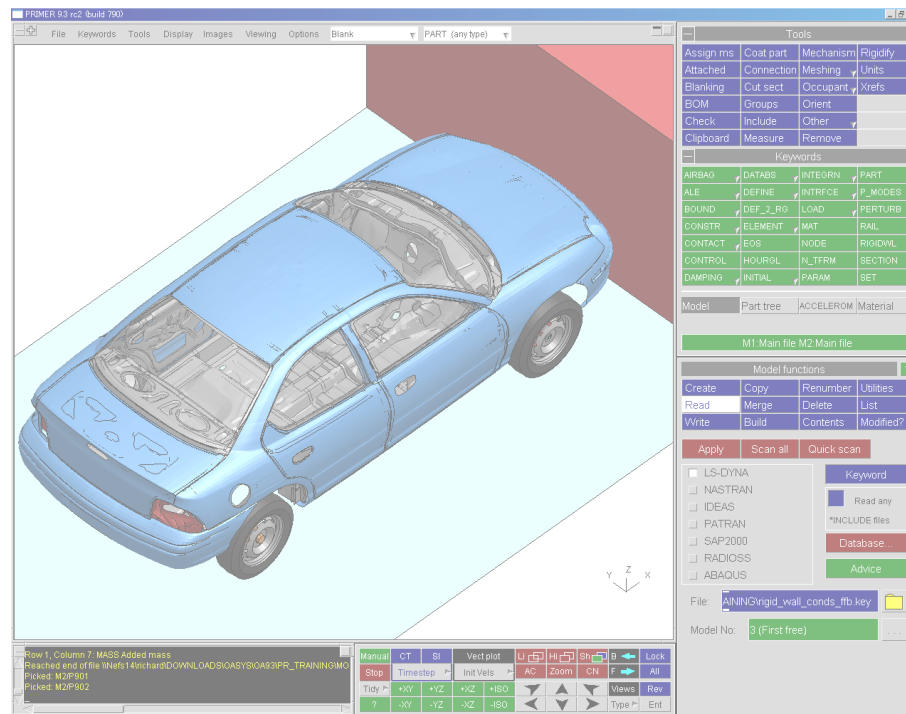


Oasys PRIMER

A Training Course for New Users

September 2014



- 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

- 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”
- See next slides for recommended preference settings.

Before starting this course

Note for course teacher: Preferences

primer*graphics_type: Opengl

primer*maximise: TRUE **Needed for cp-file operation – screen picking etc**

primer*initial_plot_mode: SHADED

primer*overlay_mode: FREE

primer*overlay_colour: BLACK

primer*text_colour: BLACK

primer*background_colour: WHITE

These settings give a good initial appearance after reading a model

primer*contour_levels: 13

primer*MASS_drawn: OFF

primer*ascii_file_format: UNIX

primer*dyna_output_version: 970v6763 (or 971R2)

primer*GLSTAT_file: ON

Needed only if you want to try the automatic model build from database/template system

primer*set_template_dir: C:\DATA\DEMO\NEW_NEON_MODEL\TEMPLATE_FILES

primer*set database dir: C:\DATA\DEMO\NEW NEON DEMO MODEL\DATABASE FILES

primer*model_timestep_check: ON

primer*model_min_timestep: 1.e-6

primer*model_max_timestep: 1.e-5

primer*model_added_mass_check: ON

primer*model_max_added_mass_percent: 5.0

primer*part_added_mass_check: ON

primer*part_max_added_mass_percent: 50

primer*spotweld_part_max_added_mass: .005

primer*element_overlap_check: ON

Model checking options (cont...)

Note for course teacher: Preferences (cont)

primer*rigid_body_merge_check: ON
primer*rigid_body_merge_max_separation: 200
primer*rigid_body_continuity_check: ON
primer*rigid_body_min_elem: 1
primer*nodal_rigid_body_size_check: ON
primer*maximum_nodal_rigid_body_size: 200

Max separation of merged rigid bodies; rigid body continuity; size of nodal rigid bodies

primer*part_element_include_check: ON
primer*element_node_include_check: ON
primer*element_quality_checks: OFF
primer*spotweldbeam_length_check: ON

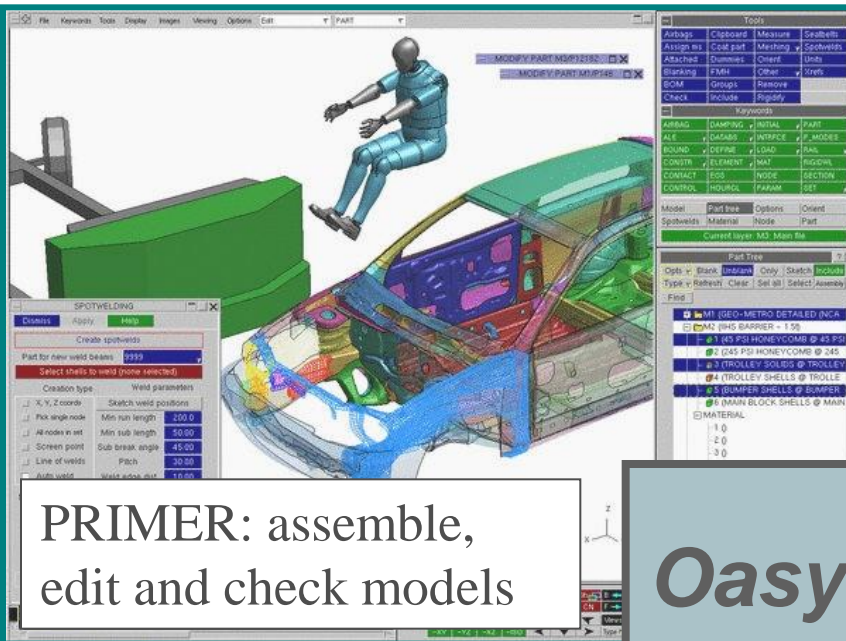
primer*spotweldbeam_min_length: 0.3
primer*spotweldbeam_max_length: 10.0
primer*spotweldbeam_max_total_length: 15.0

“looser” parameters than normal to allow for poor quality flanges

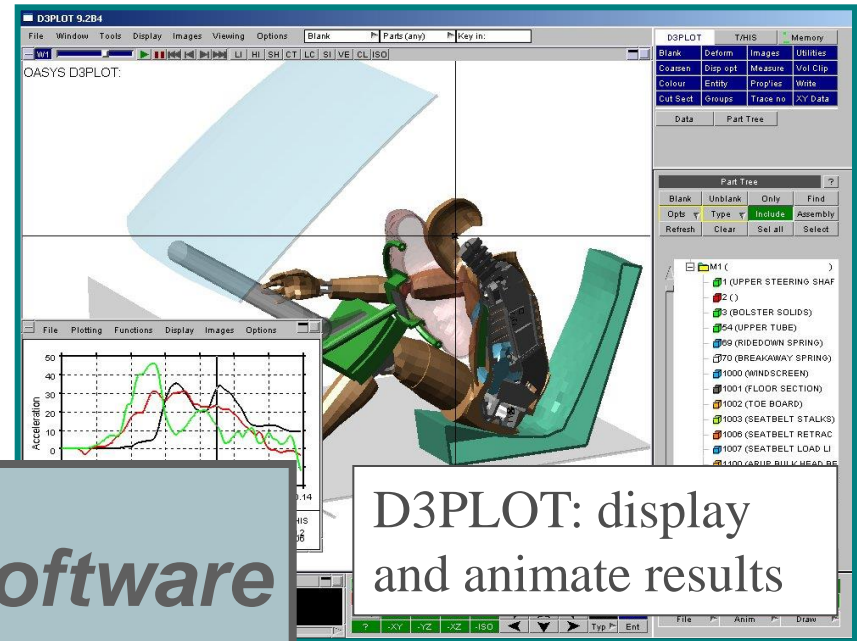
primer*spotweldbeam_distance_check: ON
primer*spotweldbeam_min_distance: 15
primer*spotweldbeam_panel_check: ON
primer*spotweldbeam_max_panels: 4
primer*spotweld_contact_check: ON
primer*database_node_check: OFF

primer*mat24_strain_check_limit: 100.0
primer*mat24_required_table_curve_separation_factor: 0.01

