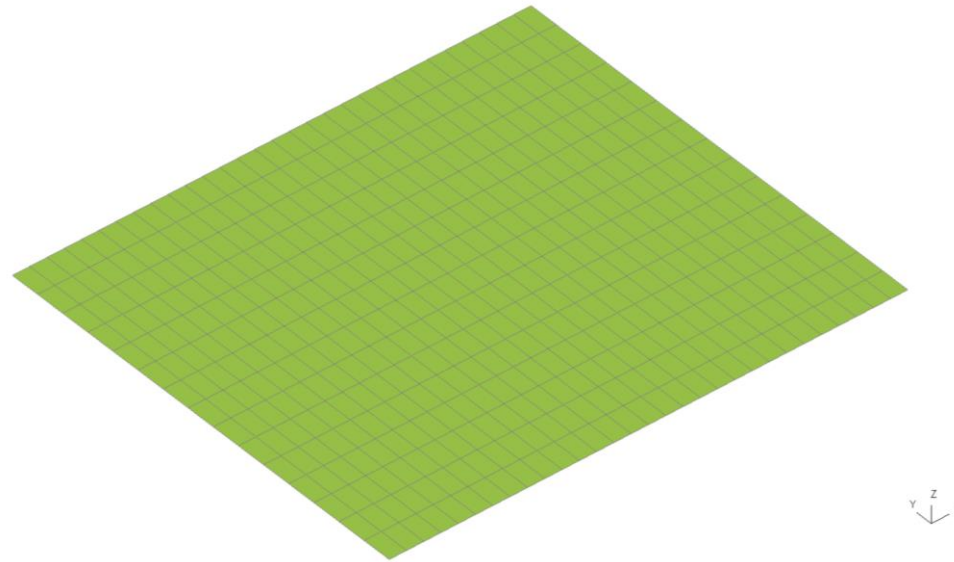
- Advantages of writing a JavaScript to create a new capability:
 - Quick turnaround – don't have to wait for new version of T/HIS, D3PLOT or PRIMER
 - Can keep your application confidential
 - Under your control – can do it yourself if you wish.
- Example applications (PRIMER):
 - Creating a simple mesh, or test models with standard loading
 - Data checking or correcting
 - Geometric morphing functions
 - Input or output translators, special-format spotweld or connections files
 - Automating routine tasks
- Example applications (D3PLOT):
 - Generating your own data components for plotting, calculated from any information already contained in the model or from external files

Scripting

- An example of a PRIMER script is to generate a simple mesh.

Multi-Model Build

1. Delete all models within PRIMER.
2. Go to **Tools->Script** and use the file selector to select make_shell_mesh.js. Click **Run**.
3. A mesh should appear.
4. Open the file make_shell_mesh.js in a text editor and change some of the variables at the top. Rerun the script to see the difference.



Parameters

*PARAMETER in PRIMER

- *PARAMETER cards allow you to specify parameter values that can then be referred to anywhere in the model.
- This allows you to change one PARAMETER rather than changing multiple fields, e.g. setting thickness on multiple section cards.

*PARAMETER

R GAUGE 5.2

*SECTION_SHELL

1	2	0.0	0	0.0	0.0	0	0
---	---	-----	---	-----	-----	---	---

&GAUGE	&GAUGE	&GAUGE	&GAUGE	0.0	0.0	0.0	0
-------------------	-------------------	-------------------	-------------------	-----	-----	-----	---

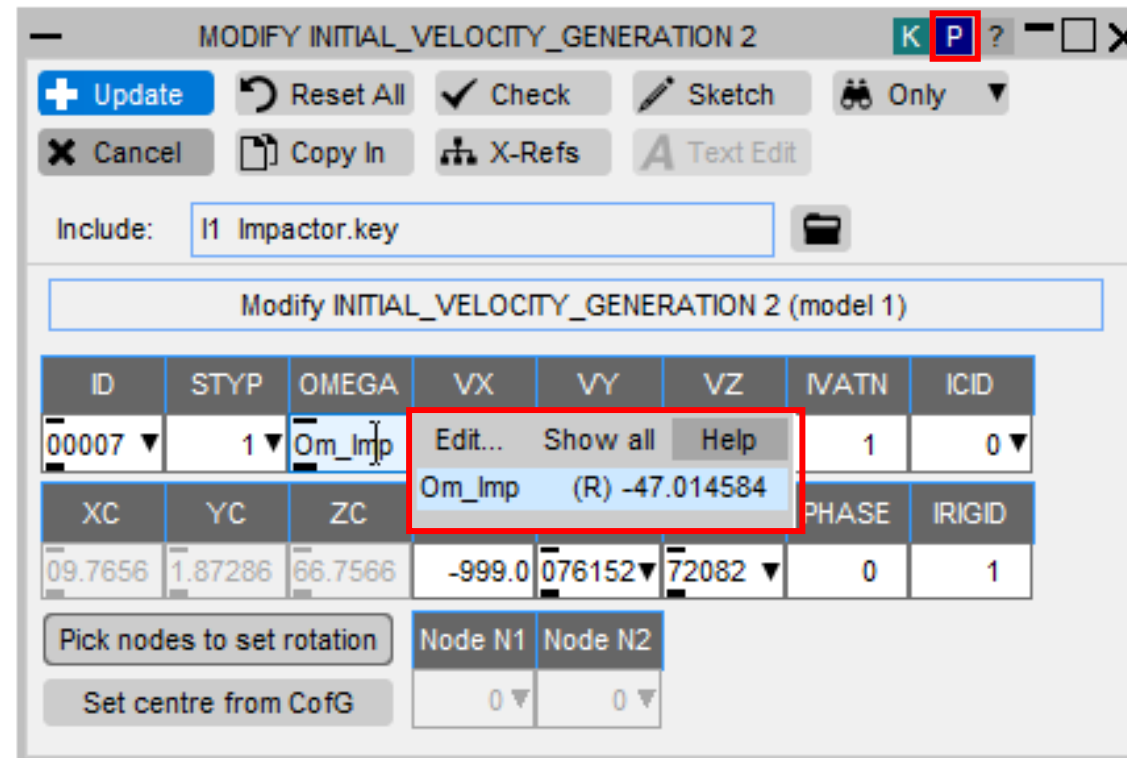
\$

2	2	0.0	0	0.0	0.0	0	0
---	---	-----	---	-----	-----	---	---

&GAUGE	&GAUGE	&GAUGE	&GAUGE	0.0	0.0	0.0	0
-------------------	-------------------	-------------------	-------------------	-----	-----	-----	---

*PARAMETER in PRIMER

- *PARAMETER is supported in PRIMER.
- Type “&” in the input boxes and a popup will allow you to select/edit/create a *PARAMETER.



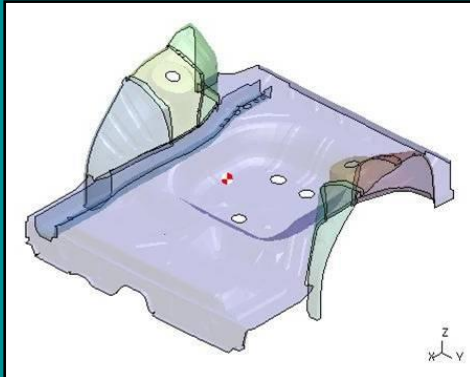
“P” button switches the display of parameters in the panel from the parameter name to the underlying values.

Functions not covered in detail in this course

Assign Mass

- Massing up an assembly:

Create Assembly



Input Required Mass

CREATE ASSIGN MASS in model 1

Buttons: Create, Reset All, Check, Sketch, Only, Calculate, Cancel, Copy in, X-Refs, Text Edit, Plot mass to be added

Include: M1 <Master file>

Create ASSIGN MASS (model 1)

ASSM Label: 2
ASSM Title:

☐ PART_SET
☐ ASSEMBLY
☒ GROUP

☒ NODE MASS
☐ PART MASS

Group: 1
Title: all parts

Set Mass: 3.0E-2
Set CoG: X: 1200.0 Y: * Z: 558.0
Set Inertia: XX: * XY: * XZ: *
Reset All
Error tolerance (percent): 5.0
Lower ID: 1 Upper ID: 99999999

☒ Include attached mass
☐ Include attached mass from ASSM of higher label?

☒ Change mass and CoG by changing entire group
☐ Change mass and CoG by changing a subset of the group

Subgroup id: N/A
Title:

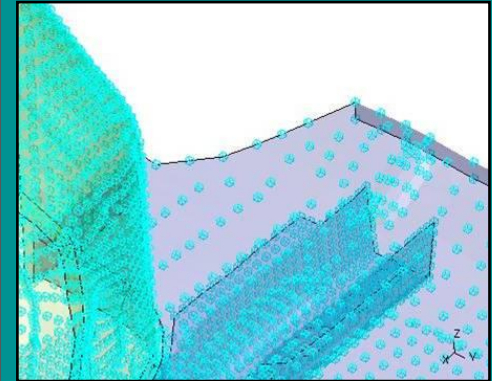
Lock Element_Mass_Node_Set
Lock assign mass hierarchy

Original mass and properties of group. Red on white CoG. Include timestep added mass: ☐

Actual mass:	1.200009	Show CoG	
Actual CoG: X:	2752.139	Y: 35.07087	Z: 635.9129
Inertia tensor: IXX:	425230.5	IYY: 1883022.	IZZ: 2084935.
		IXY: 32341.76	IYZ: 621.3727
		IXZ: 70306.9	IYZ: 621.3727

Included mass from Part Inertia(s): <none> Sketch Part Inertia
Included mass from NRB Inertia(s): <none> Sketch NRB Inertia
Excluded Part Inertia & NRB elements: <none> Sketch Excluded

Distributed mass added using *ELEMENT_MASS



Bill of Materials

- PRIMER can read any type of delimited file (e.g. CSV from Microsoft Excel)
- User selects field type from popup menu (Model PID, thickness, material, element formulation, etc)
- Data is transferred into CAE model
- PRIMER can also write a Bill of Materials file

Cancel < Prev Apply Help

Define the fields in the file

None of the parts in BOM file have differing information, therefore no PART(s) will be updated

Select fields in Bill of Materials

Select parts from BOM file to update:

☒ All in file ☐ Subset

Select PART(s)

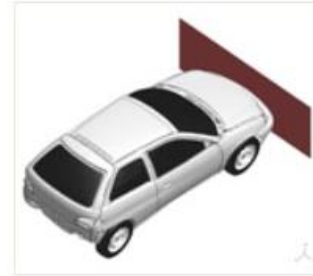
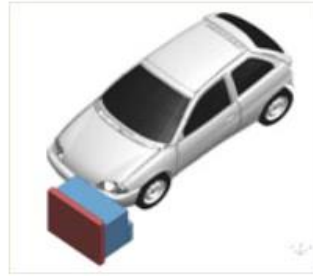
All in file

Which parts will be modified?

Sketch Only Explain

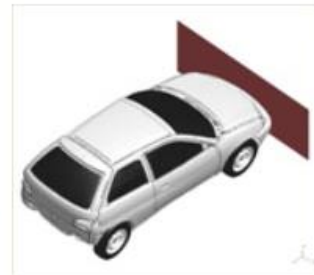
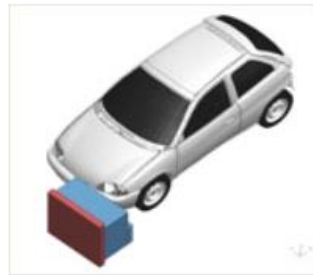
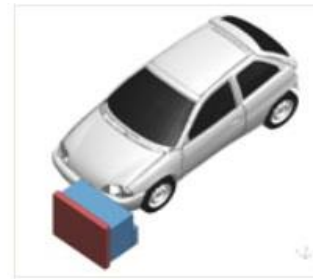
	A	B	C	D	E	F	G	H	I	J
Field	PID ▼	CAD part no. ▼	Part description ▼	Material ID ▼	Section ID ▼	Hourglass ID ▼	Material title ▼	Section title ▼	Gauge ▼	Hourglass type ▼
1	\$ Primer Bill o									
2	\$ =====									
3	\$									
4	\$ Created on: F									
5	\$									
6	\$ from model: "									
7	\$									
8	Part ID	CAD Part No	Title	Material ID	Section ID	Hourglass ID	Material name	Section name	Gauge	HG type
9	\$									
10	152		MC-A-ARM-BUSH1-1	1	1	0				