Special checks for Mat type 24

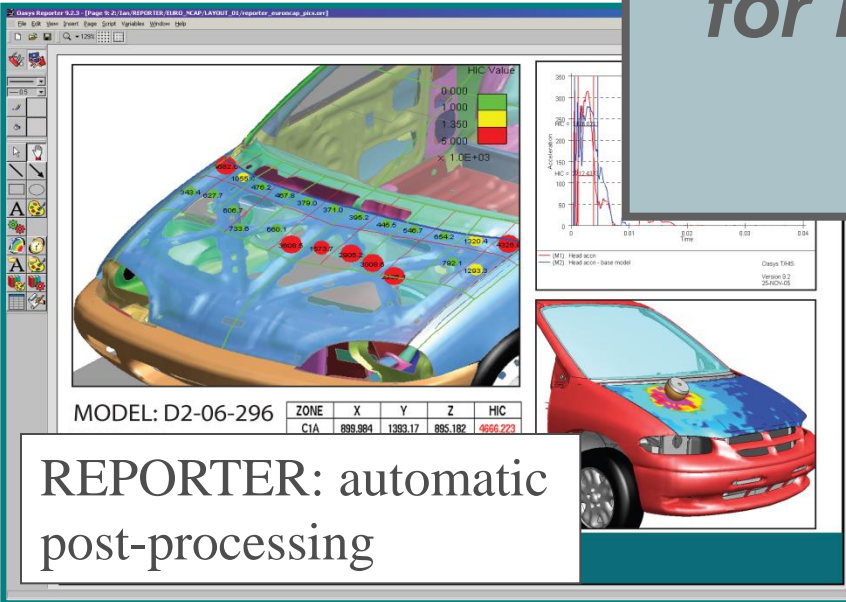


PRIMER: assemble,
edit and check models

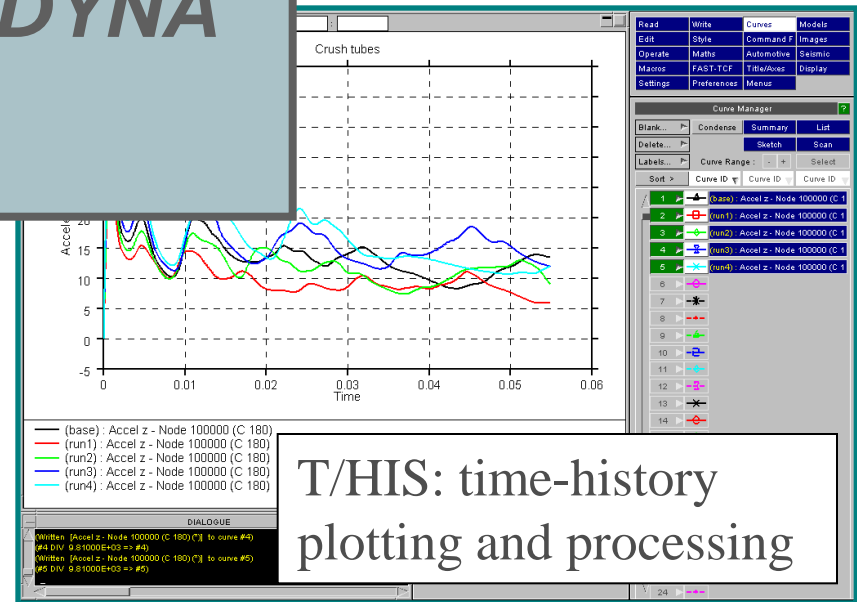


D3PLOT: display
and animate results

Oasys software for LS-DYNA



REPORTER: automatic
post-processing

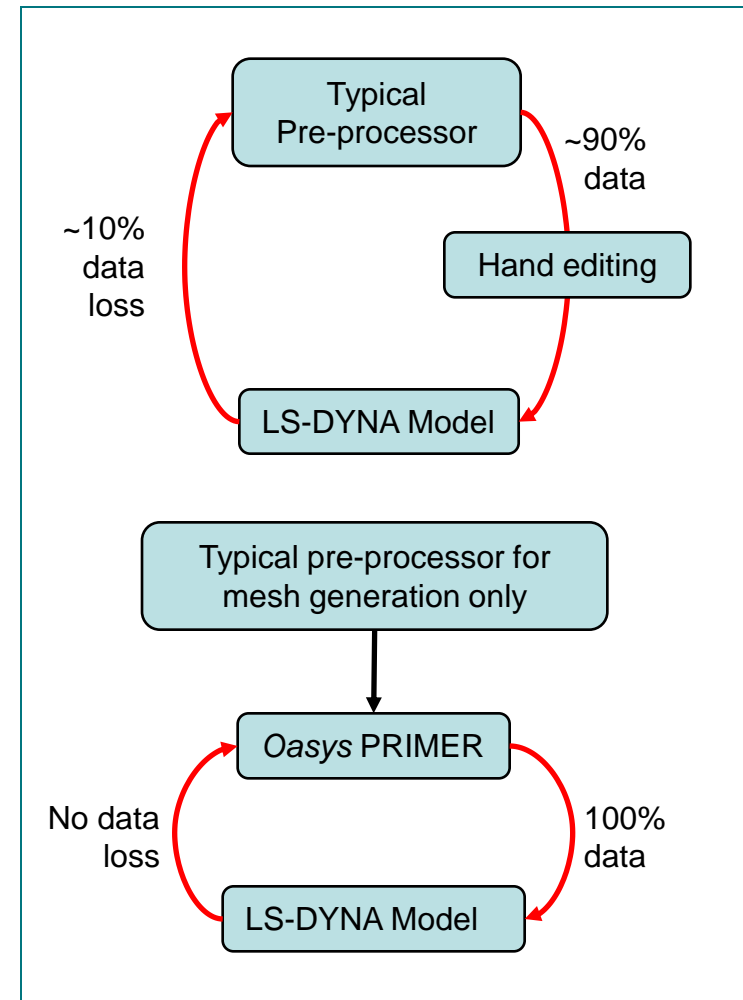


T/HIS: time-history
plotting and processing

Purpose of Oasys PRIMER

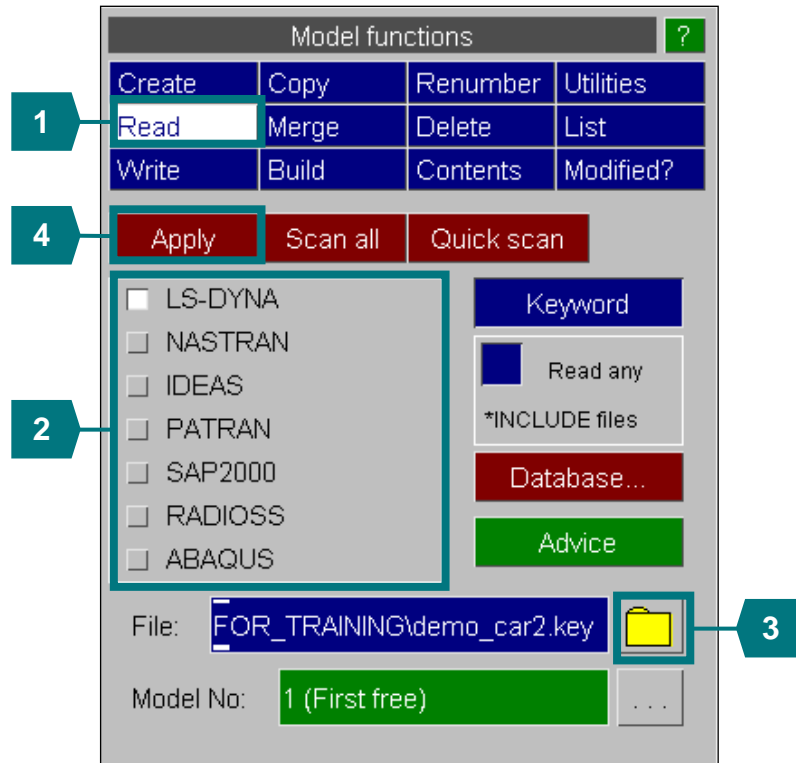


- Oasys PRIMER is a specialised pre-processor used only for LS-DYNA models.
- PRIMER's modeling tools and comprehensive error checking can reduce preparation and debugging time.
- PRIMER is fully compatible with the latest version of LS-DYNA, so no information is lost when reading, or writing out model 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 and controlling the display

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.

Graphics Area

Area within which graphics are drawn.

Dialogue & List area

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

Viewing & Drawing Commands

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

Tools

Provides access to PRIMER specific functions.

Keywords

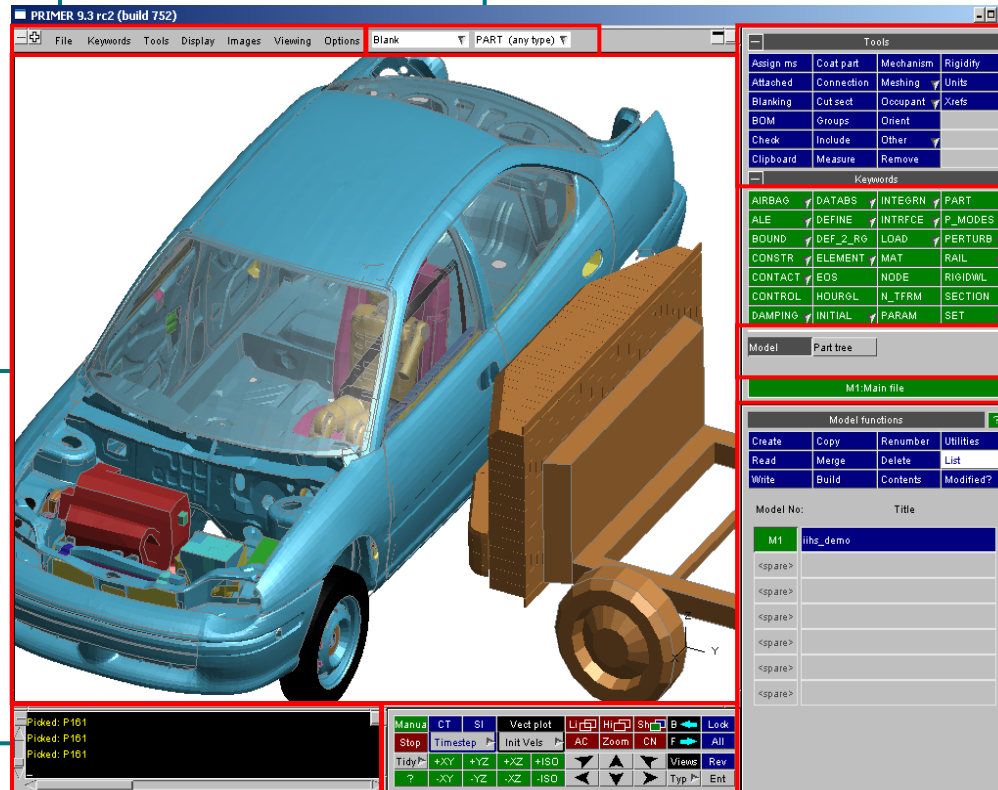
This provides access to the Keywords that are supported and can be edited by PRIMER.

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.

Current menu panel

"Current Menu Panel" Displays the menu for the option currently selected by the menu tabs.



Setting menu and text size

Controlling image/menu/text sizes:

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.

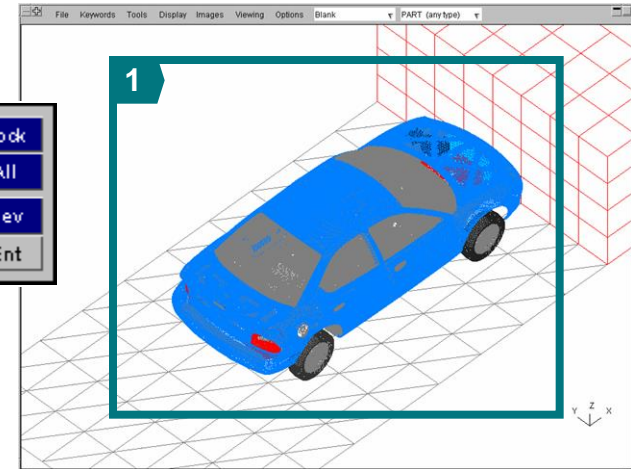
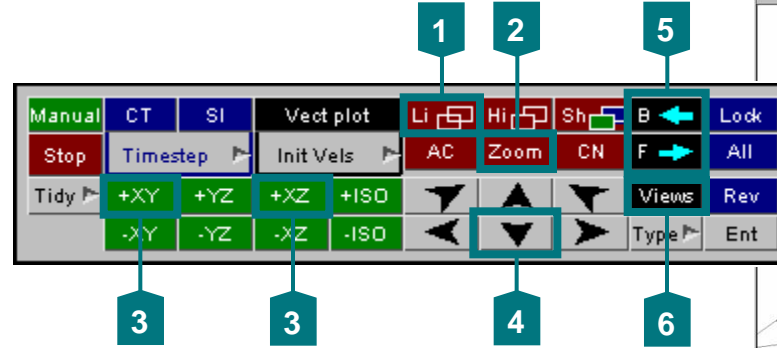
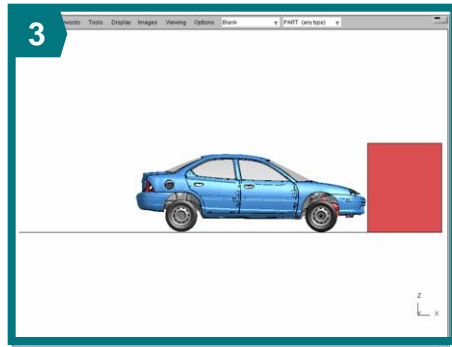
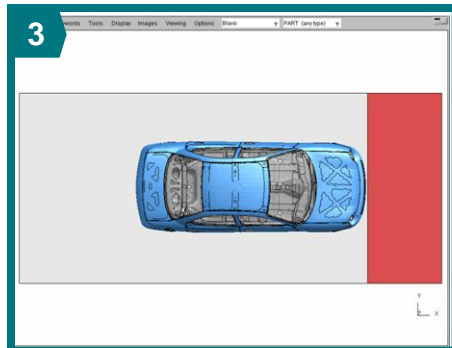
The image displays three screenshots from the Oasys PRIMER 9.3 rc2 (build 1655) software interface. The top-left screenshot shows the 'Options' menu with 'Menu attributes' highlighted. The top-right screenshot shows the main application window with a 'Display Factor = 1.4' label pointing to the menu area. The bottom-left screenshot shows the 'Menu Attributes' dialog box with 'Display Factor' set to 0.90. The bottom-right screenshot shows the 'Display Factor' dialog box with 'Display Factor' set to 1.50. A note at the bottom right states: 'Note: if Display Factor is made too large, there will not be room for text – you will see blank buttons.'

Display Factor = 1.4

Display Factor = 0.9

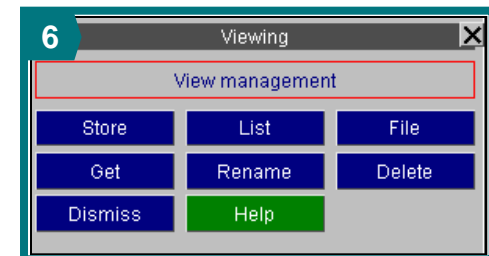
Display Factor = 1.50

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



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

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

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 = measure node-to-node.

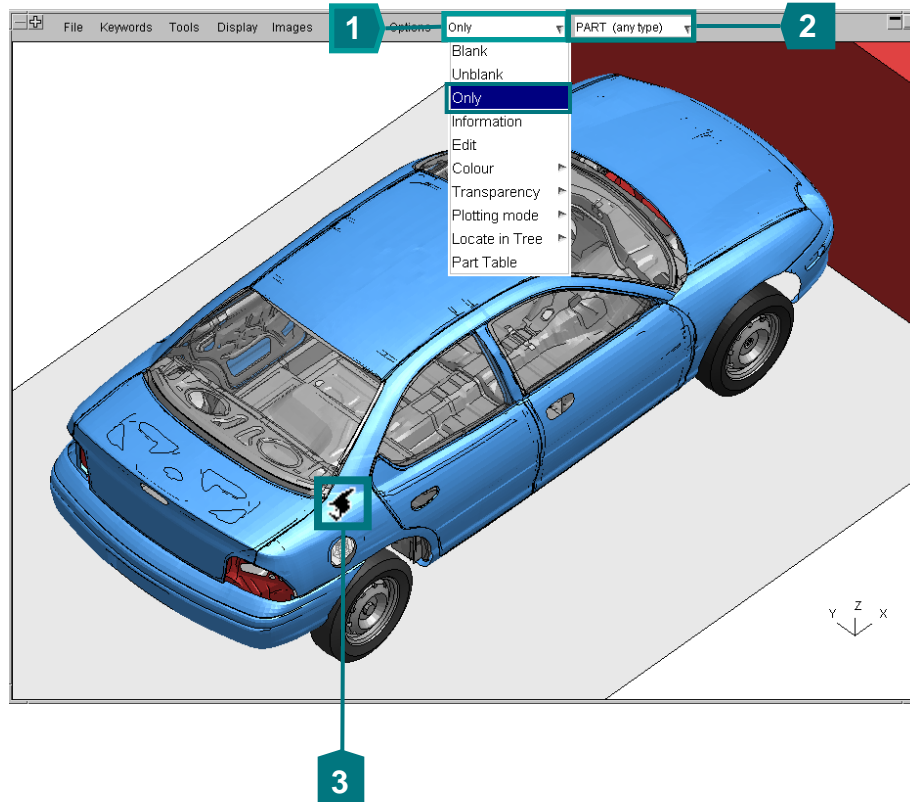
Q = return to **Q**uick-pick.

Z = **Z**oom (drag across rectangular area).

+ and **-** = zoom in and out.

T = tidy all floating menus; **C** = close all floating menus.

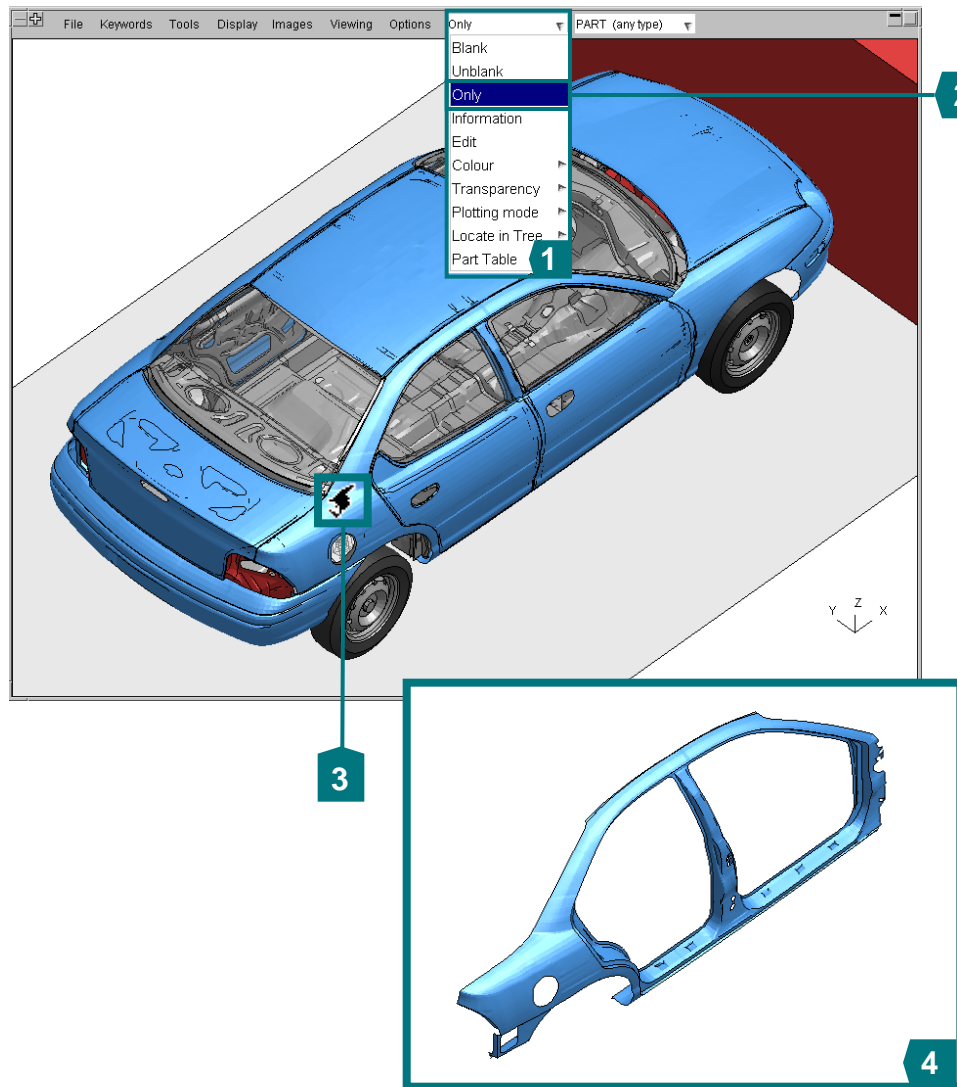
I = iconise all floating menus.



Quick-Pick

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

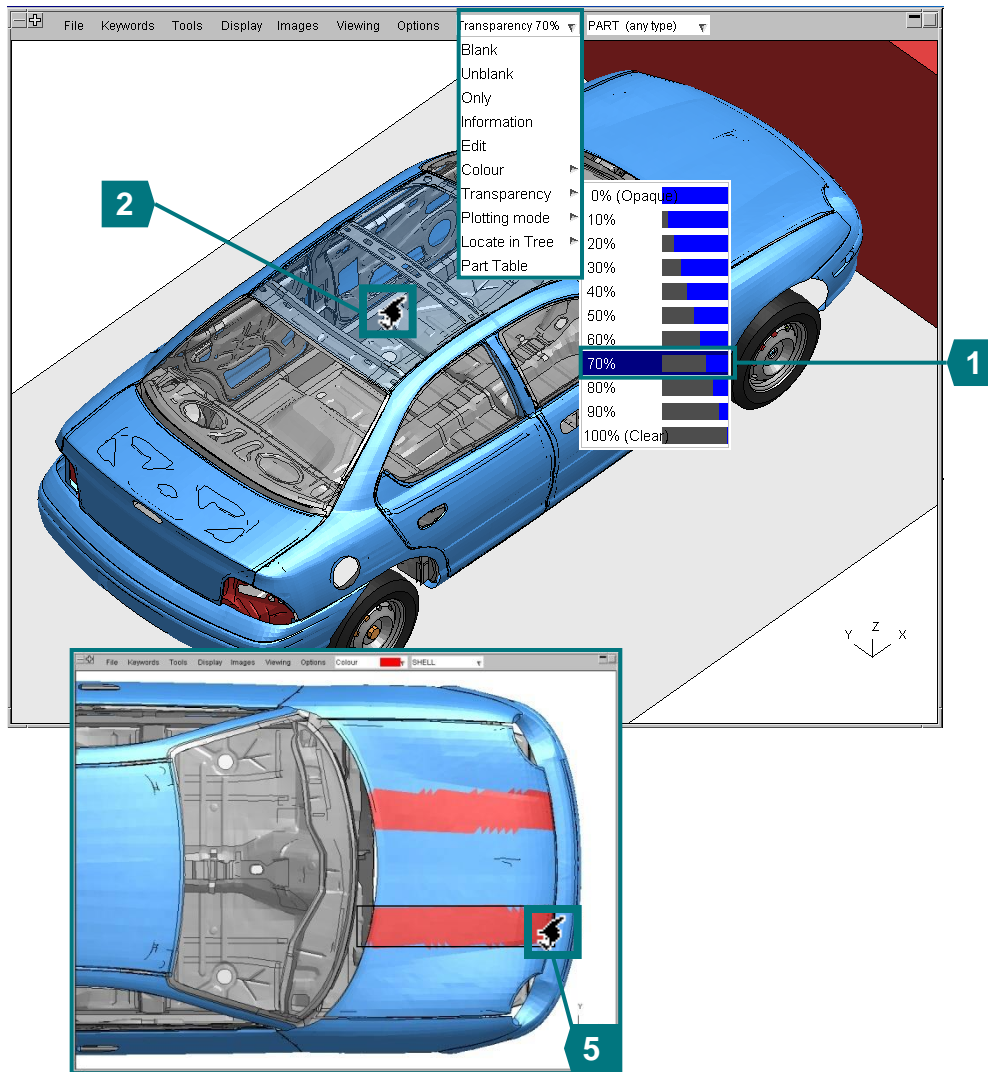
1. This menu decides what action will happen when you click on an entity. Leave this set to Blank.
2. This menu decides what type of entity you will select – leave it set to Part
3. Click on any part – it disappears (is blanked).
4. Drag across an area – all the parts in that area are blanked.
5. Middle-click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



Quick-Pick

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 **reverse** the blanking
6. Press **U** to **unblank** 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.



More Quick-Pick

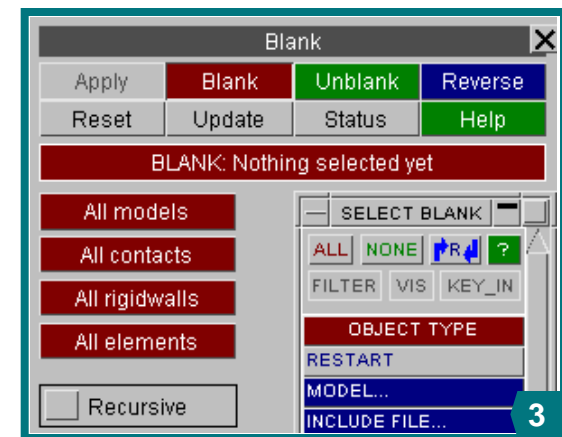
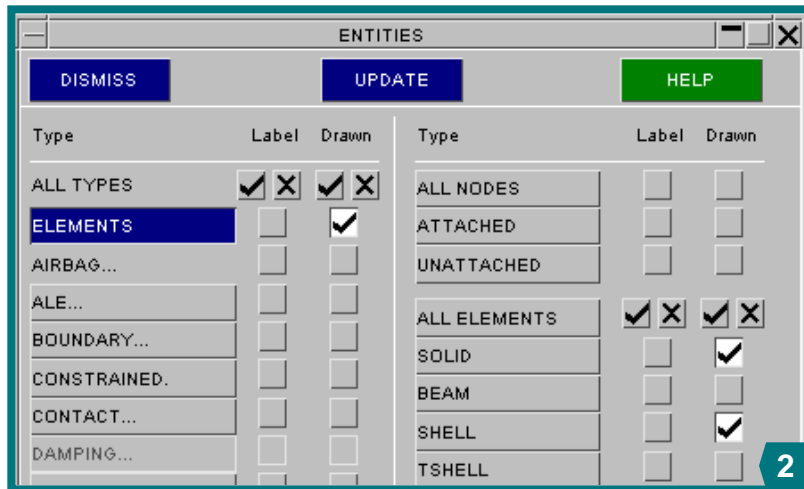
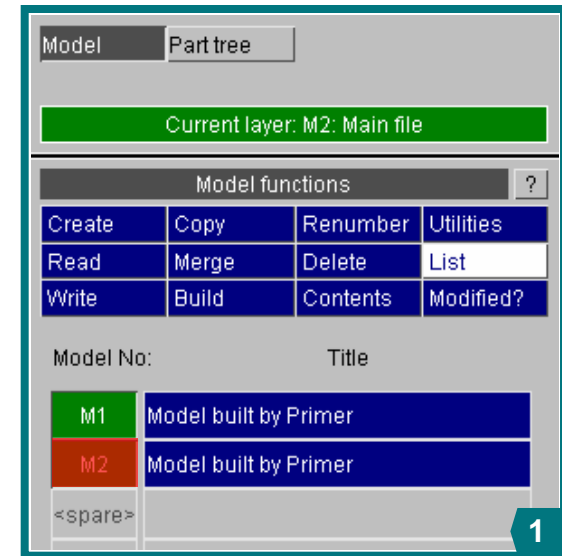
Try other quick-pick combinations:

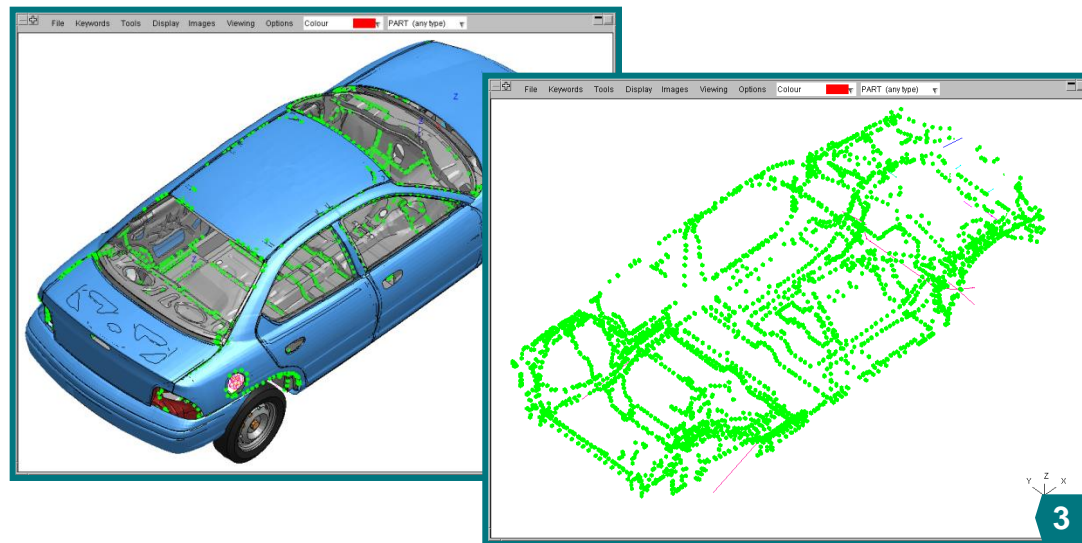
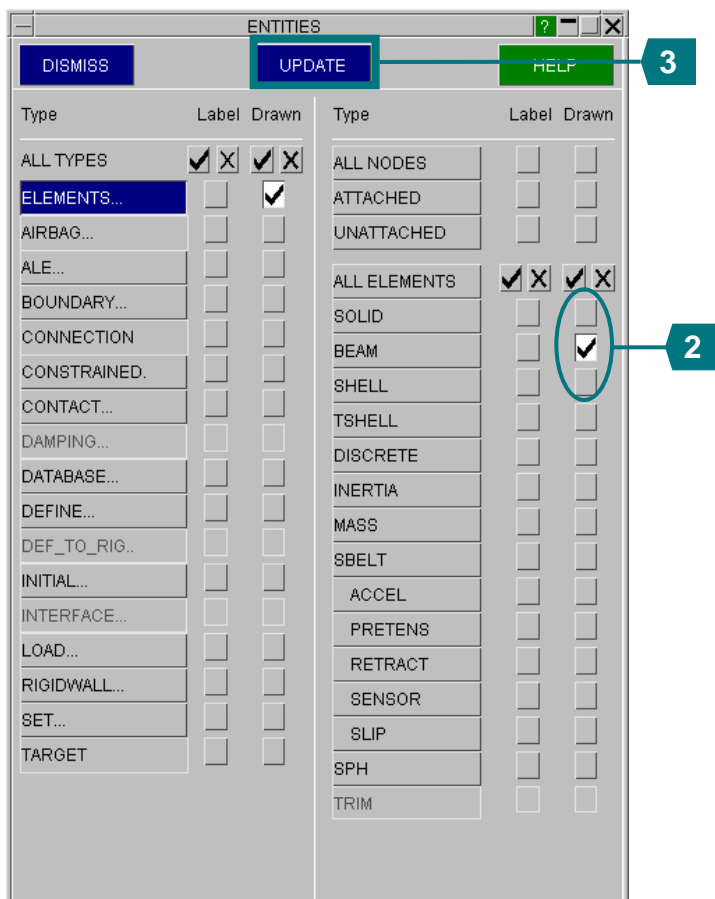
1. Change the action (left box) to **Transparency 70%**
2. Pick the roof to make it transparent
3. Middle-click to reset
4. Change the action to **colour – red** and entity (right box) to **Element – Shell**
5. Drag across an area to turn the shells red
6. Change the entity back to **Part**
7. Change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

Control of Visibility of Entities

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.