Automatic Assembly and Loadcase Control

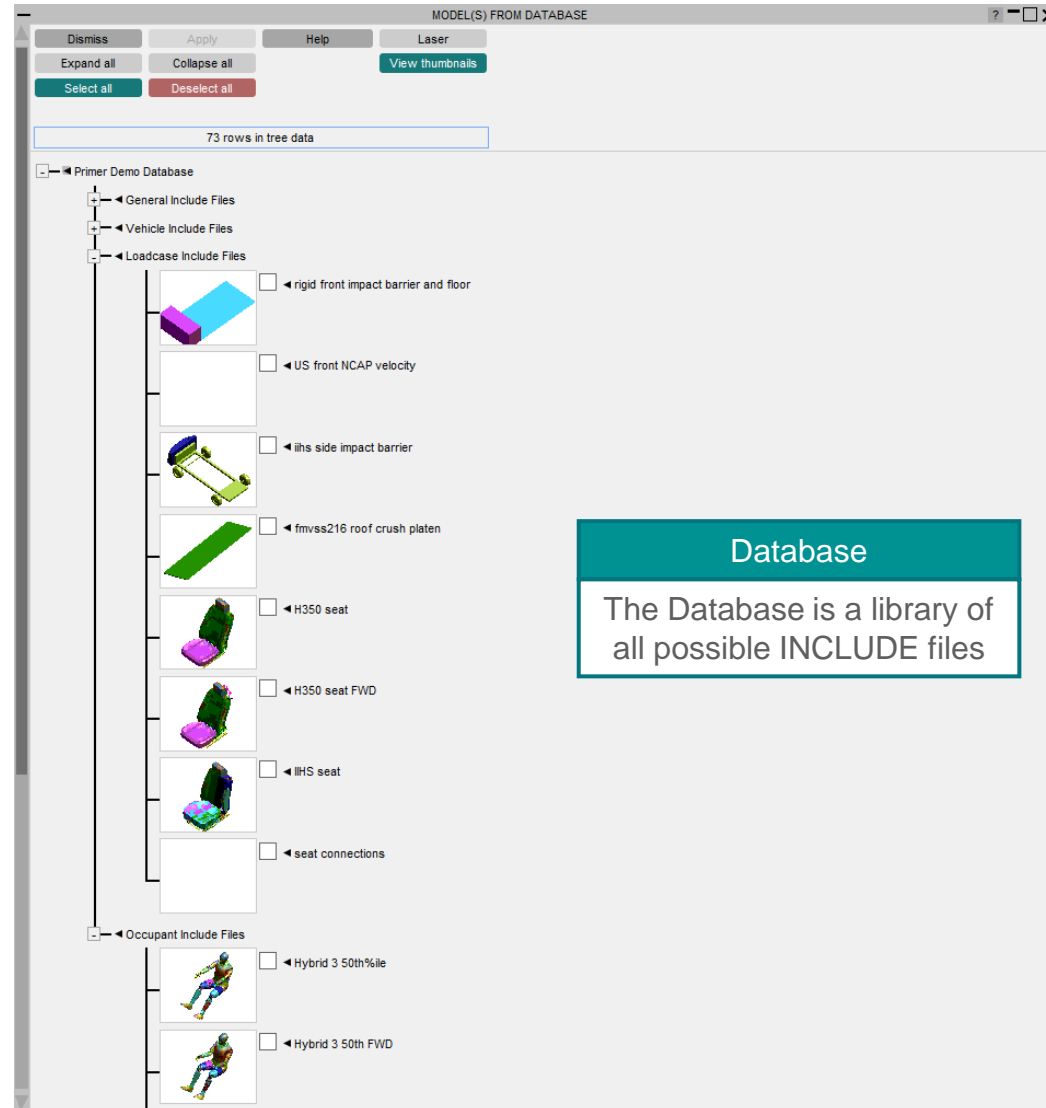


Automatic Model Assembly

PRIMER can assemble models from a Database of INCLUDE files. For each loadcase or model configuration, a template defines which INCLUDE files are selected.

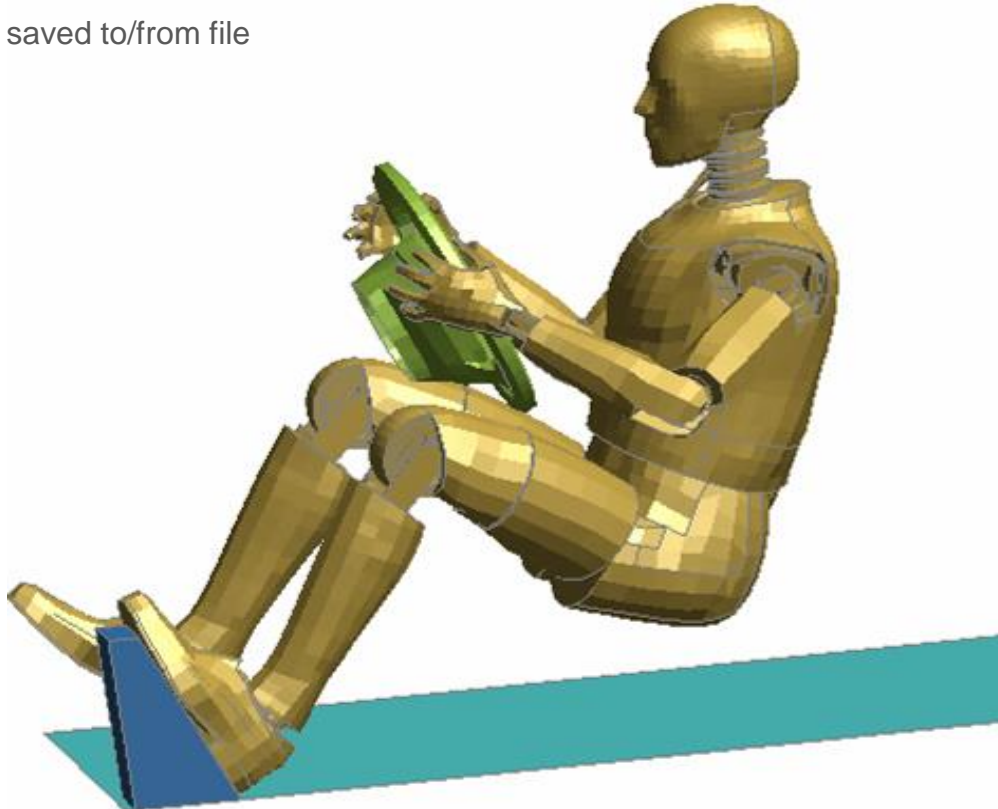


Automatic Assembly and Loadcase Control



Dummy Positioning

- Dummy can be positioned by
 - Rotating joints
 - Dragging parts, restraining other parts
 - Typing desired coordinates for H-point or other reference points
 - Positions saved to/from file



Dummies

Curr: M1/DUMM100000. H-pt: 420.0 200.0 215.0

Position parts

Accept Accept and save Global Accuracy
Reject Reject and undo 1.0 ?

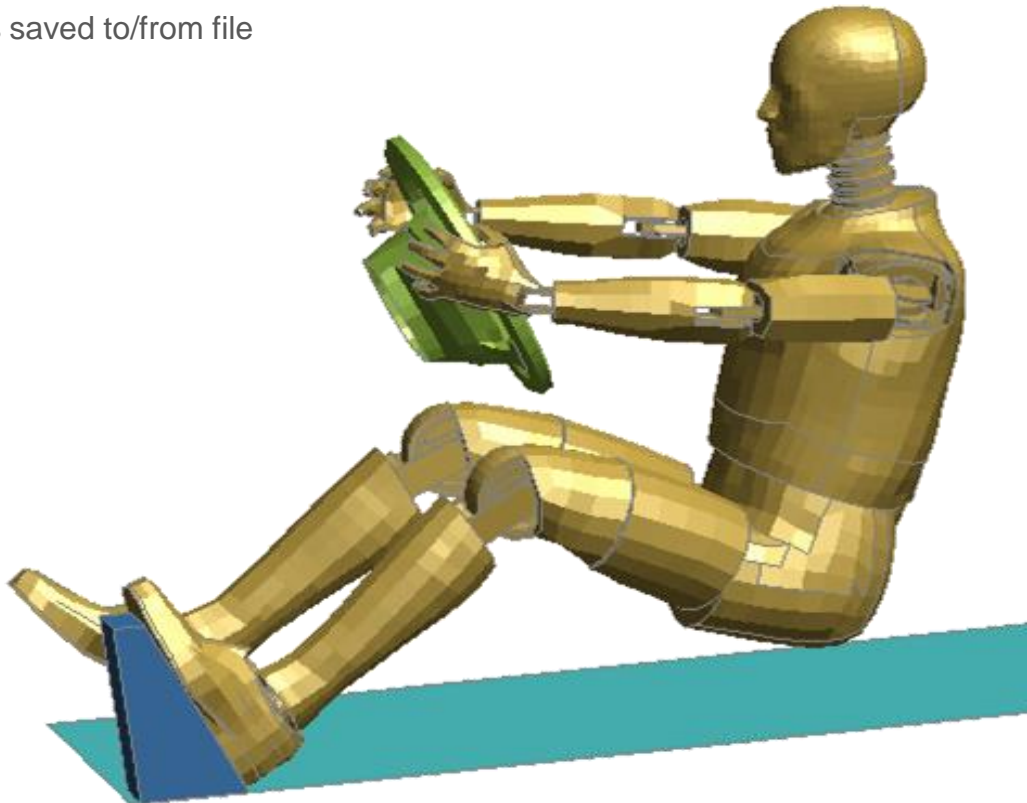
☐ Rotate angles ☒ Drag assembly ☐ Move points Explain Options...
Reset all Save/Retrieve

Assembly (click to edit) Cont Lock tr Lock rot

1: TORSO	T all x y z	R all x y z
2: UPPER_ARM_LEFT	T all x y z	R all x y z
3: LOWER_ARM_LEFT	T all x y z	R all x y z
4: HAND_LEFT	T all x y z	R all x y z
5: UPPER_ARM_RIGHT	T all x y z	R all x y z
6: LOWER_ARM_RIGHT	T all x y z	R all x y z
7: HAND_RIGHT	T all x y z	R all x y z
8: UPPER_LEG_LEFT	T all x y z	R all x y z
9: LOWER_LEG_LEFT	T all x y z	R all x y z
10: FOOT_LEFT	T all x y z	R all x y z
11: UPPER_LEG_RIGHT	T all x y z	R all x y z
12: LOWER_LEG_RIGHT	T all x y z	R all x y z
13: FOOT_RIGHT	T all x y z	R all x y z
14: YOKE_LEFT	T all x y z	R all x y z
15: YOKE_RIGHT	T all x y z	R all x y z

Dummy Positioning

- Dummy can be positioned by
 - Rotating joints
 - Dragging parts, restraining other parts
 - Typing desired coordinates for H-point or other reference points
 - Positions saved to/from file



Dummies

Curr: M1/DUMM100000. H-pt: 420.0 200.0 215.0

Position parts

Accept Accept and save Global Accuracy
Reject Reject and undo 1.0 ?

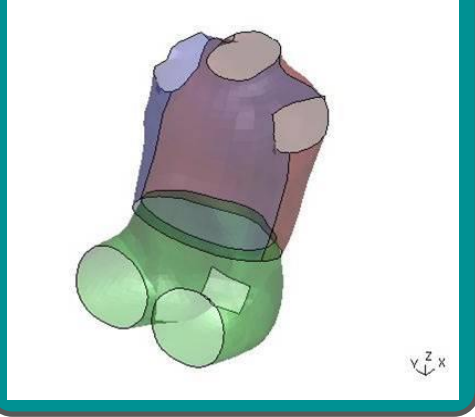
☐ Rotate angles ☒ Drag assembly ☐ Move points Explain Options...
Reset all Save/Retrieve

Assembly (click to edit) Cont Lock tr Lock rot

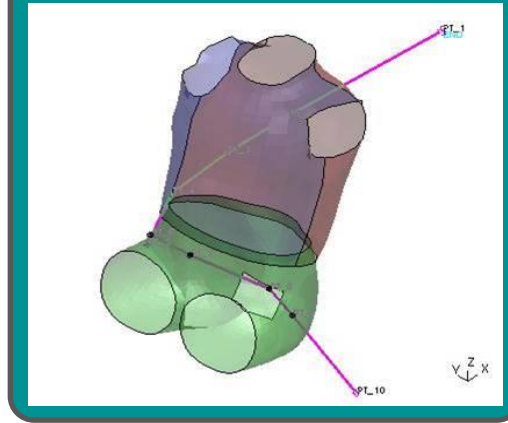
1: TORSO	T all x y z	R all x y z
2: UPPER_ARM_LEFT	T all x y z	R all x y z
3: LOWER_ARM_LEFT	T all x y z	R all x y z
4: HAND_LEFT	T all x y z	R all x y z
5: UPPER_ARM_RIGHT	T all x y z	R all x y z
6: LOWER_ARM_RIGHT	T all x y z	R all x y z
7: HAND_RIGHT	T all x y z	R all x y z
8: UPPER_LEG_LEFT	T all x y z	R all x y z
9: LOWER_LEG_LEFT	T all x y z	R all x y z
10: FOOT_LEFT	T all x y z	R all x y z
11: UPPER_LEG_RIGHT	T all x y z	R all x y z
12: LOWER_LEG_RIGHT	T all x y z	R all x y z
13: FOOT_RIGHT	T all x y z	R all x y z
14: YOKE_LEFT	T all x y z	R all x y z
15: YOKE_RIGHT	T all x y z	R all x y z