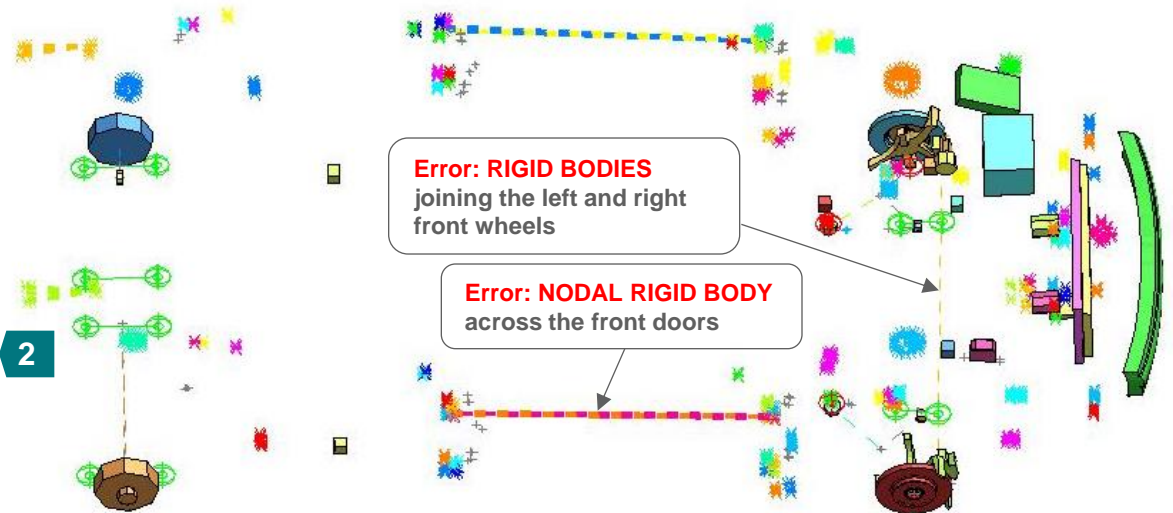
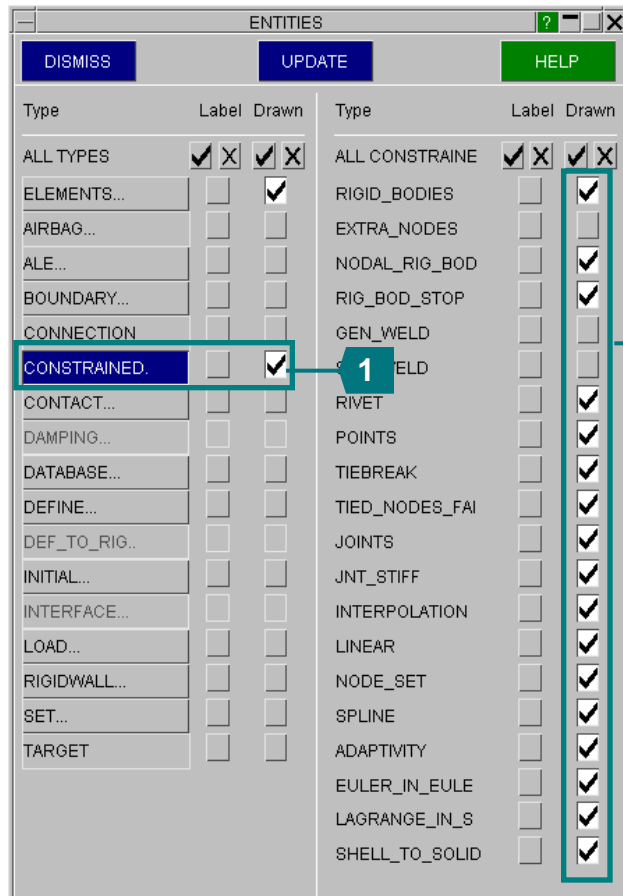


Entities Displayed

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.

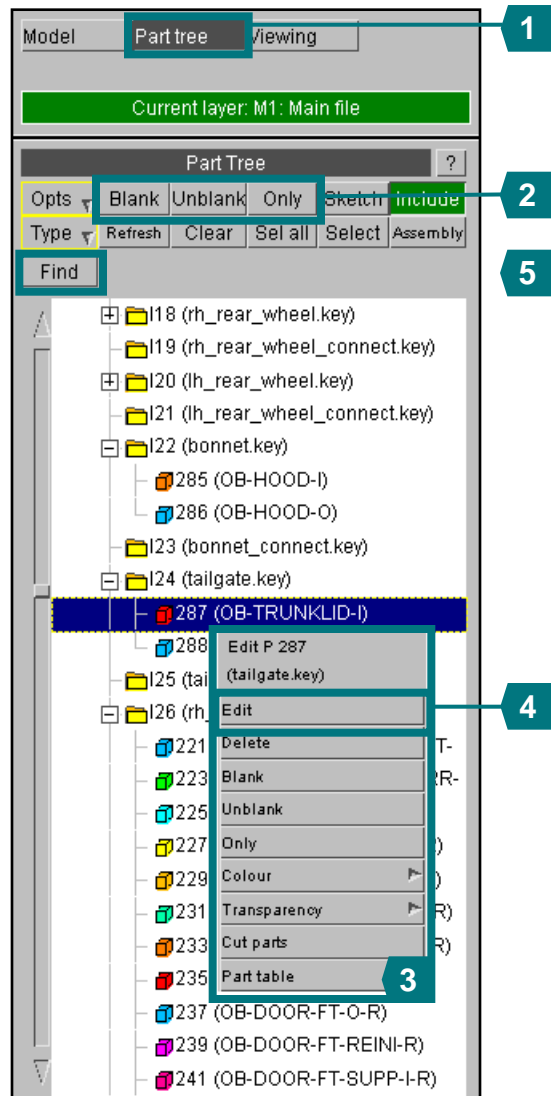


Entities Displayed Contd.

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
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.*

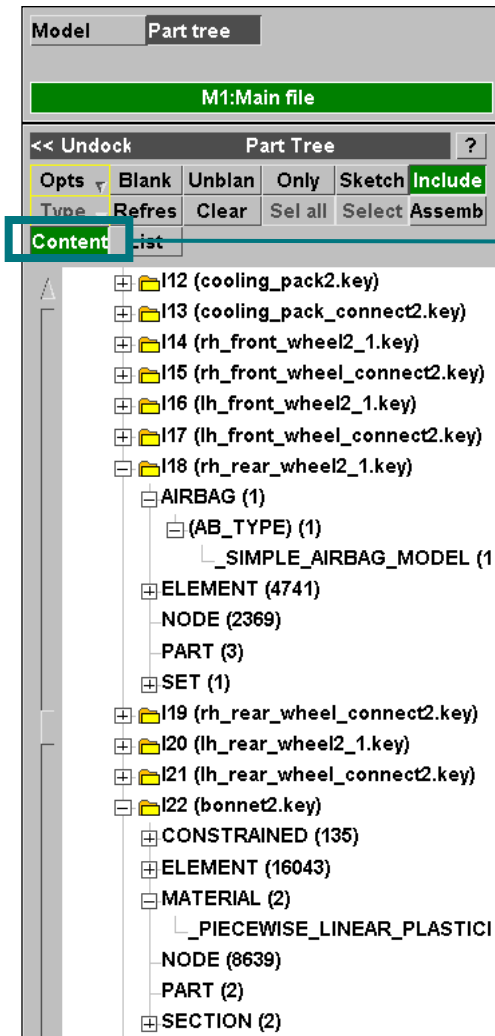


Part Tree

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

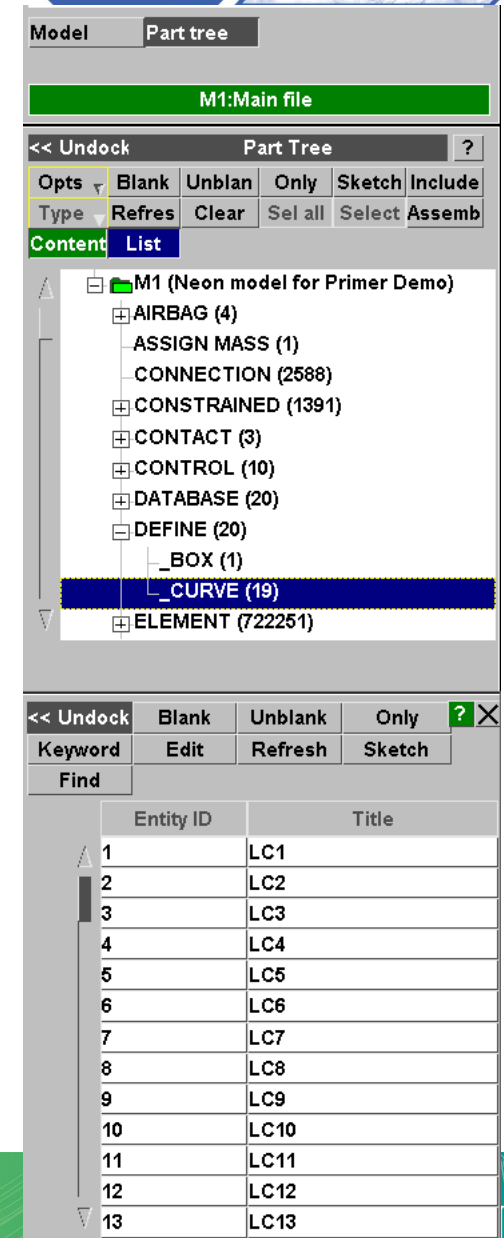


1

Part Tree

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

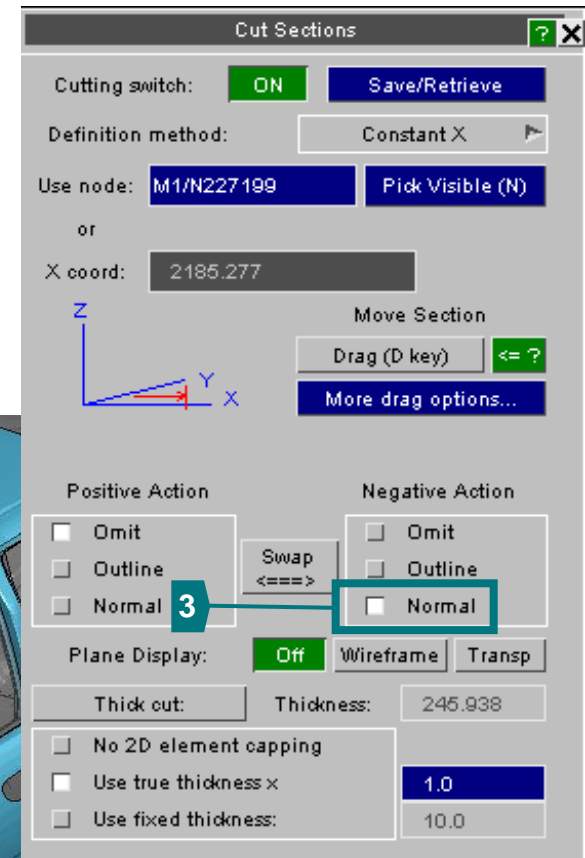
1. Click the **Part Tree** tab to bring up the Part Tree.
2. Click **CONTENTS**.
3. Expand an **INCLUDE** file (using the +).
4. Right-click, change the **colour**.
5. Press **INCLUDE** to turn off the display of include files.
6. Click **LIST** then select some entities



Cut section

Cut Section

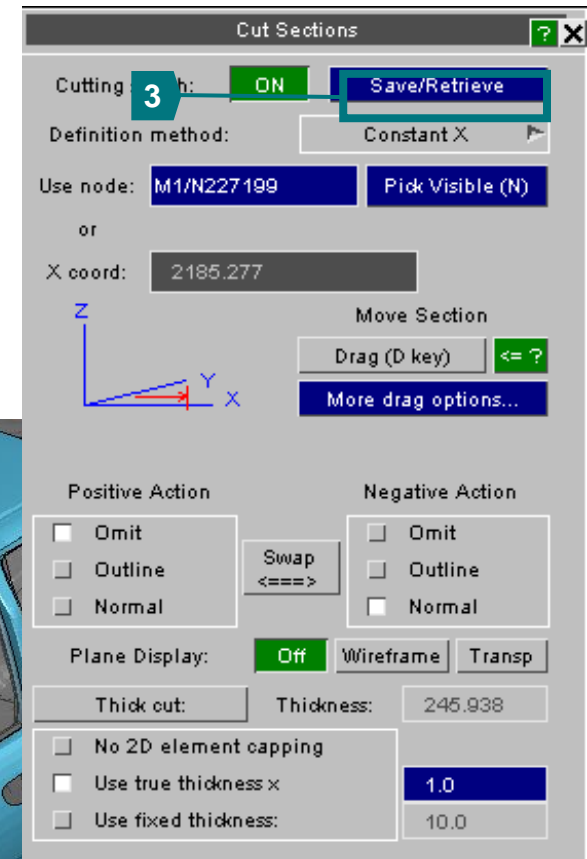
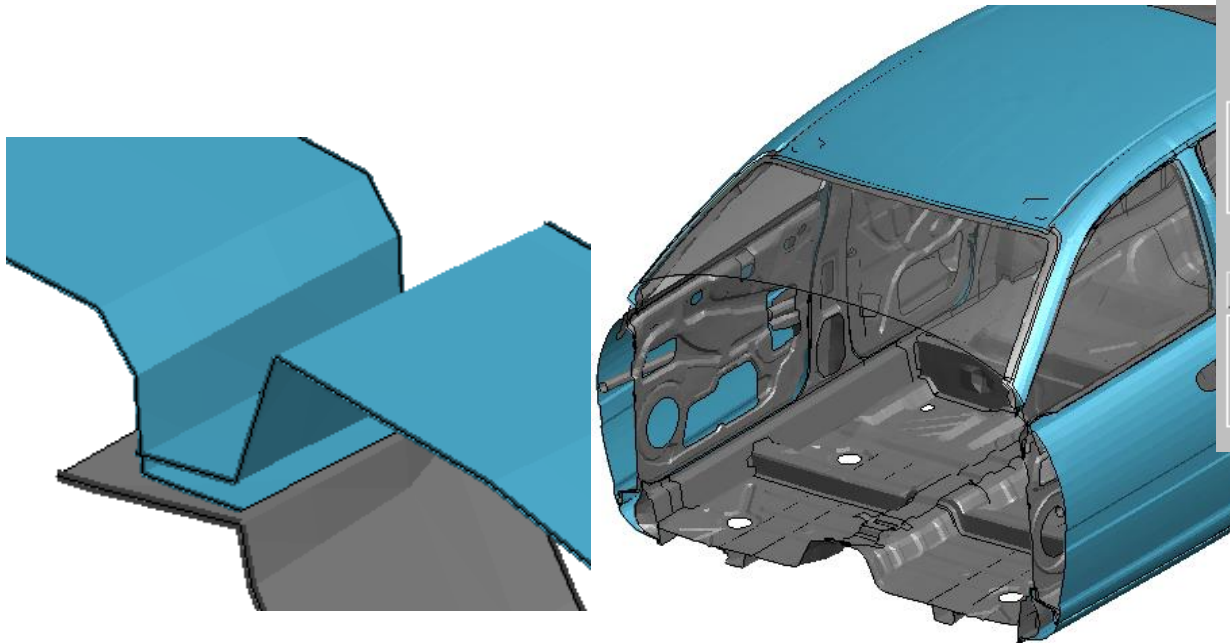
1. Shortcut **X** to bring up the cut-section menu.
2. Shortcut **N** to activate node-picking, then click near the centre of the roof. Display changes to a cut-section at constant X through the picked node.
3. In the menu, change Negative Action to **Normal** (i.e elements behind the plane are drawn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



Cut section

Cut Section

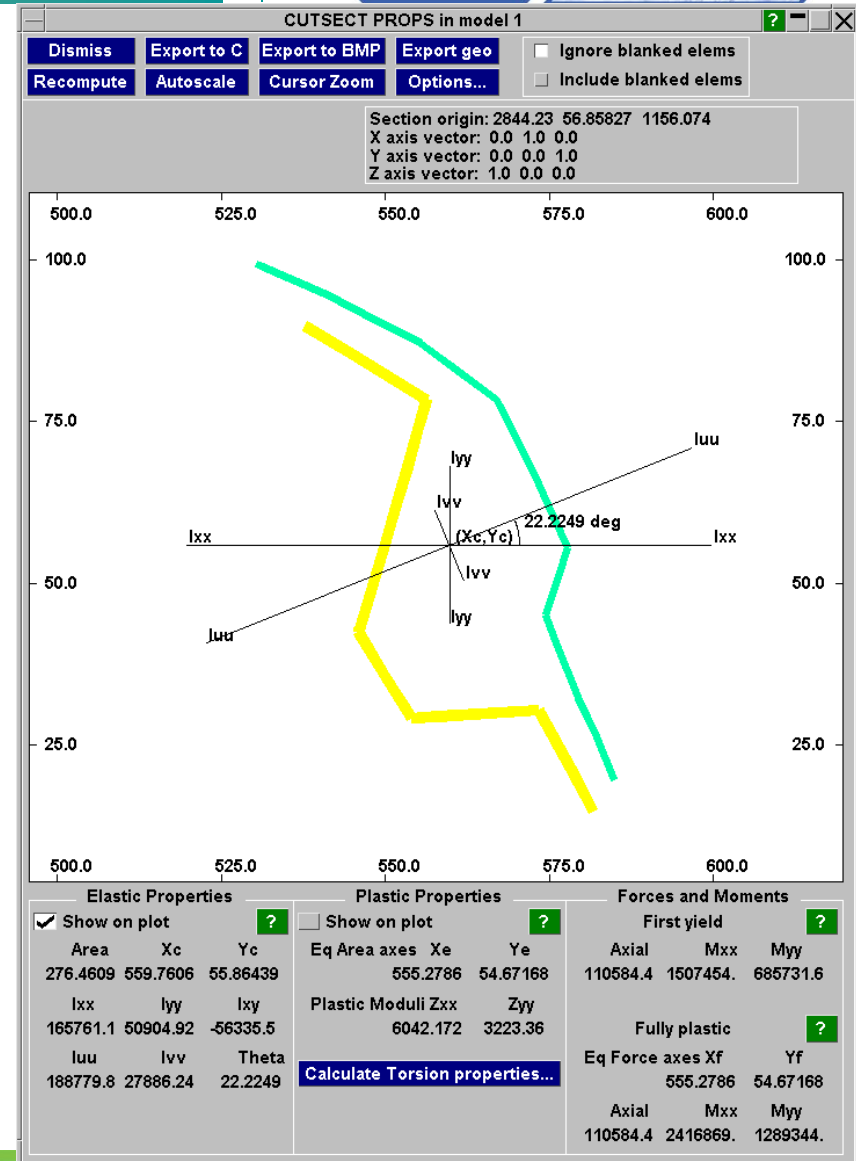
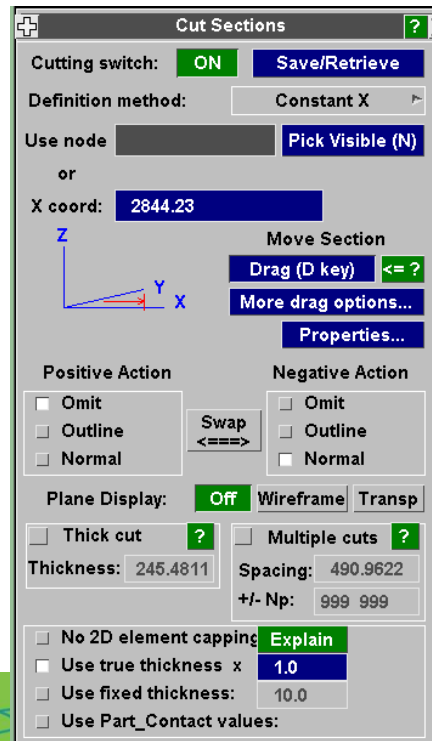
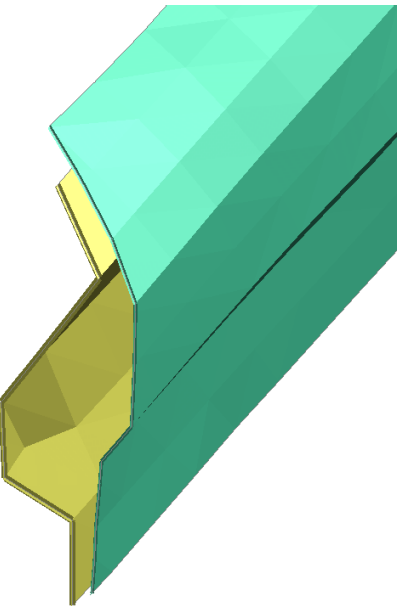
1. Zoom in on the cantrail. The cut edges of shells are drawn with their actual thickness – useful for seeing whether there are gaps (as in this model) or interferences.
2. Shortcut **D**, drag the section through the model.
3. Try switching the section to **Constant Y**.



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.



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

Using the Editing panels

1 Keywords panel: 'Part' is selected. The Part list shows 'M1/P163 (MC-BRAKE-CALIPER1-FT-L)' selected.

2 'MODIFY PART M1/P163' dialog box: The material ID '12' is highlighted, and a right-click context menu is open with options: MATERIAL 12, PICK..., SELECT..., CREATE..., EDIT..., BROWSE..., LABEL, SKETCH, STATUS.

3 'MODIFY MATERIAL M1/MAT12' dialog box: The material ID '12' has been changed to '1'. The material type is 'MAT_020: RIGID'.

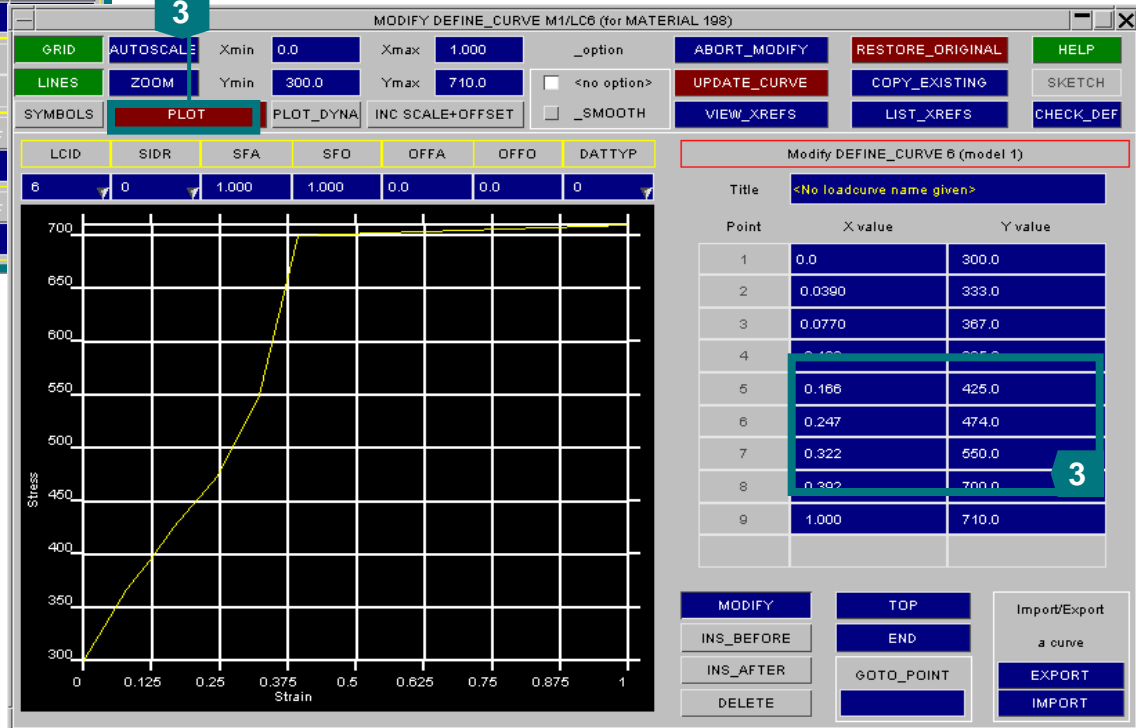
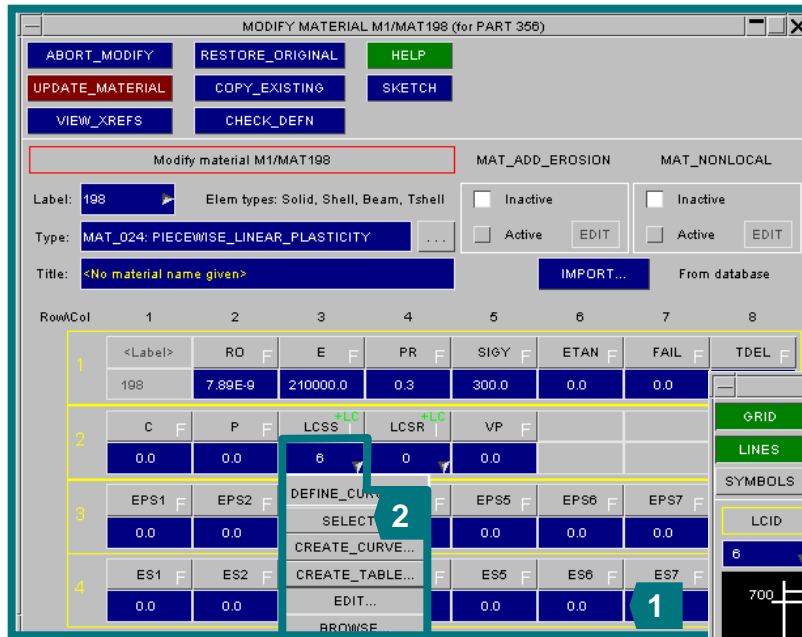
Example – editing a Part and its Material