Seatbelt Fitting

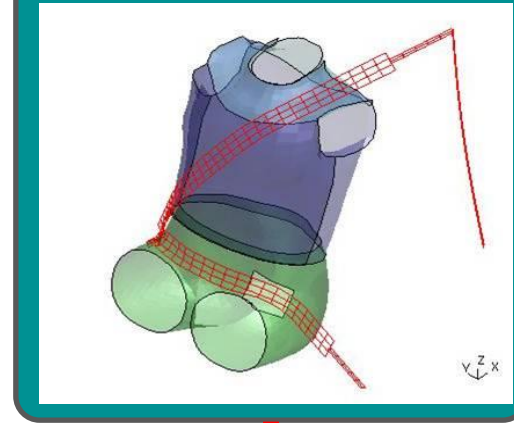
1. Define belt



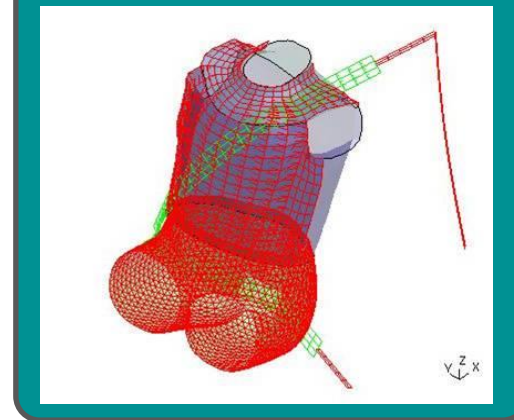
2. Define path



3. Mesh belt



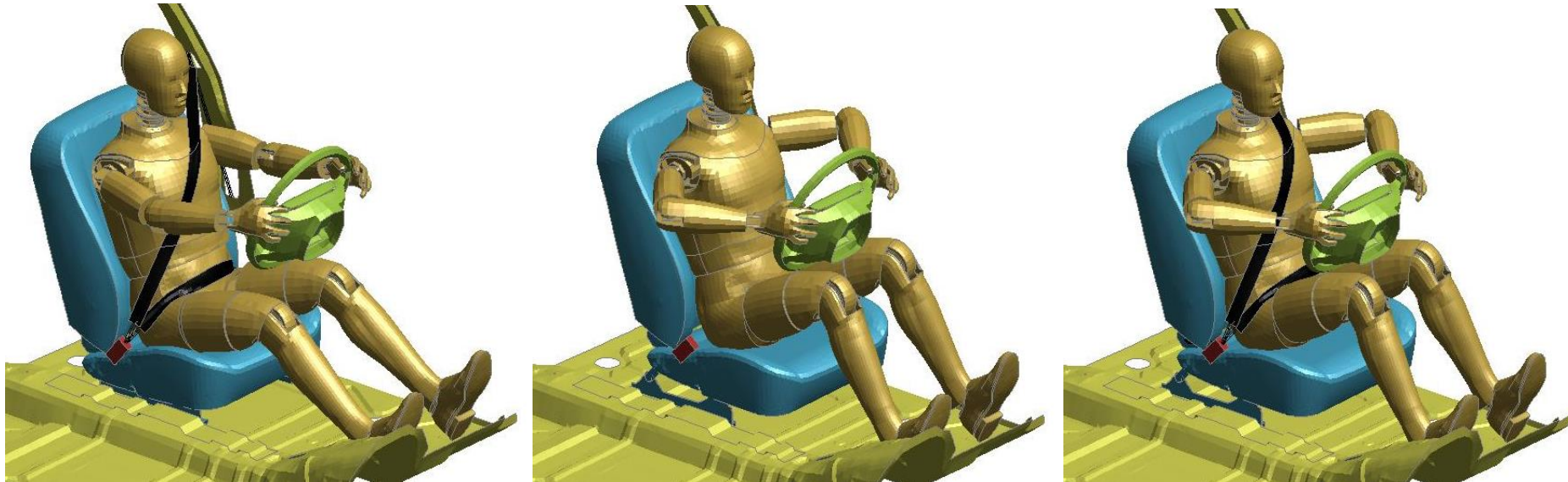
4. Setup contact



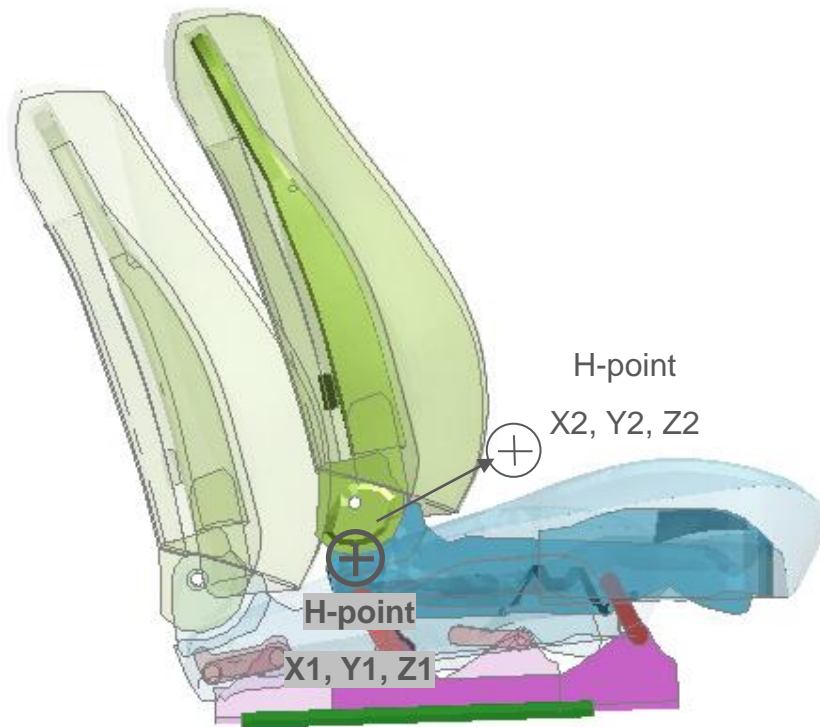
- Belt creation in 4 steps
- Once created slipping, pretensioner, retractor can be added from belt menu

Seatbelt Fitting

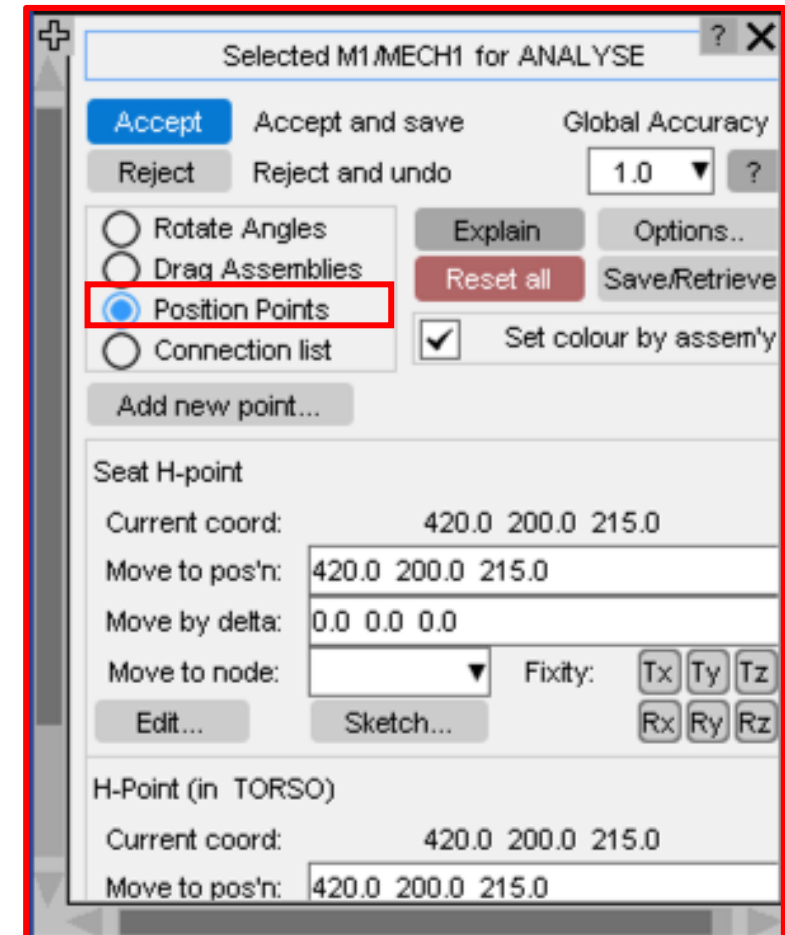
- PRIMER remembers the belt information at the end of the keyword files so the belt can be re-fitted in one operation if the dummy is moved.



Mechanisms

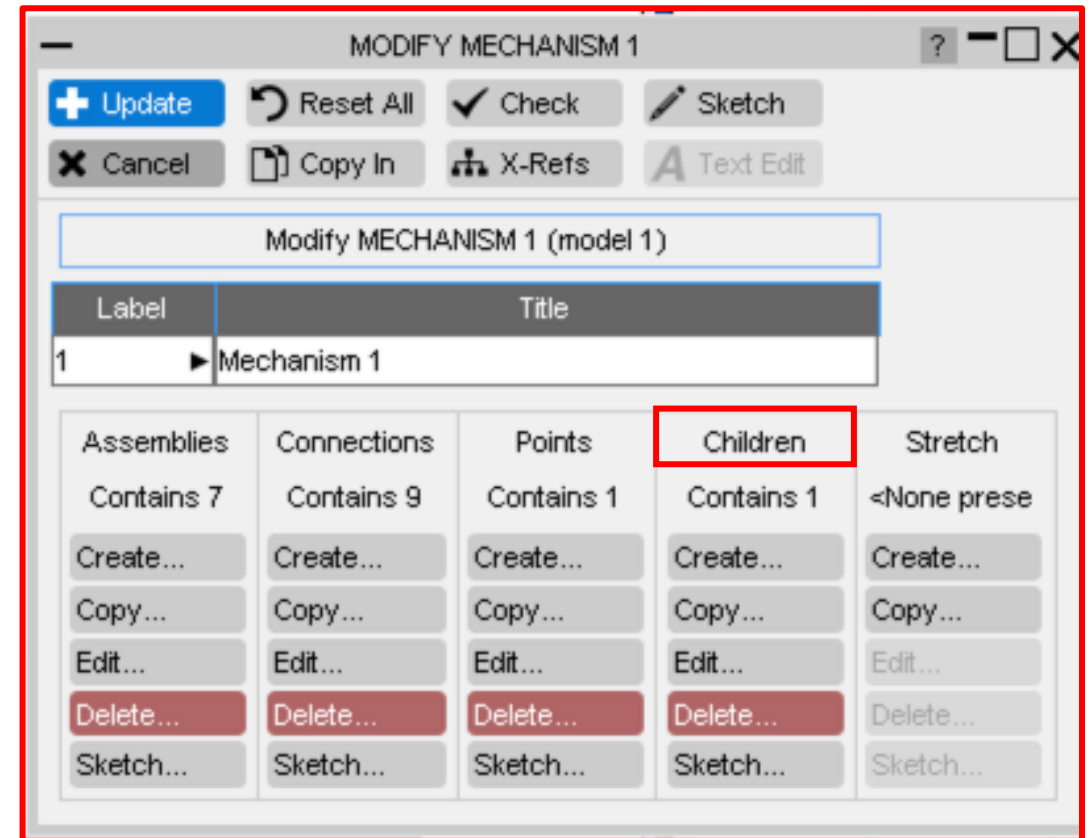
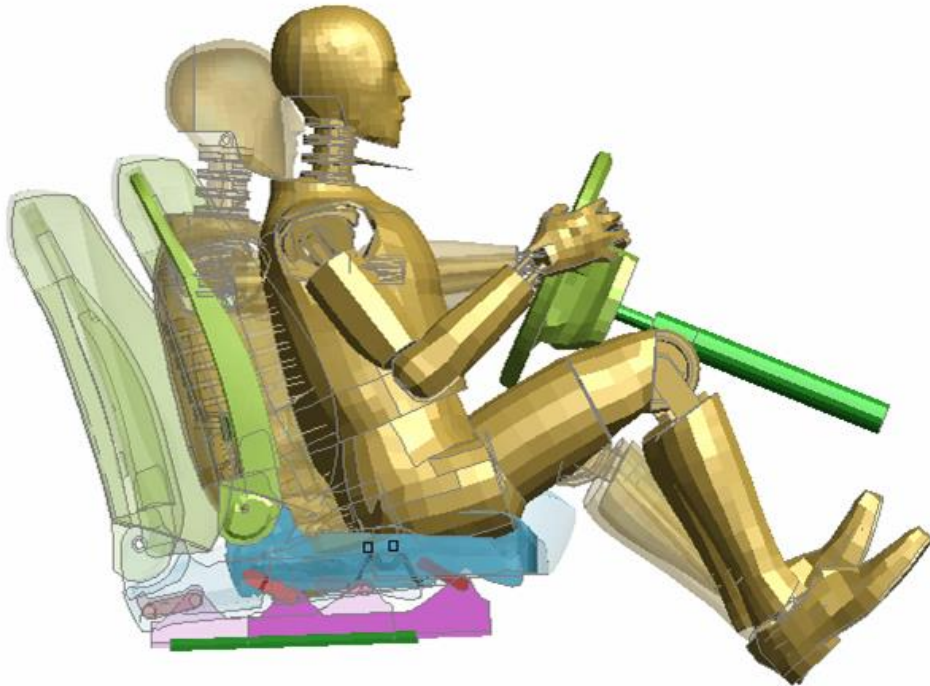


- A mechanism (for example, a seat) can be defined within PRIMER. The mechanism consists of Assemblies (groups of parts) and joints (pin, hinge or sliding).
- The mechanism can be positioned by dragging, or by typing in the desired coordinates of “reference points”

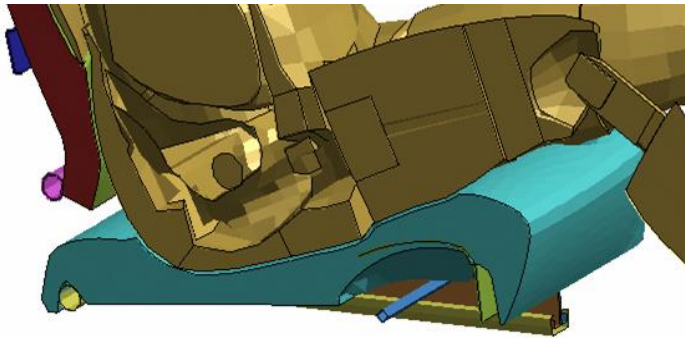
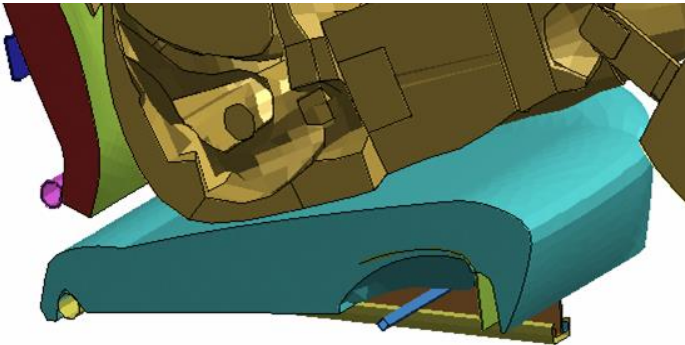
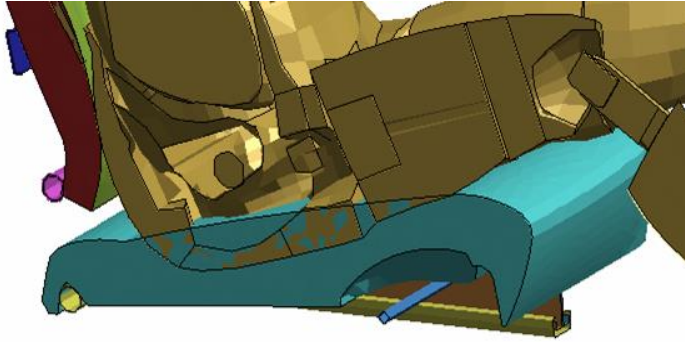


Combined Mechanisms

- One mechanism may be linked to another, e.g. seat squab to dummy pelvis. Then the dummy and seat can be dragged together in a single action.