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.

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**.



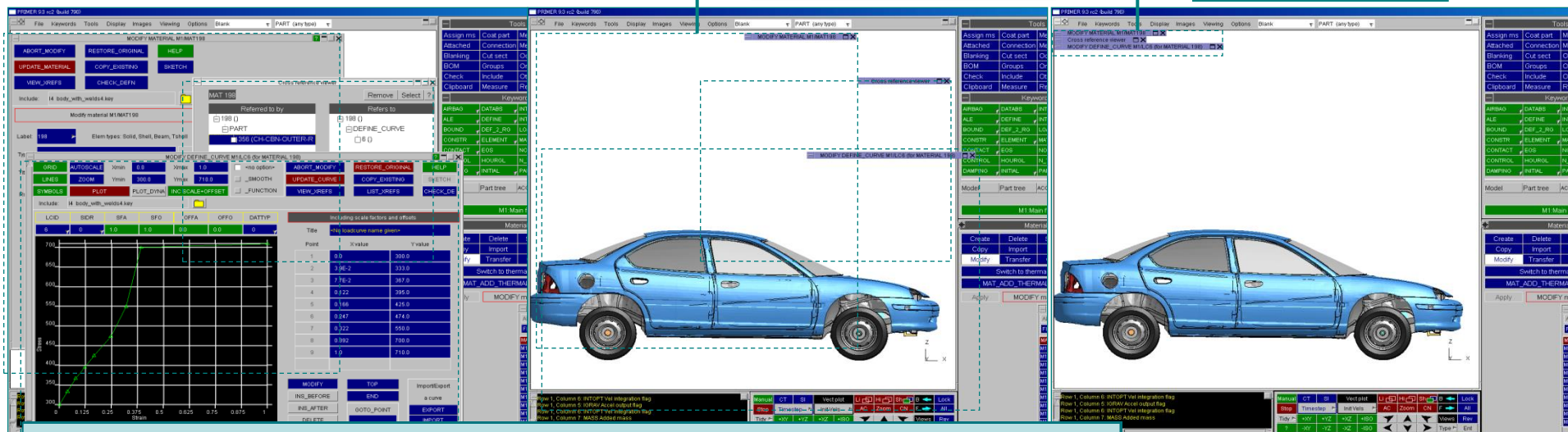
Floating Menus

Press **i** to **iconise**

1

Press **t** to **tidy**

2

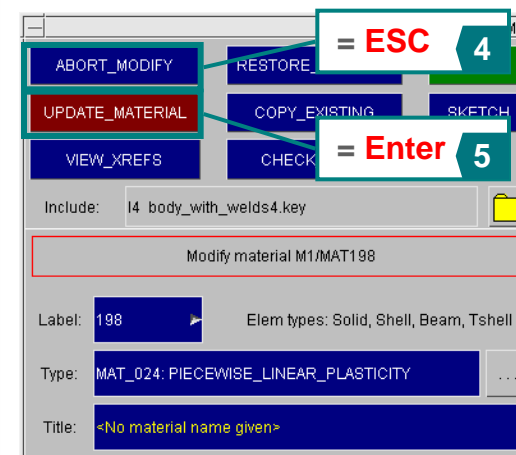


Floating Menus

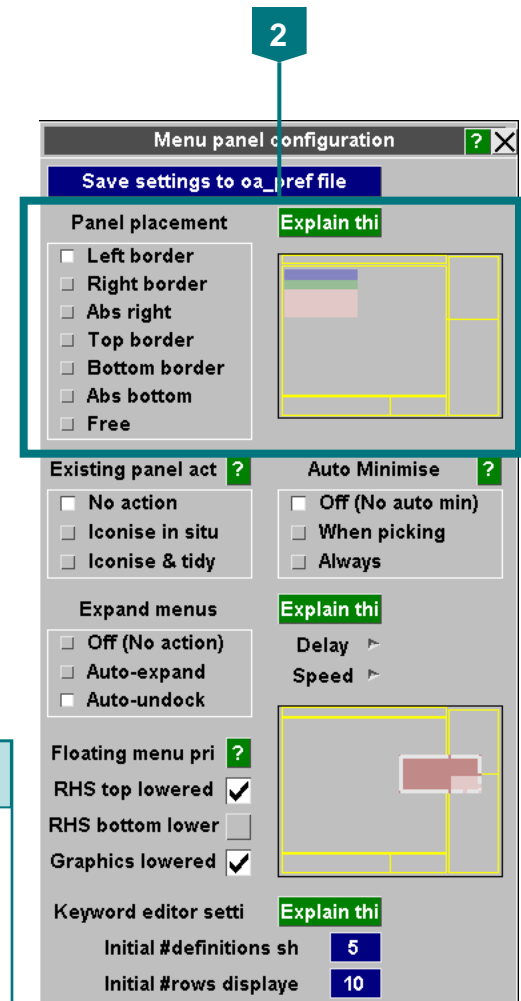
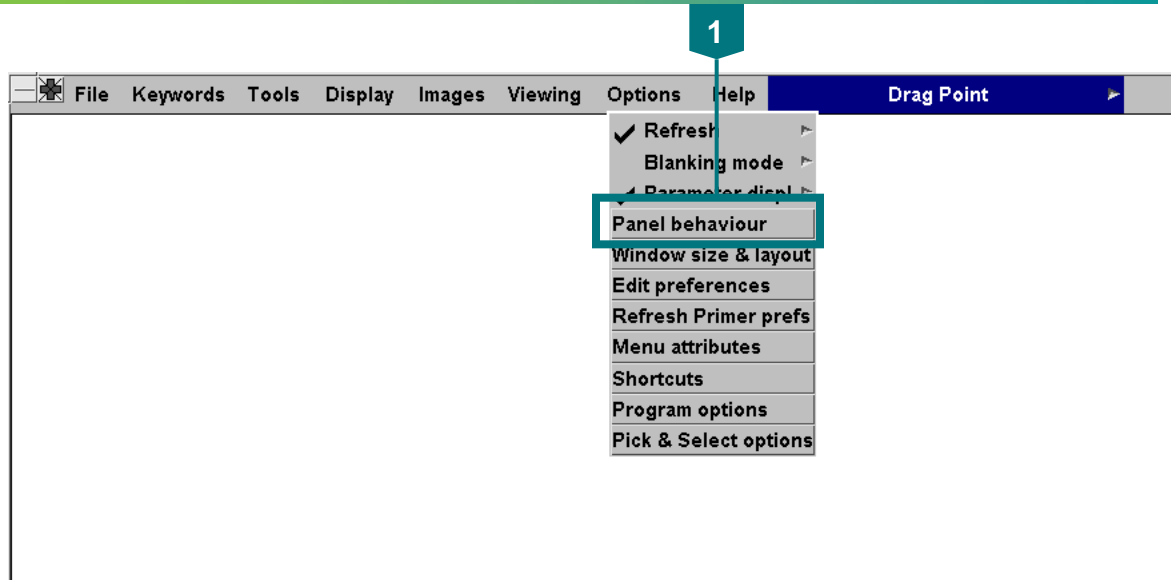
Many menus “float” over the graphics – they can be controlled in several ways:

1. Press **i** to **iconise** all floating menus – they will shrink to their top bar
2. Press **t** to **tidy** 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 (= **ABORT_MODIFY**)
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



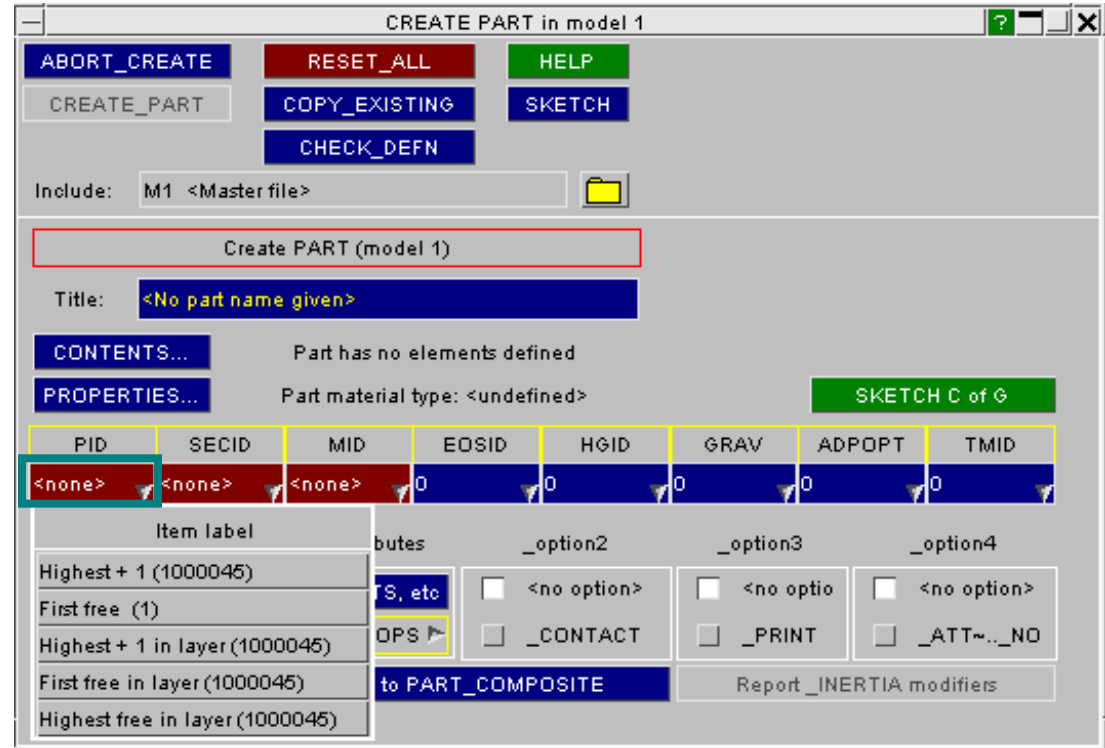
Floating Menus



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**).