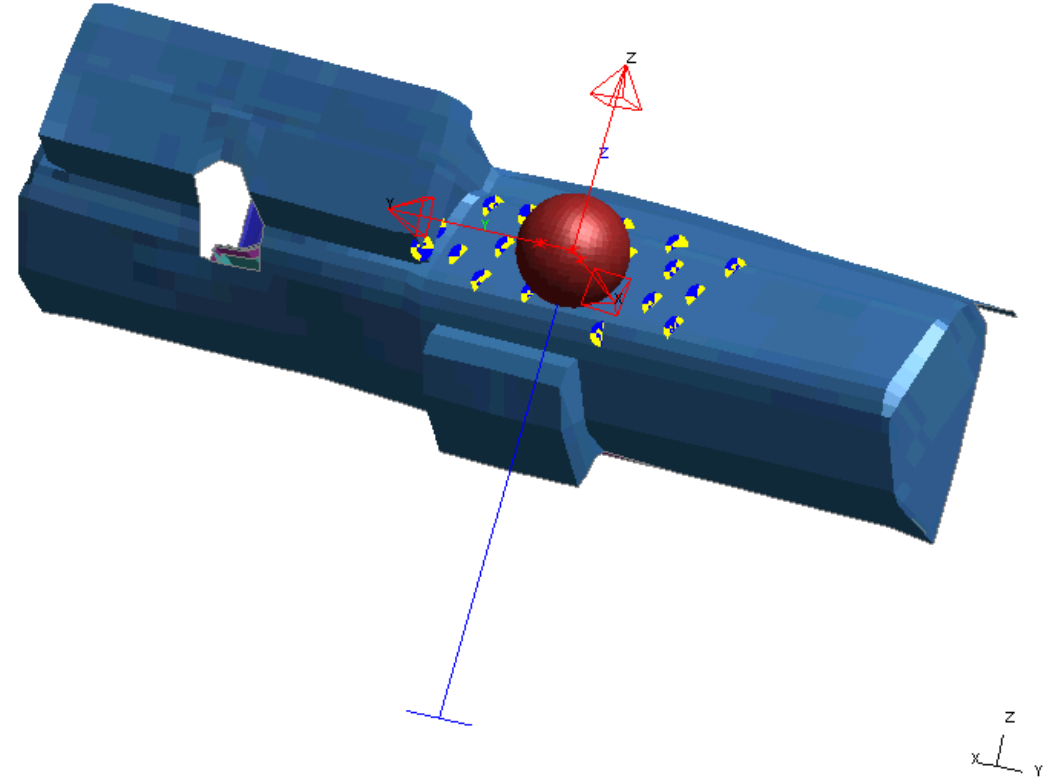
Seat Foam Compression



- Dummy has been positioned, penetrating the foam.
- “PRIMER” method depenetrates the top surface of the foam, and morphs the interior foam nodes to spread the strain evenly.
- “Dyna” method helps the user to set up an LS-DYNA analysis in which a rigid dummy compresses the seat foam until the correct H-point is achieved.

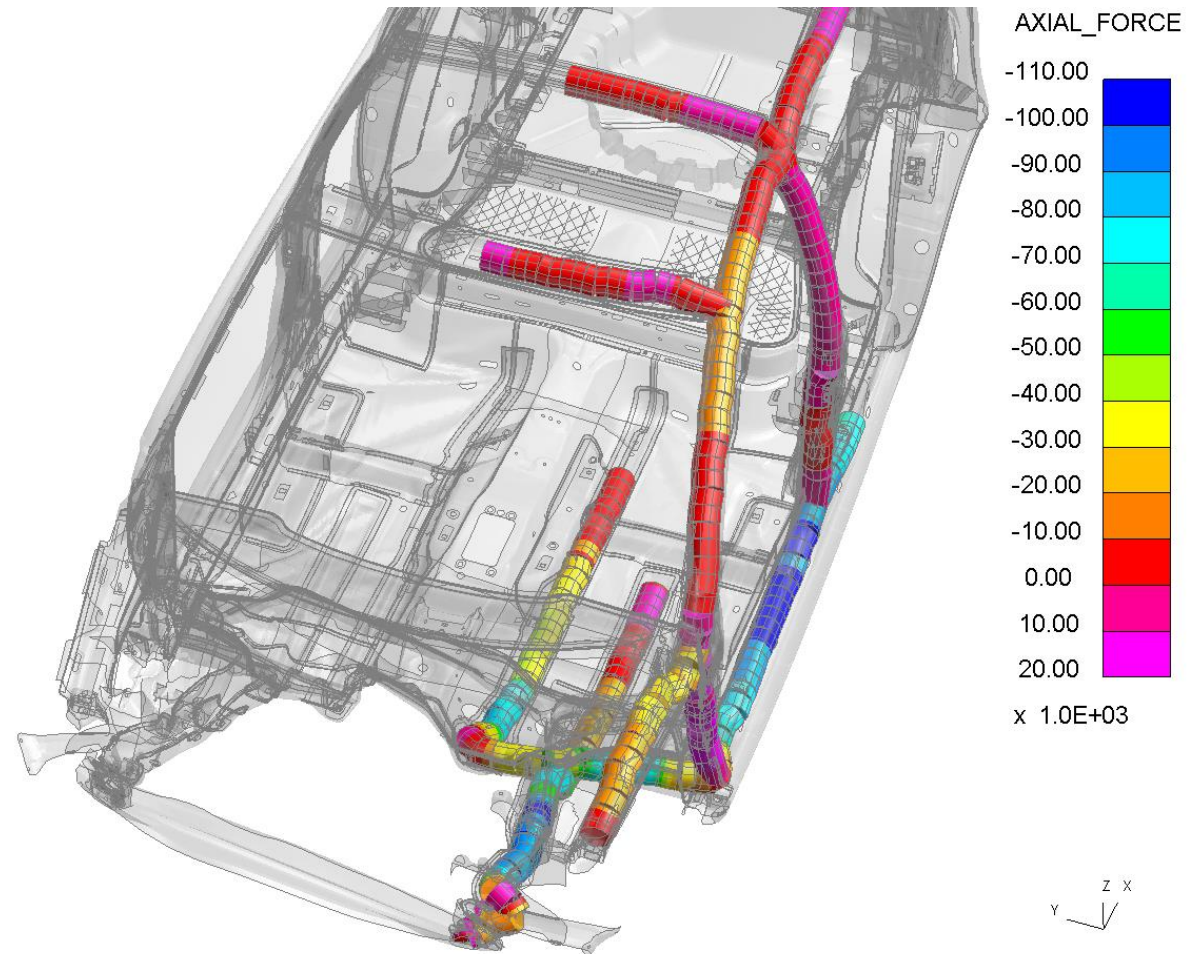
Instrument Panel Pendulum Impact

- Setup of models for ECER21.
- Both interactive and batch mode supported.
- Easy creation/modification of target points.
- Automated positioning and de-penetration.
- True and realigned approach angle supported.
- Data saved in model or written to csv target file.
- Multiple models (for multiple impact points) can be set up in a single operation



Load Paths

*DATABASE_CROSS_SECTION's can be grouped together in PRIMER to create load paths. These can be viewed and contoured within D3PLOT.

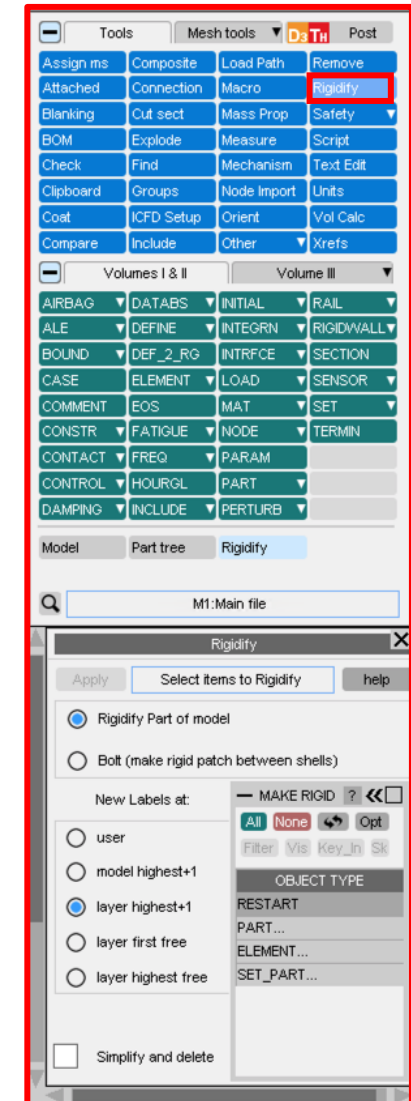


- Various tools are available within PRIMER for zone markup, targeting and multi-model setup:



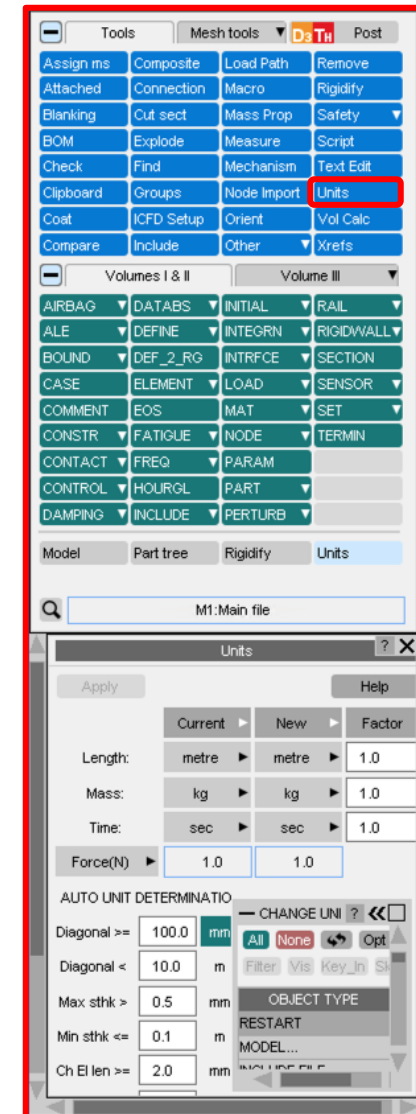
Rigidify

- Rigidify is used to rigidify selected parts of the model.
- Example uses include replacing complex deformable meshes with rigidified representations to reduce runtime.
- Rigid constraint clashes, tied contacts etc. are all taken care of during the rigidification process.



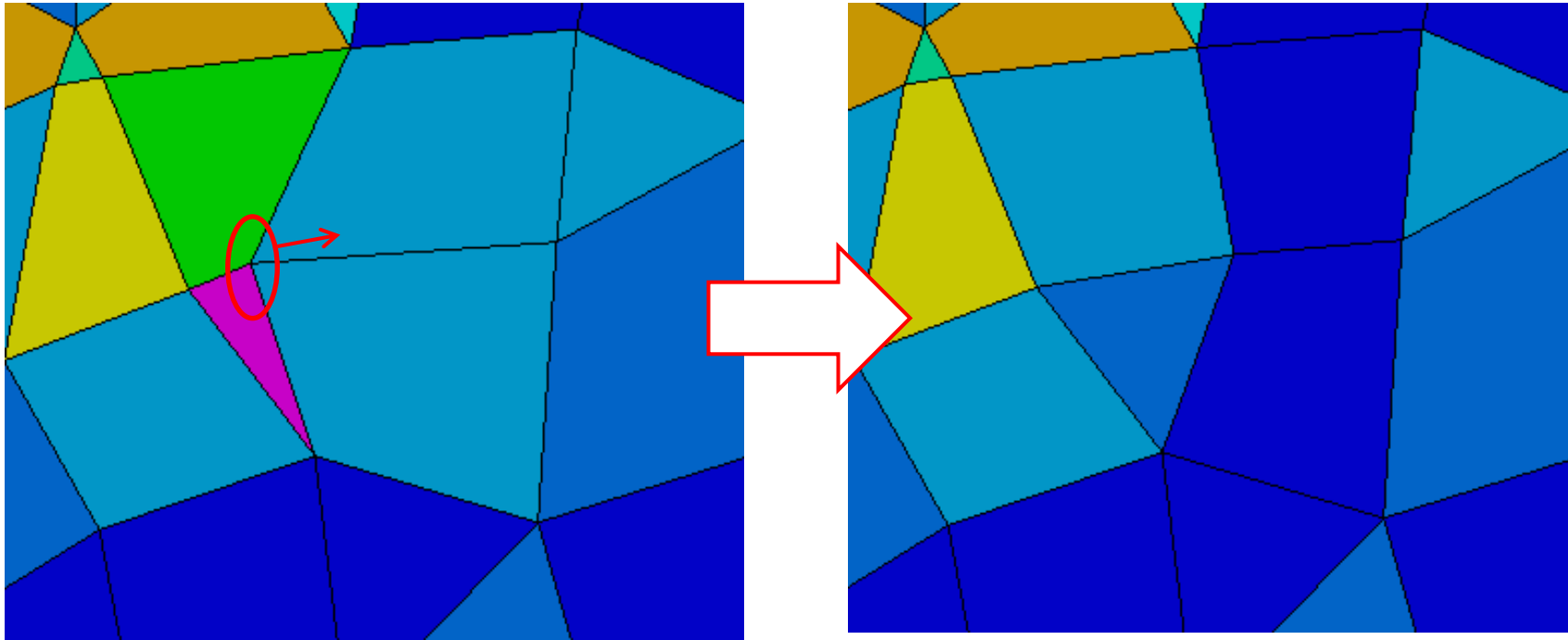
Units

- Model units can be easily modified in PRIMER.
- If you change the units system, all appropriate fields are updated automatically to account for the change.



Meshing Tools

- Various basic meshing tools are available in PRIMER:
- Mesh surface drag tool to improve mesh quality

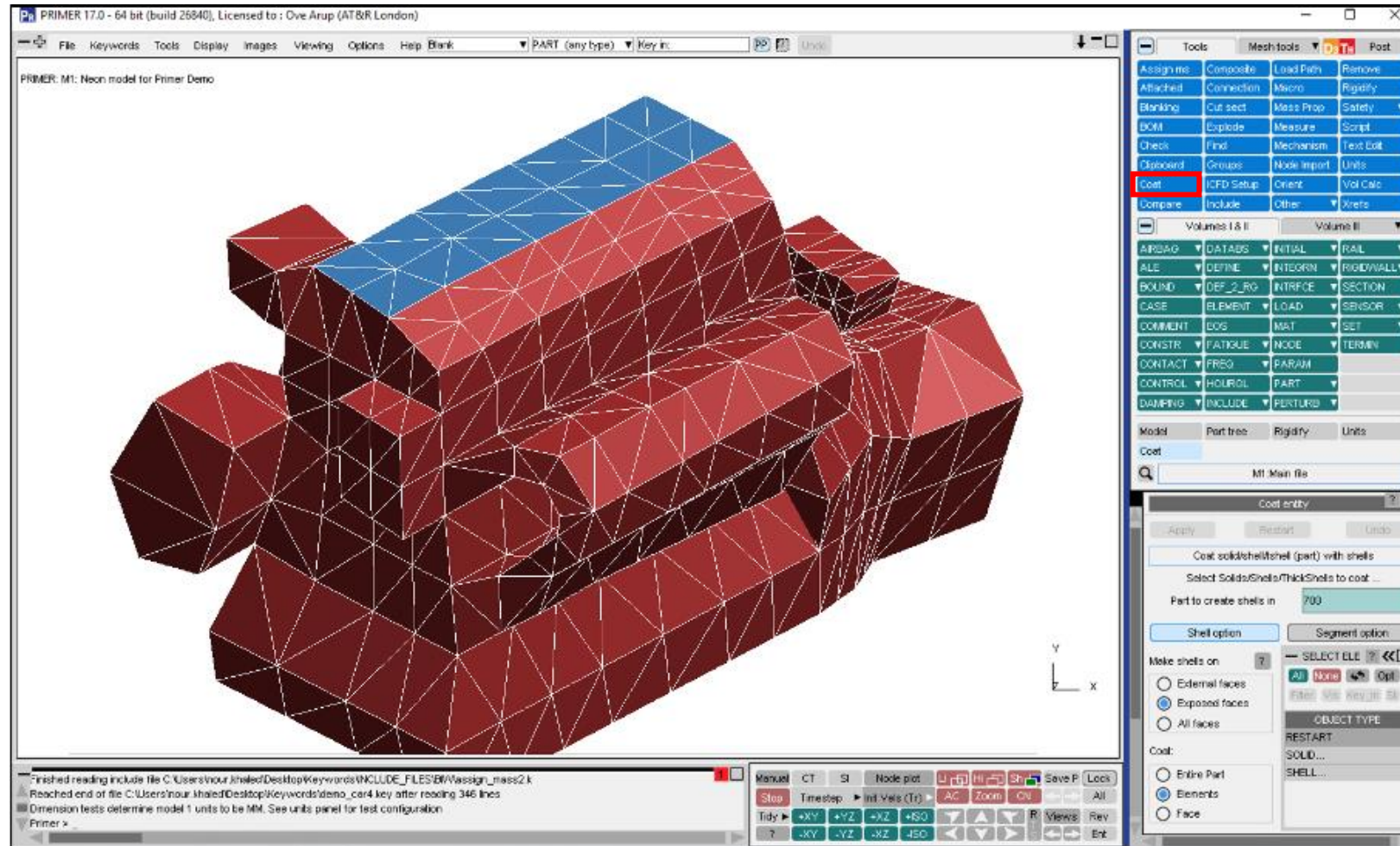


Meshing Tools

- Element mesh creation:
 - Standard shapes:
 - Plate
 - Cylinder
 - Block
 - Sphere/Hemisphere
 - Standard creation methods
 - Area
 - Extrude
 - Offset
 - Ruled
 - Standard modification tools:
 - Split
 - Combine
 - Align normals

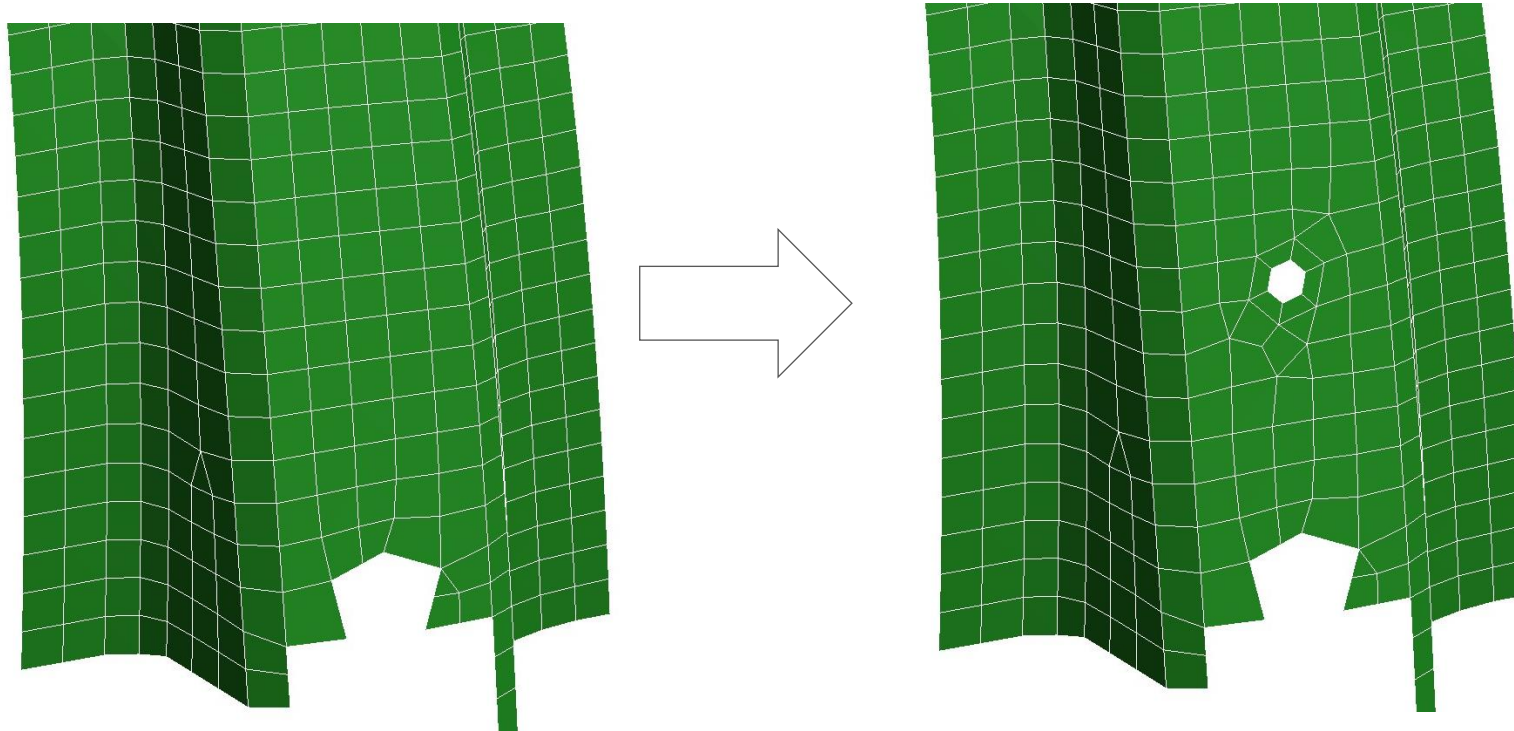
Meshing Tools

- Coat Part – coat solids/shells with shells/segments:



Meshing Tools

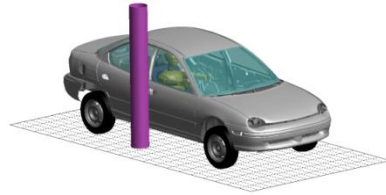
- Create/remove holes in shell mesh:



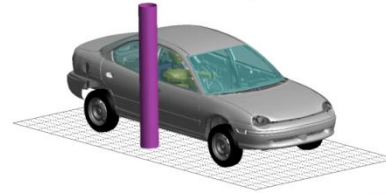
Barrier Positioning Tool

The supported crash test protocols are listed here. Small changes to these crash types (e.g. impact velocity) can be made by the user and saved easily:

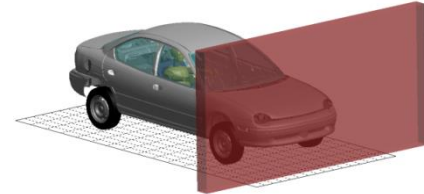
EuroNCAP:
Side Impact (Pole)



FMVSS 214P: Side
Oblique Impact (Pole)



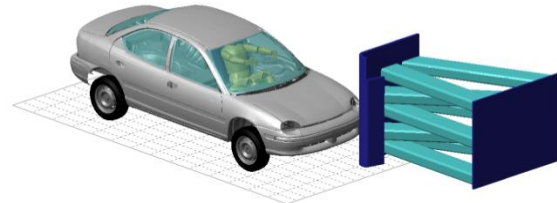
FMVSS, UN-ECE:
Frontal Impact (Rigid)



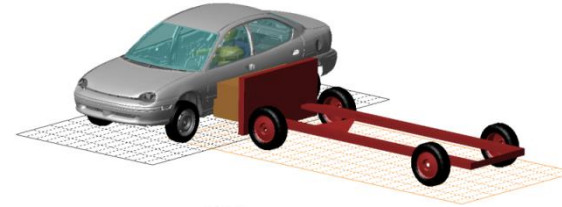
FMVSS 216A:
Roof Crush



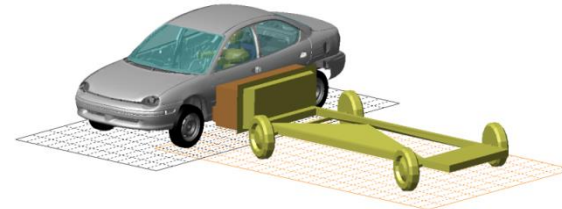
IIHS: Frontal
Small Overlap



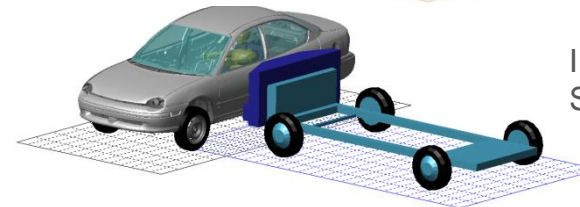
EuroNCAP, UN-ECE:
Side Impact (R-Point)



FMVSS 214D:
Angled Side Impact



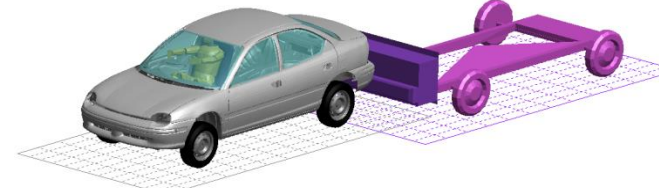
IIHS:
Side Impact (IRD)



EuroNCAP, FMVSS, IIHS,
UN-ECE: Frontal Impact (ODB)