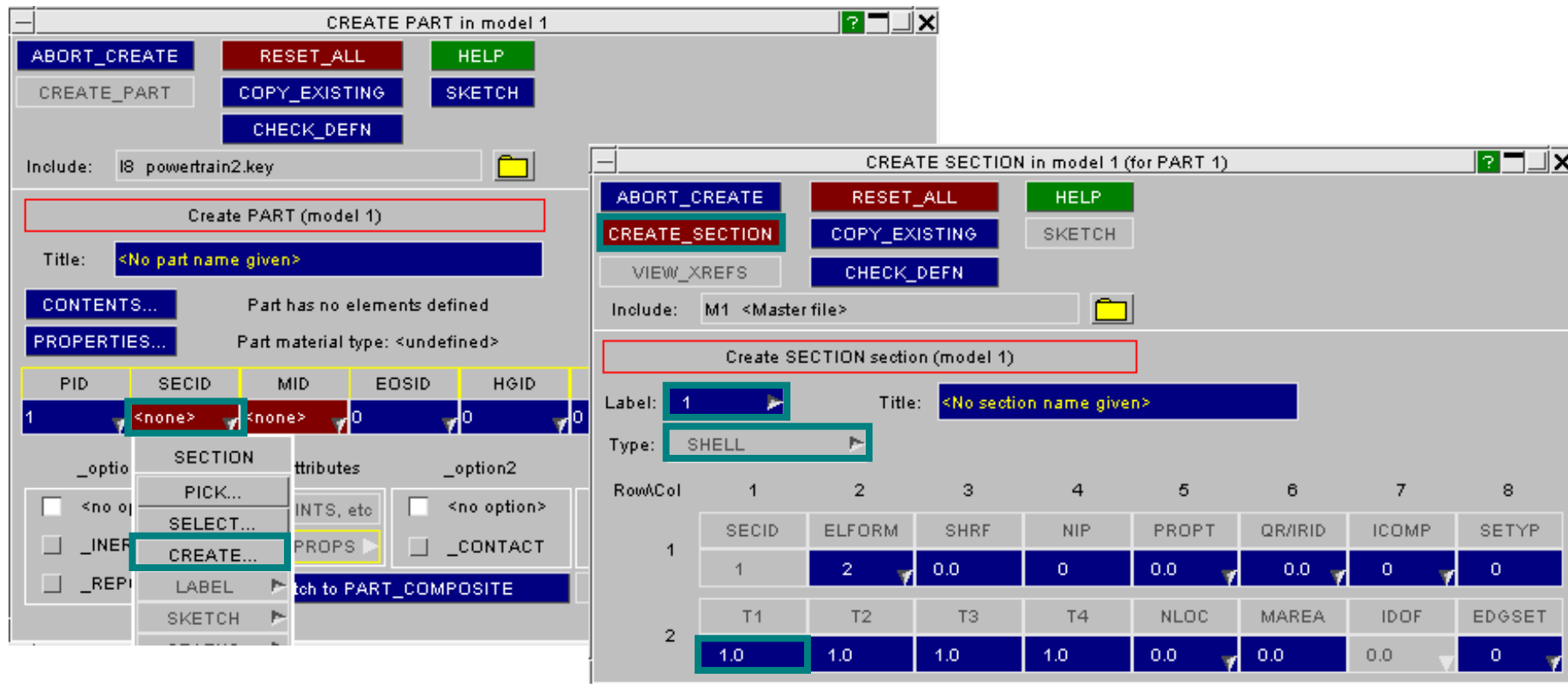


Creating data

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.

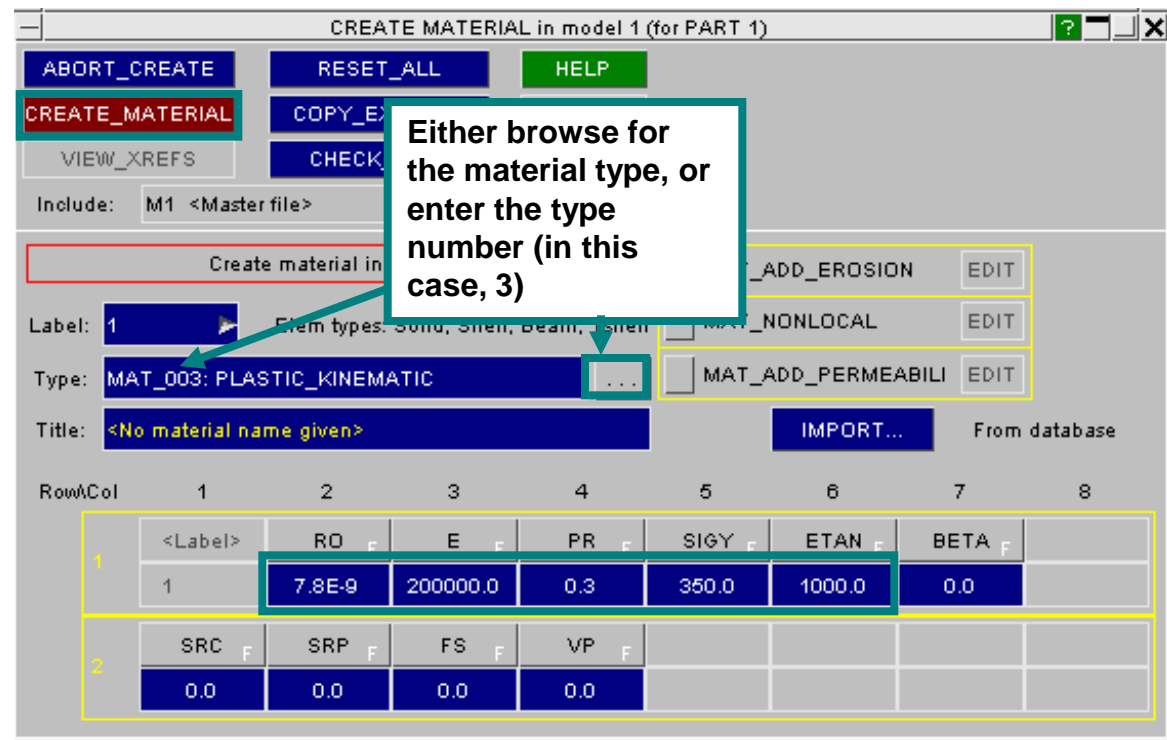
Creating data using Edit menus



Creating data

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

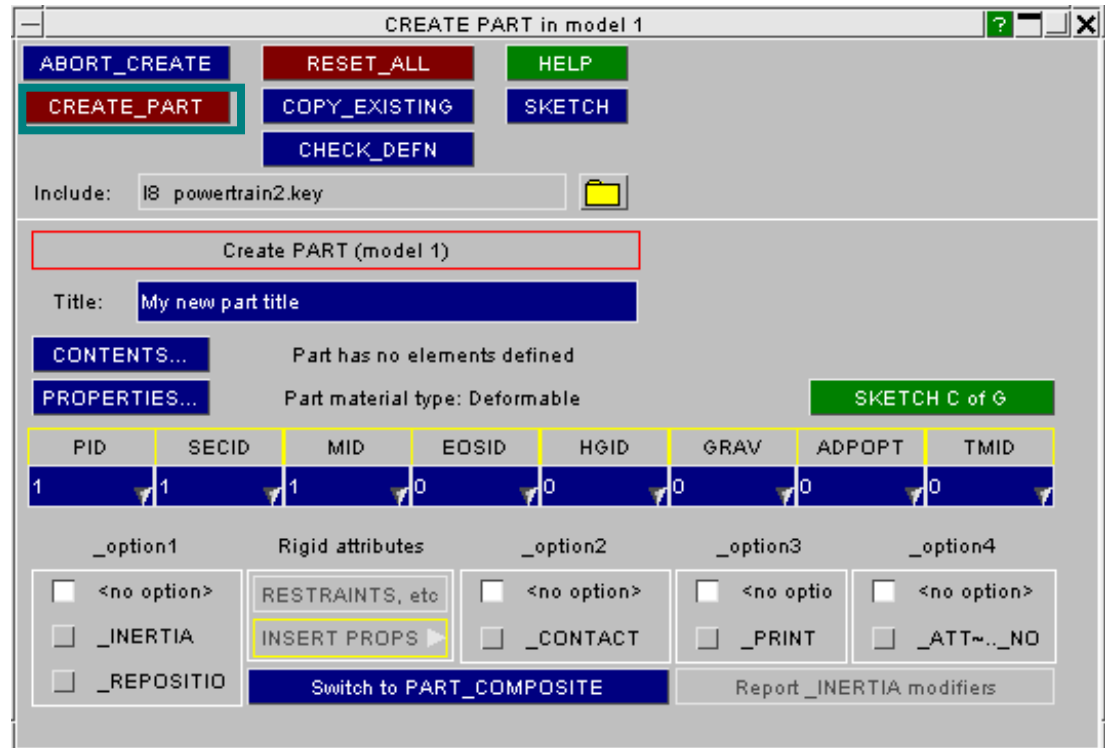
1. From Section ID, drop-down to Create.
2. Create a new section with ID 3001, element type Shell, thickness 1.0, and leave all other entries as the default. Press CREATE_SECTION.



Creating data

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.

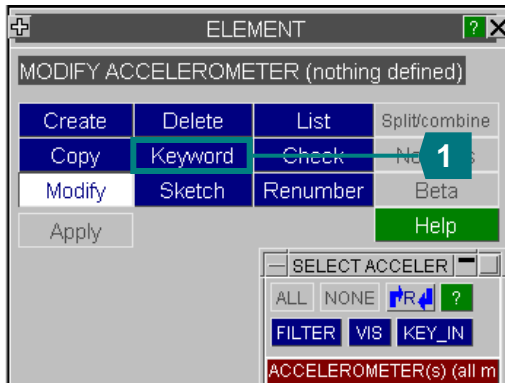


Creating data

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 Title"
2. In the Part menu, all the data input is now complete – press Create.

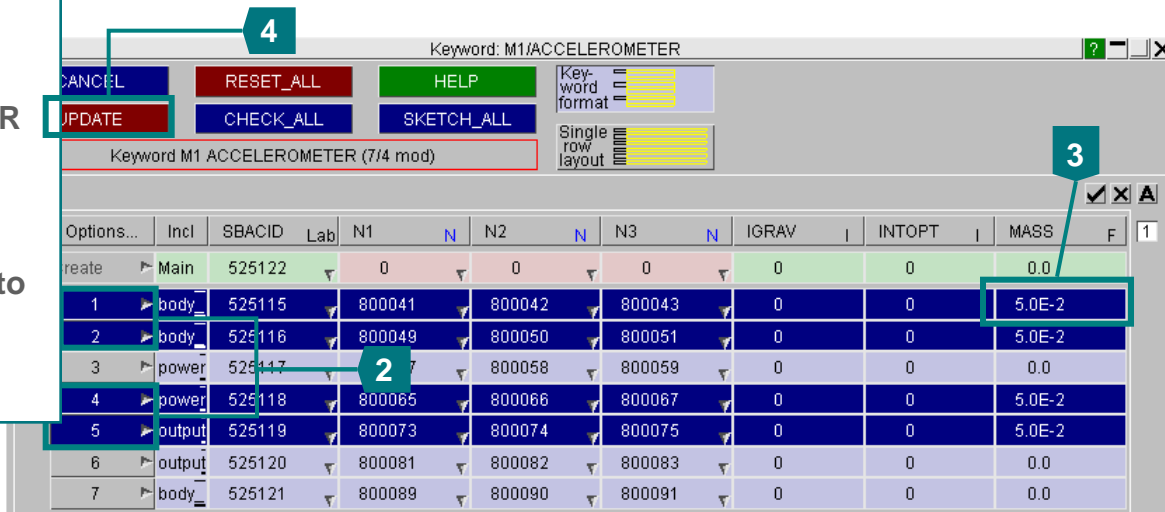
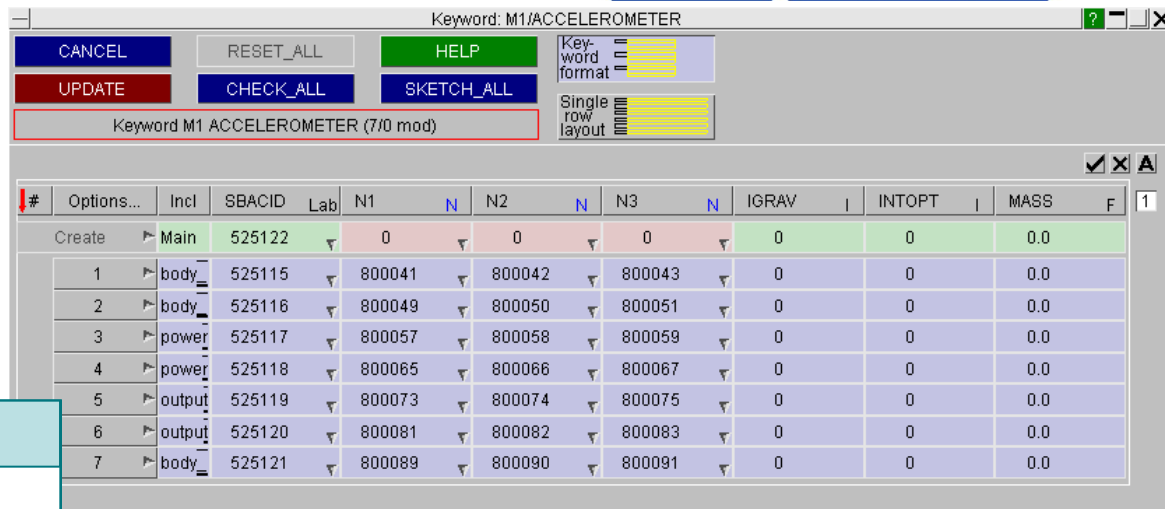
Keyword Editor – editing existing data



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: M1/INITIAL_VELOCITY

CANCEL RESET_ALL HELP UPDATE CHECK_ALL SKETCH_ALL

Keyword M1 INITIAL_VELOCITY (0/0 mod)

Key-word format Single row layout

#	Options...	Incl	NSID S_NO	NSIDEX S_NO	BOXID BOX	IRIGID S_PT			
			VX F	VY F	VZ F	VXR F	VYR F	VZR F	
			VXE F	VYE F	VZE F	VXRE F	VYRE F	VZRE F	
	Create	Main	0	0	0	0			
			13500.0	0.0	0.0	0.0	0.0	0.0	

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 Blue
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

The keyword editor displays all the _OPTIONS for this keyword together.
The _OPTIONS are shown in a column “Suffices”.