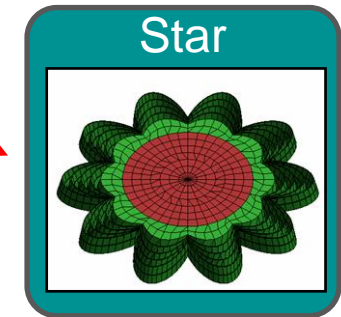
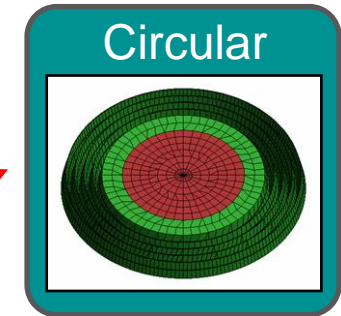
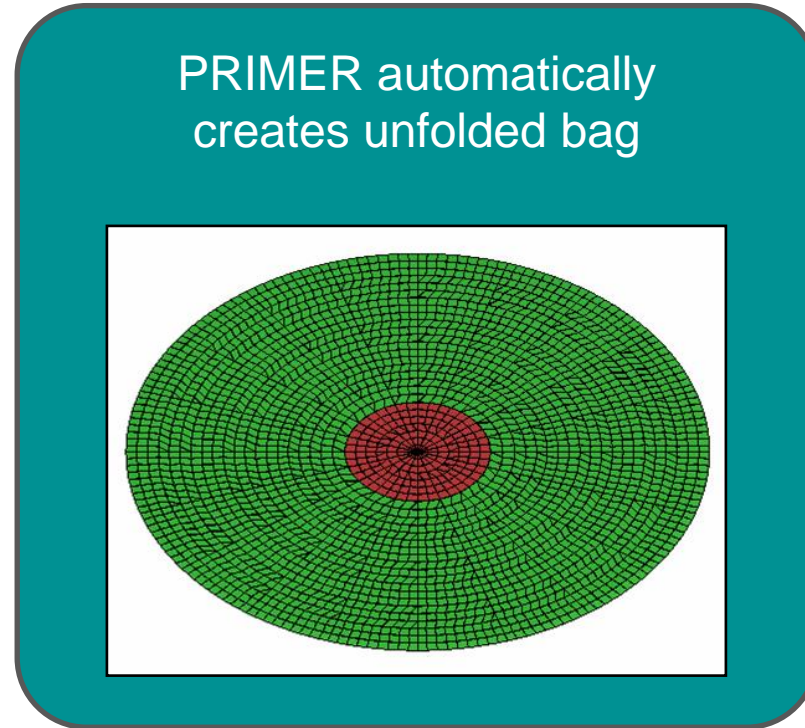
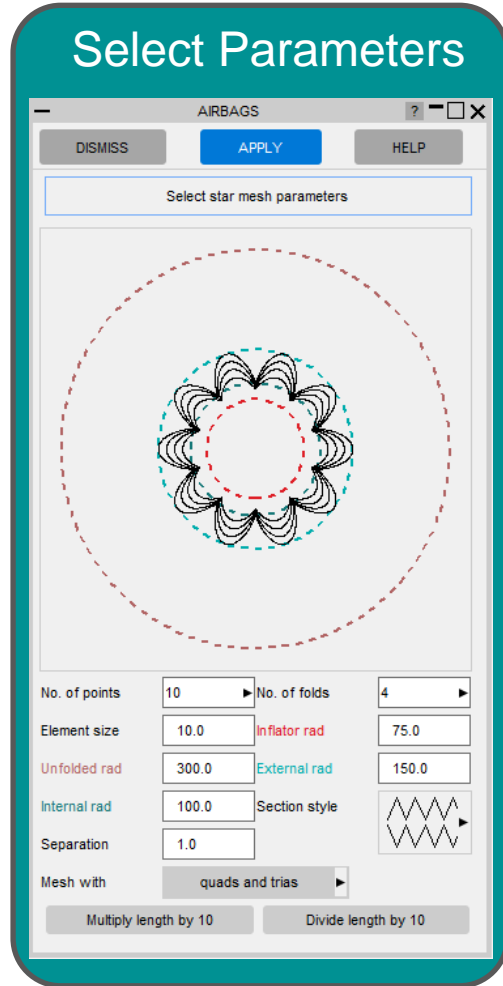


FMVSS 301R:
Rear Impact
(MDB)



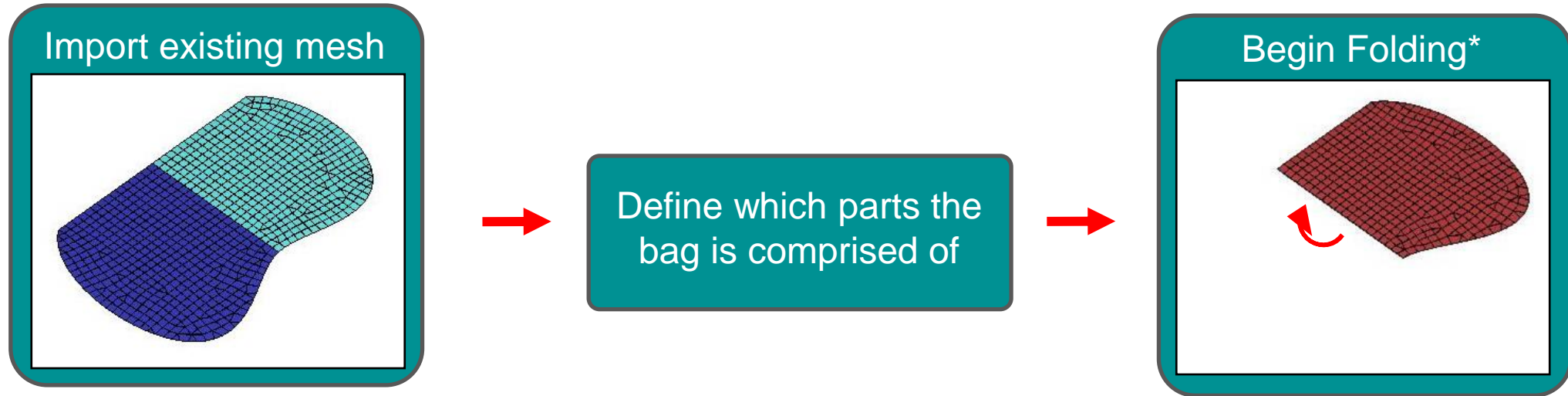
Airbag Folding

1. Automatically create 'star' or 'circular' type folded bag in 3 steps



Airbag Folding

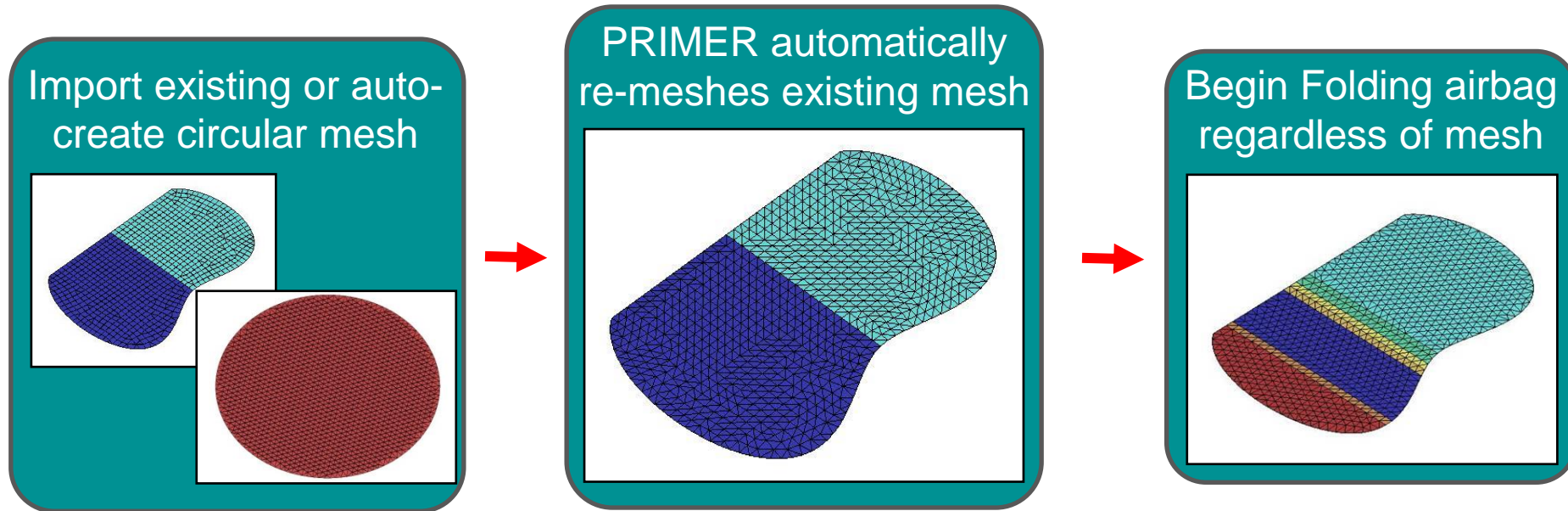
2. Fold an existing mesh using PRIMER's step-by-step functions



*If an existing mesh is used, the fold lines must be already defined

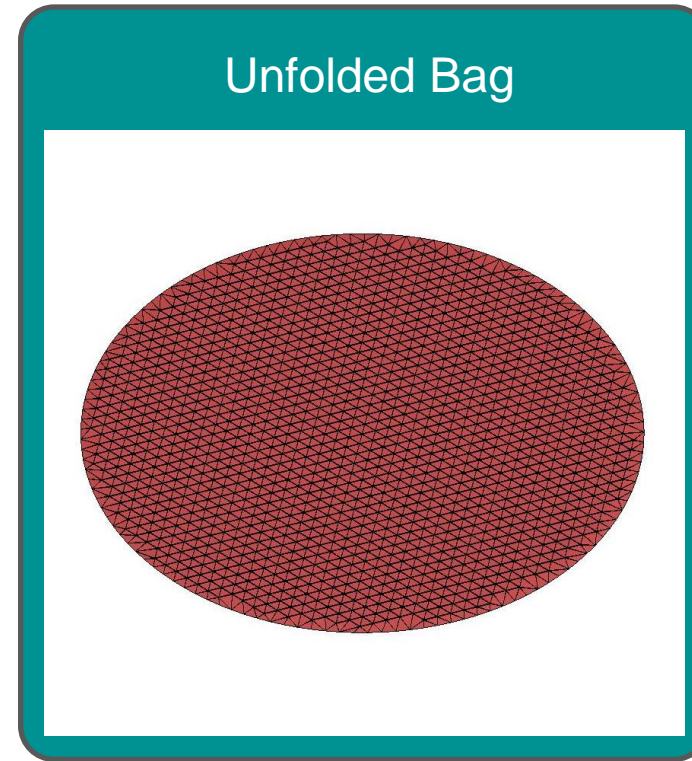
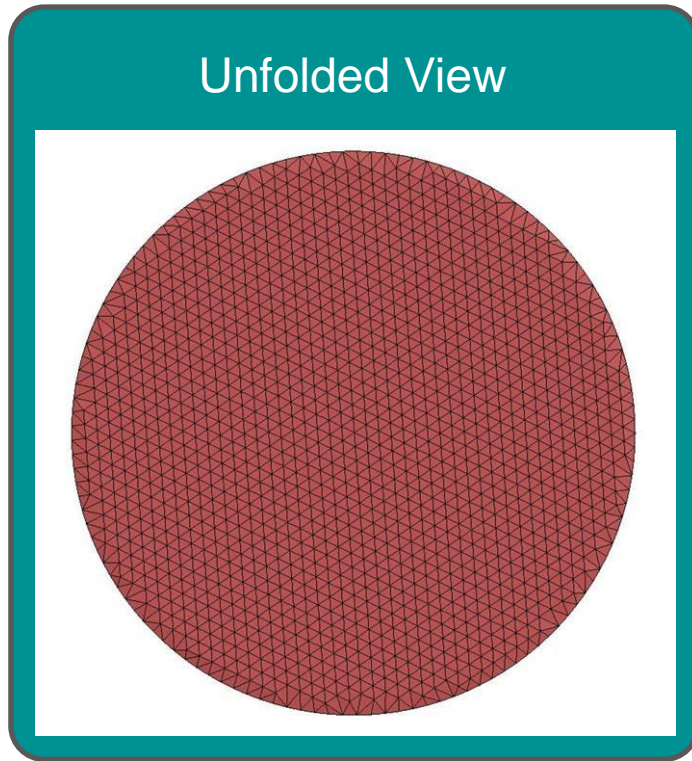
Airbag Folding

3. Use PRIMER's mesh-free function to fold an existing airbag shape or automatically create a circular airbag



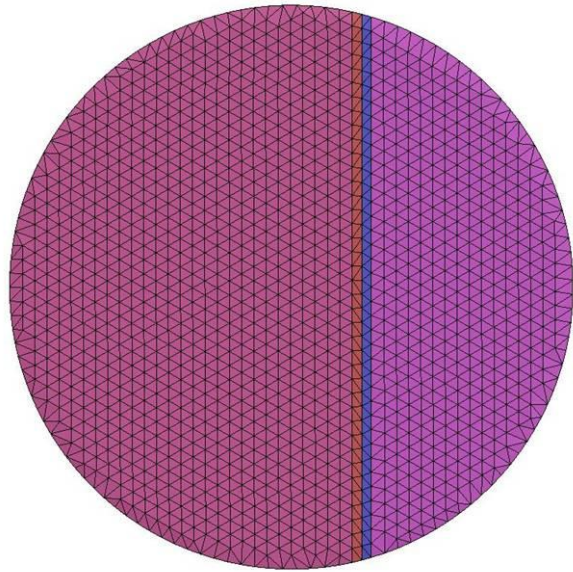
Mesh-free folding allows any airbag shape to be folded in any pattern. PRIMER takes care of creating fold lines and aligning the mesh

Mesh independent Folding Example

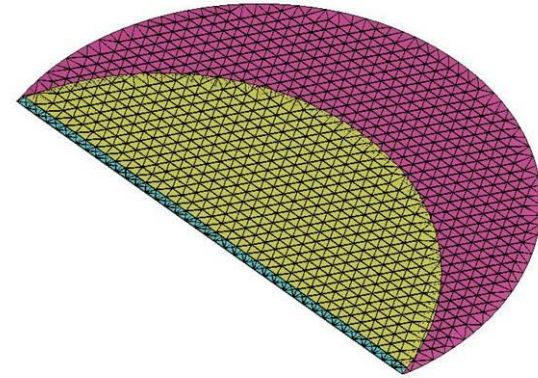


Mesh independent Folding Example

Unfolded View

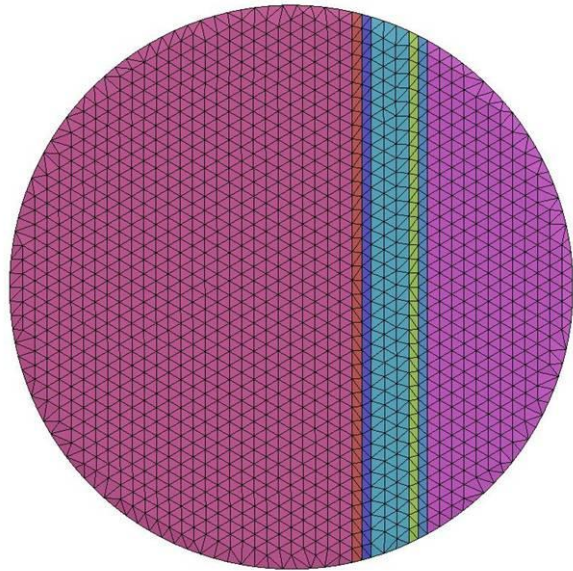


1st Fold: Forward

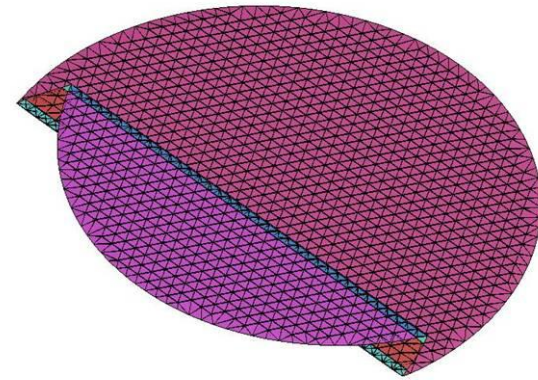


Mesh independent Folding Example

Unfolded View

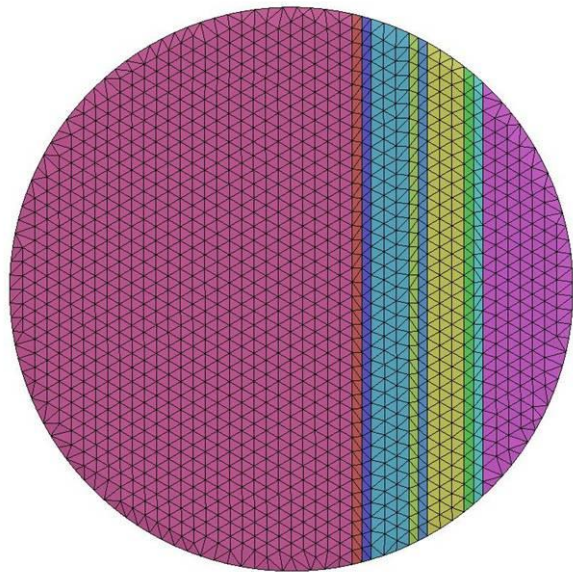


2nd Fold: Reverse

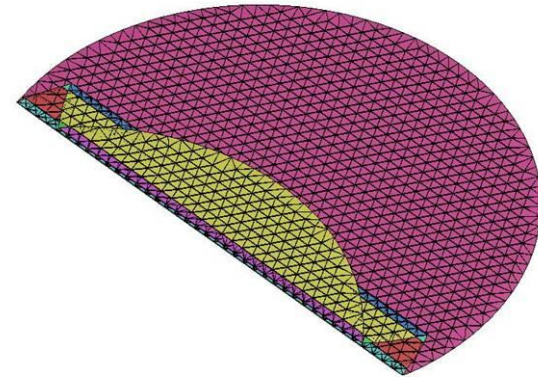


Mesh independent Folding Example

Unfolded View

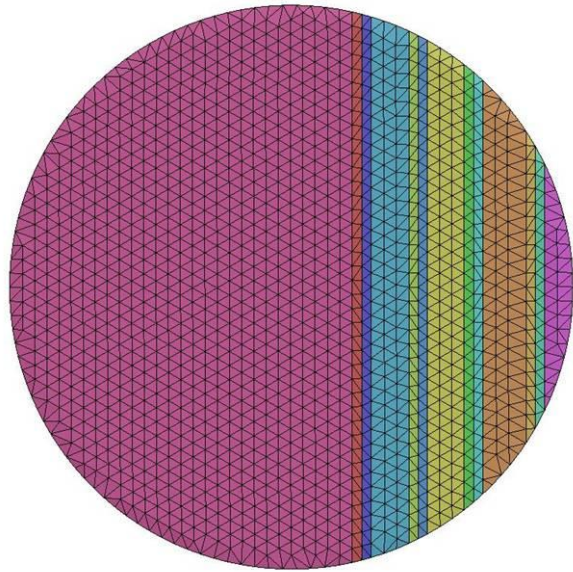


3rd Fold: Forward Subset

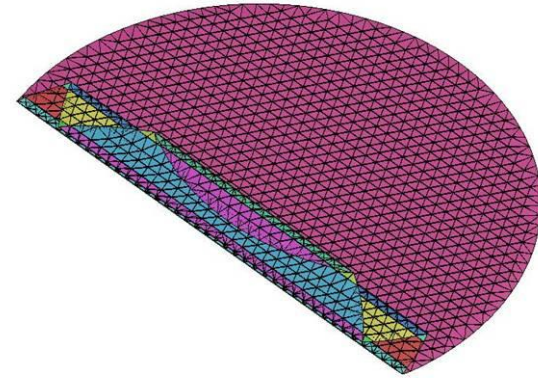


Mesh independent Folding Example

Unfolded View

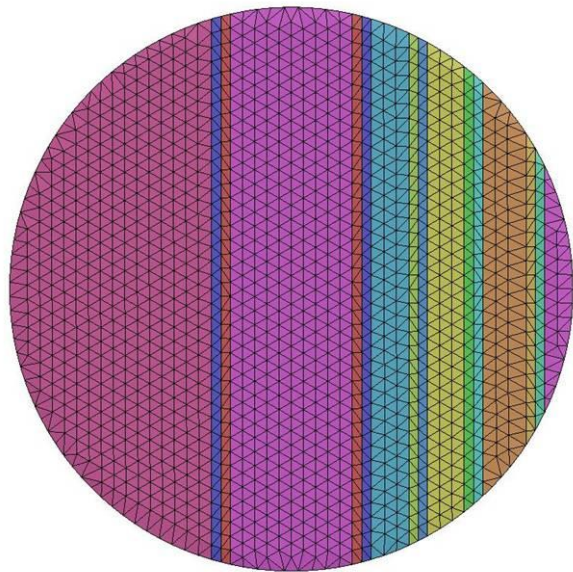


4th Fold: Reverse Subset

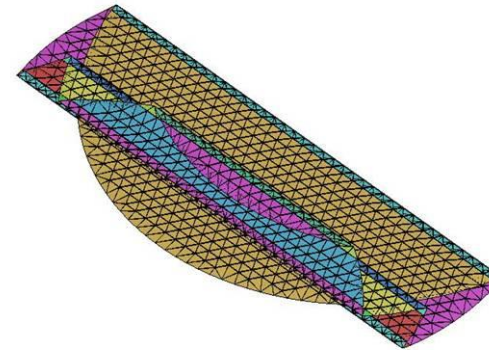


Mesh independent Folding Example

Unfolded View

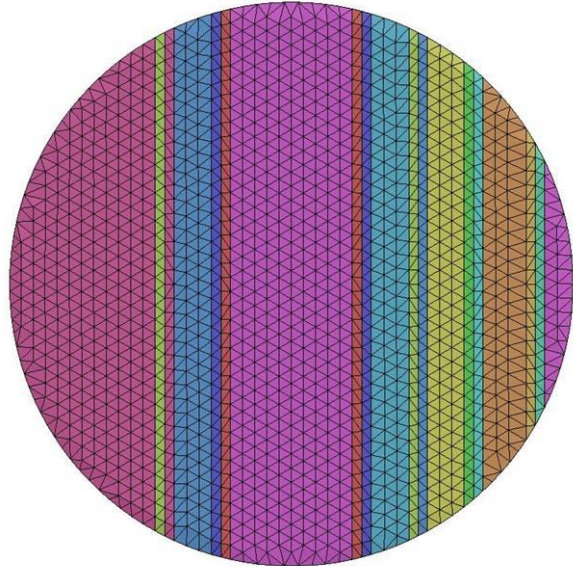


5th Fold: Reverse

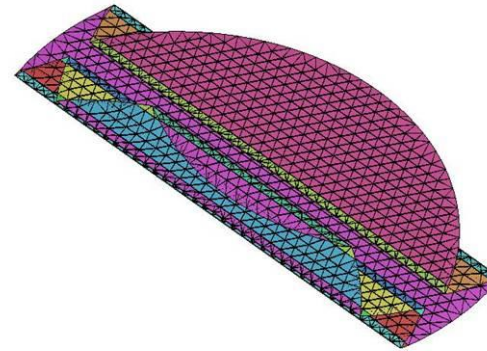


Mesh independent Folding Example

Unfolded View

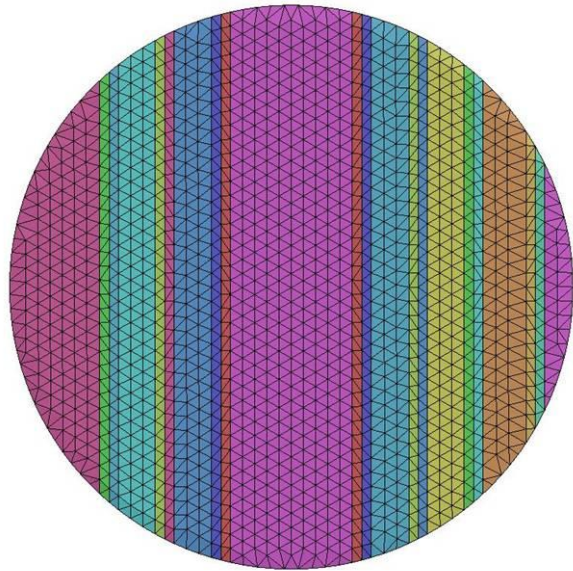


6th Fold: Forward

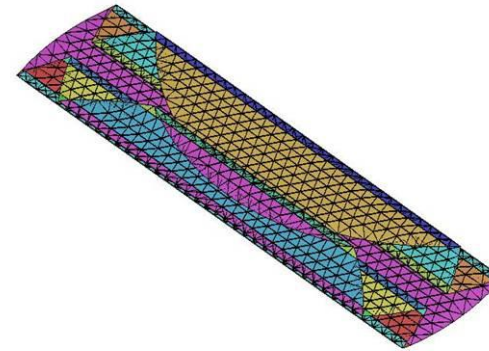


Mesh independent Folding Example

Unfolded View

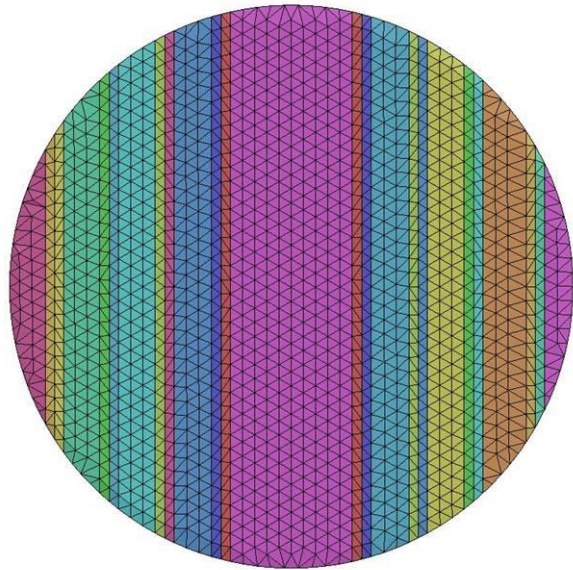


7th Fold: Reverse Subset

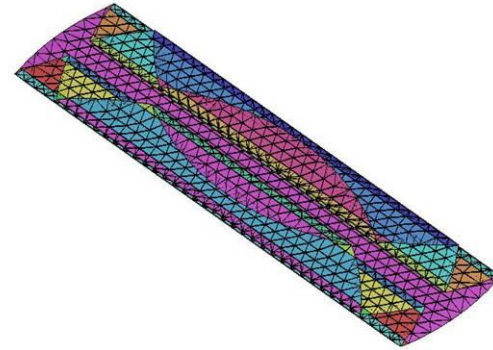


Mesh independent Folding Example

Unfolded View

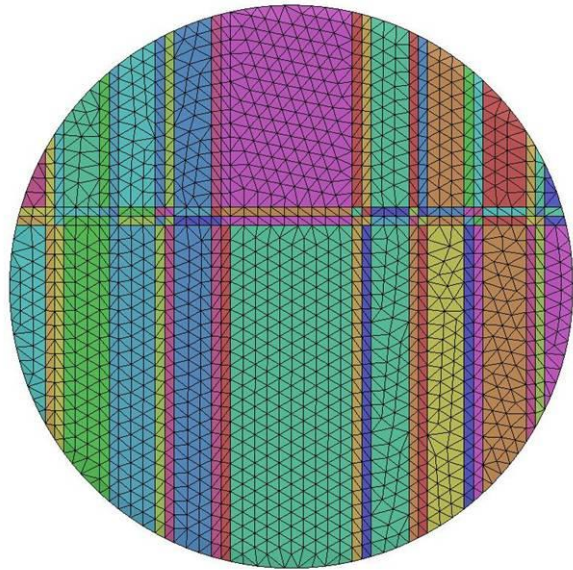


8th Fold: Forward Subset

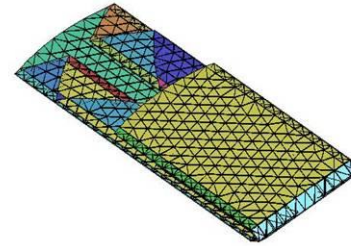


Mesh independent Folding Example

Unfolded View

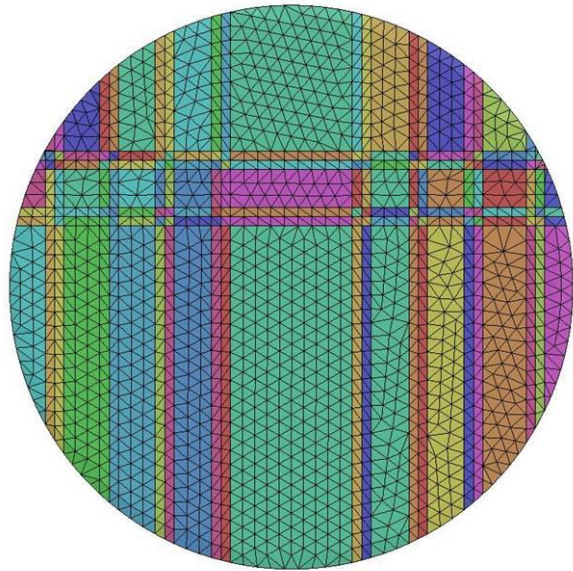


9th Fold: 90deg Reverse

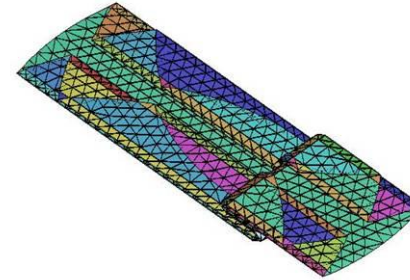


Mesh independent Folding Example

Unfolded View

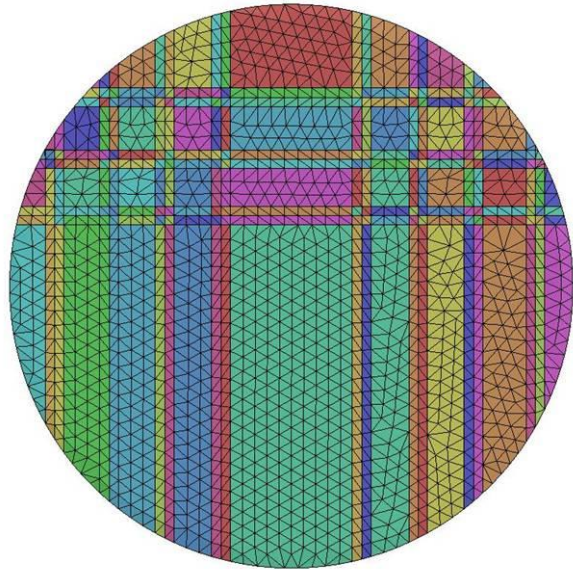


10th Fold: 90deg Forward

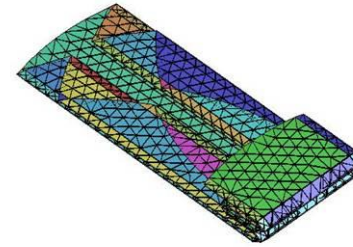


Mesh independent Folding Example

Unfolded View

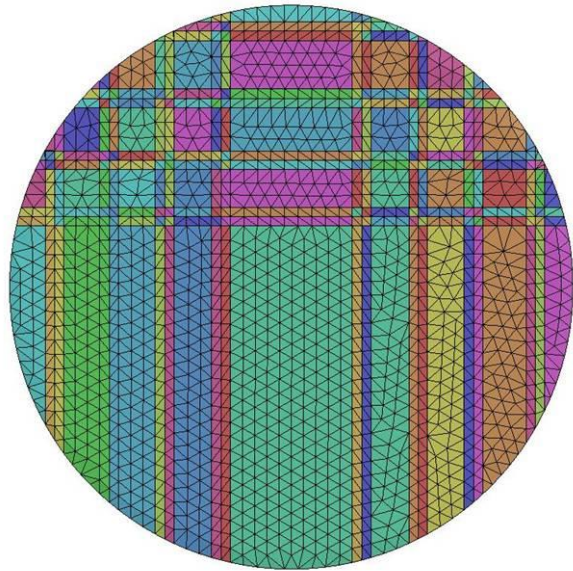


11th Fold: 90deg Rev. Subset



Mesh independent Folding Example

Unfolded View

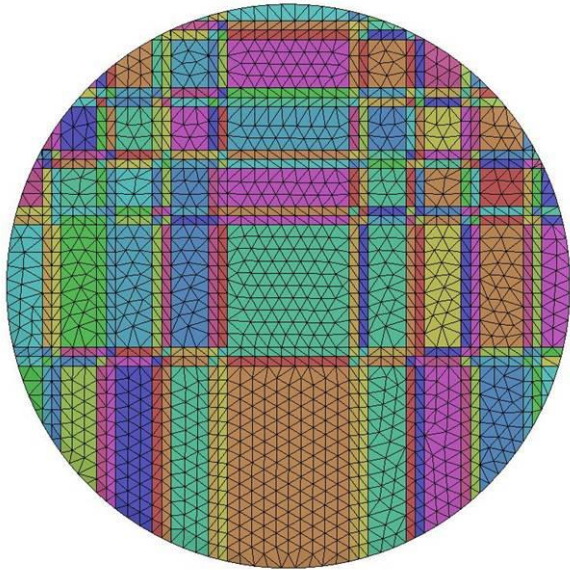


12th Fold: 90deg Fwd. Subset

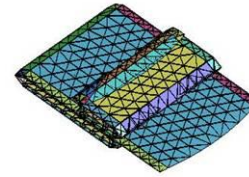


Mesh independent Folding Example

Unfolded View

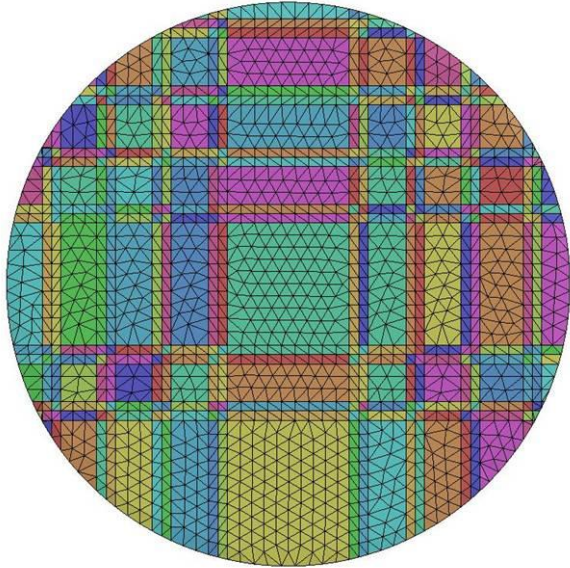


13th Fold: 90deg Forward



Mesh independent Folding Example

Unfolded View

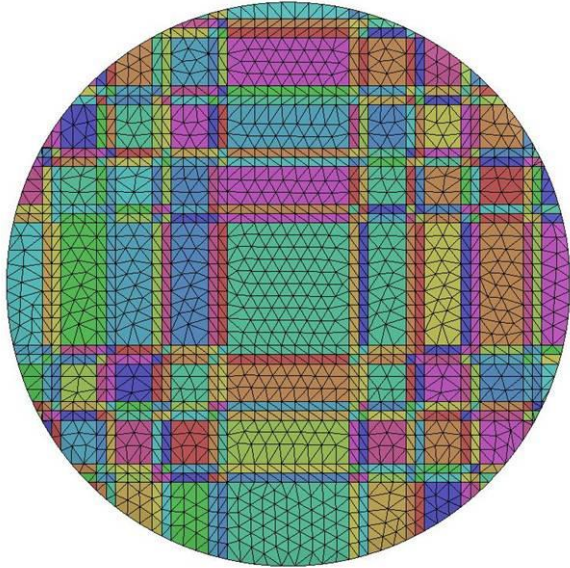


14th Fold: 90deg Reverse

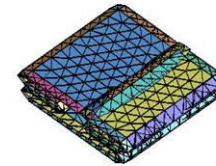


Mesh independent Folding Example

Unfolded View

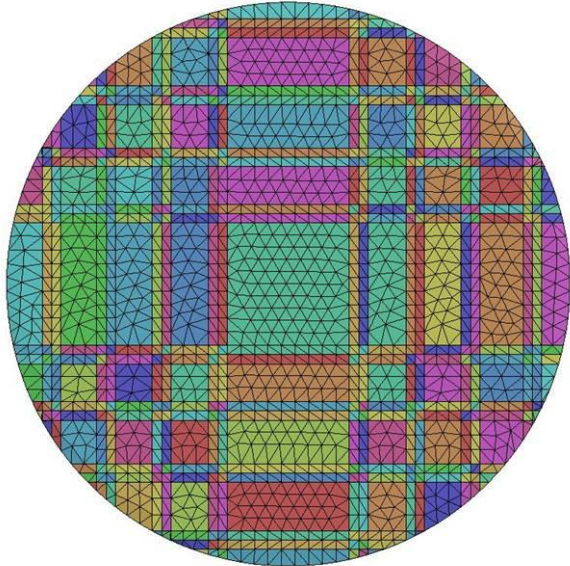


15th Fold: 90deg Fwd. Subset



Mesh independent Folding Example

Unfolded View

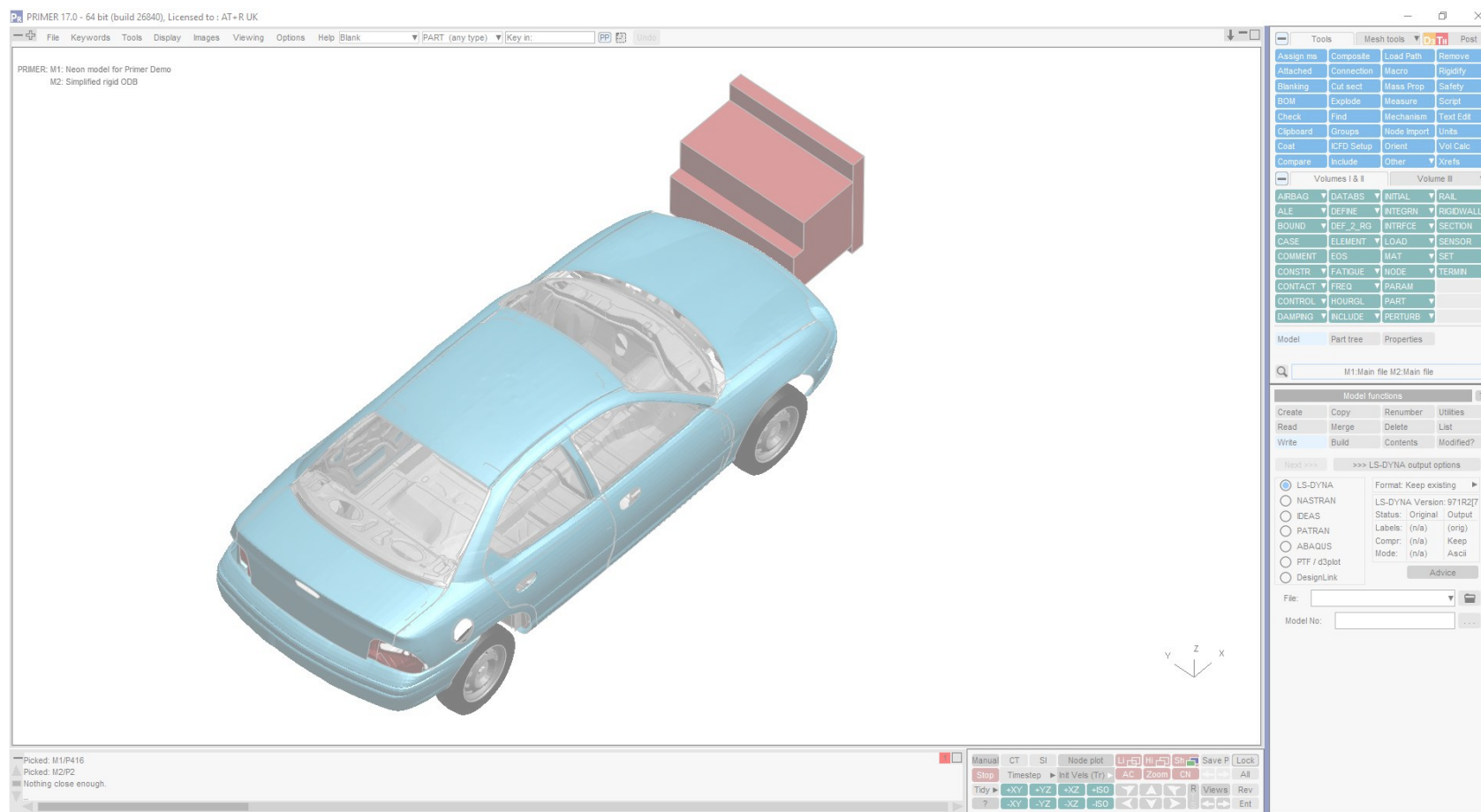


15th Fold: 90deg Rev. Subset



Oasys PRIMER

A training course for new users



Contact Information



www.arup.com/dyna

For more information please contact us:

UK

The Arup Campus
Blythe Valley Park
Solihull
B90 8AE
United Kingdom

T: +44 121 213 3399
dyna.support@arup.com

China

Arup China
39/F-41/F Huaihai Plaza
1045 Huaihai Road (M)
Xuhui District, Shanghai
200031
China

T: +86 21 3118 8875
china.support@arup.com

India

Arup India Pvt Ltd
10th floor, Western Dallas
Center
Plot no. 83/1, Knowledge City
Rai Durg, Hyderabad-500032
Telangana, India

T: +91 40 44369797 / 98
india.support@arup.com

USA West

Oasys Ltd
c/o 560 Mission Street Suite 700
San Francisco
CA 94105
United States

T: +1 415 940 0959
us.support@arup.com

or your local Oasys distributor