Keyword: M2/PREScribed_MOTION

CANCEL RESET_ALL HELP UPDATE CHECK_ALL SKETCH_ALL

Keyword M2 PREScribed_MOTION (9/0 mod)

Filter by: BOUNDARY_PREScribed_MOTION <auto> <auto>

#	Options...	Incl	Suffices	ID	La	TITLE	C												
				NID/NSID...	Var	DOF	I	VAD	I	LCID	+LC	SF	F	VID	VECT	DEATH	F	BIRTH	F
				OFFSET1	F	OFFSET2	F	MRB	P	NODE1	N	NODE2	N						
Create	Main		NODE_S	1500001		1		0		10000		0.0		0		0.0		0.0	
1	wheel		RIGID	86009															
2	wheel		RIGID	86009															
3	wheel		RIGID	86009															
4	wheel		RIGID	86009															
5	belt		NODE	2028644															
6	belt		NODE	2021124															
7	belt		NODE_S	1500083															
8	wheel		RIGID_ID	86009															
9	wheel		RIGID_ID	86009															

*BOUNDARY_PREScribed_MOTION_RIGID

*BOUNDARY_PREScribed_MOTION_NODE

*BOUNDARY_PREScribed_MOTION_NODE_SET

*BOUNDARY_PREScribed_MOTION_RIGID_ID

Keyword Editor

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

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

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

Keyword Editor

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

CANCEL RESET_ALL HELP
UPDATE CHECK_ALL SKETCH_ALL

Keyword M1 BEAM (2981/0 mod)

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

Key-word format
Single row layout

Line 1 Line 2

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

Keyword Editor

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

CANCEL RESET_ALL HELP UPDATE CHECK_ALL SKETCH_ALL

Keyword M1 BEAM (2981/0 mod)

Filter by: ELEMENT_BEAM <auto> ☒ ☒ **A** ☒ ? ☒ to> <auto>

Key-word format Single row layout

“A” means automatically filter to include all _OPTIONS present in the model

#	Options...	Incl	Suffices	EID		N	N2	N	N3	N	RT1	RR1	RT2	RR2	LOCAL	PID1	P
Create	Main		<none>	4600													
1	body	PID		1													
2	body	PID		2													
3	body	PID		3													
4	body	PID		4													
5	body	PID		5													
6	body	PID		6													
7	body	PID		7													
8	body	PID		8													
9	body	PID		9													

☒ <none>
☐ THICKNESS
☐ SCALAR
☐ SCALR
☐ SECTION

Hide/show entities with these OPTIONS

Keyword Editor

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

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

#	Options...	Incl	Surfaces	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	susp	<none>		460006		306		297650		460004		310567		0		0		0		0		0	
2	ancill	<none>		300729		289		300419		157516		300413		0		0		0		0		0	
3	power	<none>		303502		2103		303603		303631		303622		0		0		0		0		0	
4	power	<none>		303500		174		303707		303747		303622		0		0		0		0		0	

Keyword Editor

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

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

Options...

- Options..
- Sel_All
- Unsel_All
- Select...
- Show_All
- Only_Sel
- Hide_Sel
- Sketch_Sel
- Reset_Sel
- Delete_Sel
- Help

#	EID	Lab	PID	P	N1	N	N2	N	N3	N	RT1	I	RR1	I	RT2	I	RR2	I	LOCAL	I
	PID1	P	PID2	P																
	460053		0		0		0		0		0		0		0		0		0	
1	460006		306		297650		460004		310567		0		0		0		0		0	
2	300729		289		300419		157516		300413		0		0		0		0		0	
3	303502		2103		303603		303631		303622		0		0		0		0		0	
4	303500		174		303707		303747		303622		0		0		0		0		0	

Keyword Editor

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

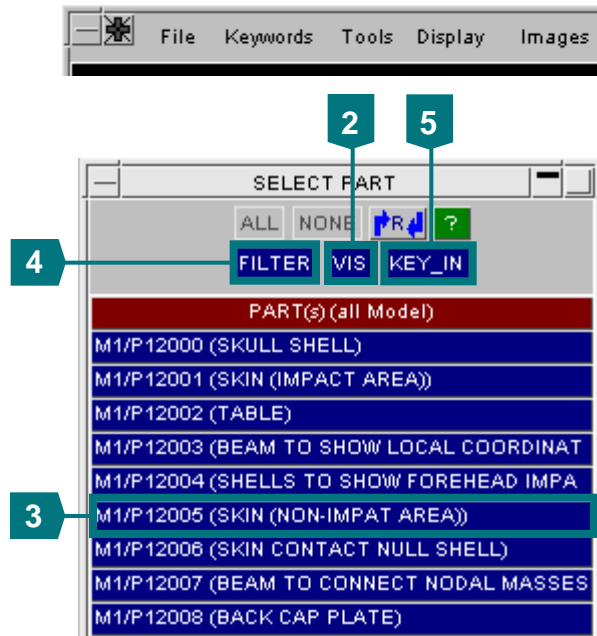
Filter by: ELEMENT_BEAM <auto> <auto> <auto> <auto> <auto>

#	Options...	Incl	Suffixes	EID	Lab	PID	P	N1	N	N2	N	N3	N	RT1	RR1	RT2	RR2	LOCAL
				PID1	P	PID2	P											
Create	Main	<none>		460053		0		0		0		0		0	0	0	0	0
1				460006		306		297650		460004		310567		0	0	0	0	0
2				300729		289		300419		157516		300413		0	0	0	0	0
3				303502		2103		303603		303631		303622		0	0	0	0	0
4				303500		174		303707		303747		303622		0	0	0	0	0

Right-click context menu for entity 1:

- B460006
- Edit...
- Check
- Reset
- Xrefs
- Sketch
- Blank
- Unblank
- Only
- Delete
- Help

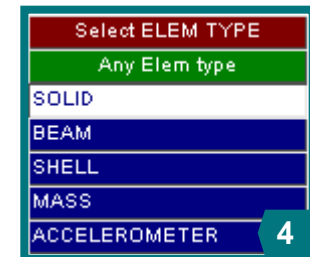
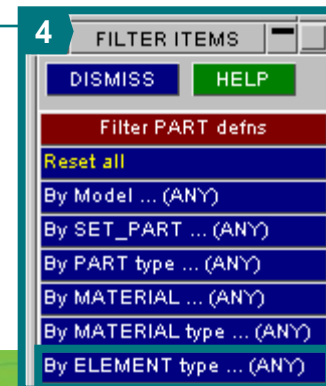
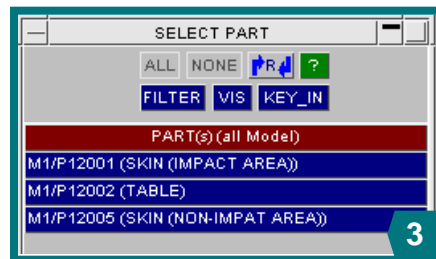
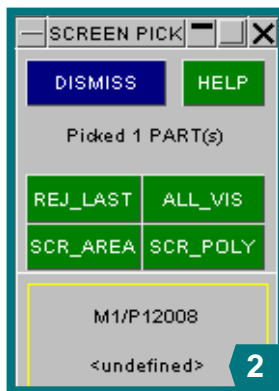
Using Object Selection Menu



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 **solid**.
5. A further alternative is to choose **KEY_IN** & type in the entity ID.



Using Object Selection Menu

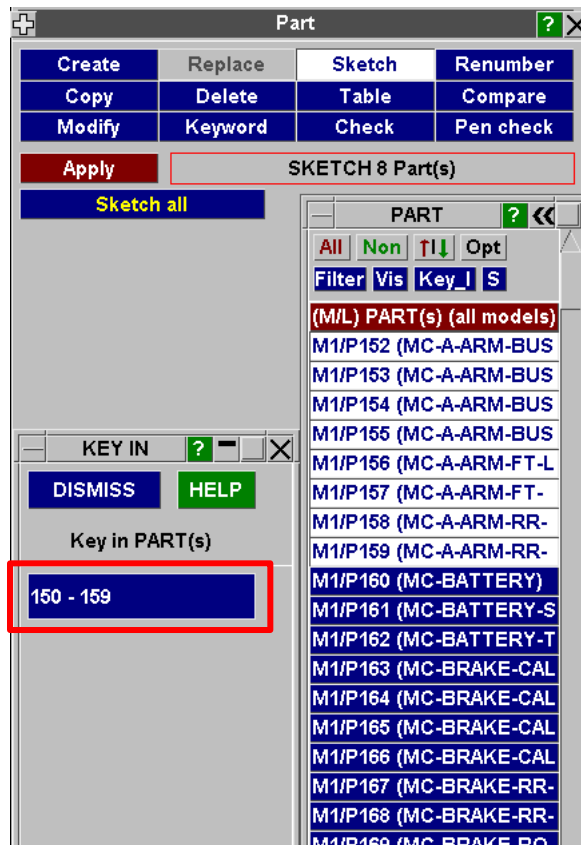
Object menu order

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

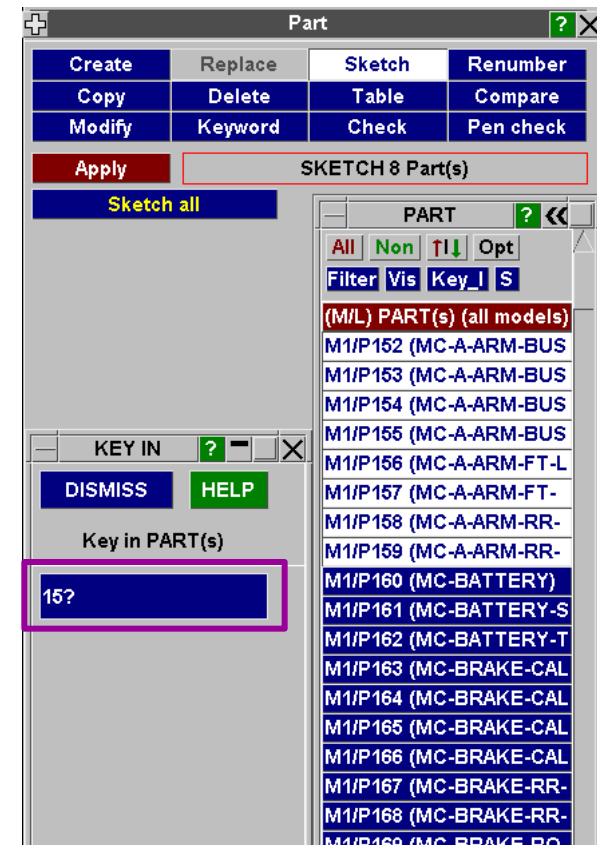
SELECT PART ?	SELECT PART ?	SELECT PART ?	SELECT PART ?
All None ↑↓ 0 Filter Vis Key_In	All None ↑↓ 0 Filter Vis Key_In	All None ↑↓ 0 Filter Vis Key_In	All None ↑↓ Opt Filter Vis Key_In Sk
(A-Z) PART(s) (all models)	(Z-A) PART(s) (all models)	(0-9) PART(s) (all models)	(9-0) PART(s) (all models)
M1/P20912 (ACCEL PEDAL BRACKET)	M1/P4003 (WHEEL SOLID RUBBER)	M1/P1000 (H3.50 HEAD - NULL SHELLS FC	M1/P21915 (IN - mid seat lower cushion)
M1/P20911 (ACCEL PEDAL PAD)	M1/P4005 (WHEEL SEAM FAIL)	M1/P1001 (H3.50 HEAD - NULL SHELLS FC	M1/P21914 (IN - mid seat tray 2)
M1/P20902 (ACCEL PEDAL RIGID PIVOT)	M1/P4004 (WHEEL MODULE LID)	M1/P1002 (H3.50 NECK - NULL SHELLS FC	M1/P21913 (IN - mid seat bracket2)
M1/P20913 (ACCEL PEDAL ROT SPRING)	M1/P4007 (WHEEL MODULE CASE)	M1/P1003 (H3.50 NECK - NULL SHELLS FC	M1/P21912 (IN - mid seat bracket1)
M1/P20910 (ACCEL PEDAL SPOTWELDS)	M1/P4002 (WHEEL FRAME)	M1/P1004 (H3.50 THORAX - NULL SHELLS	M1/P21911 (IN - mid seat tray bracket)
M1/P20918 (ACCEL PEDAL)	M1/P4001 (WHEEL CASTING MOUNT)	M1/P1005 (H3.50 THORAX - NULL SHELLS	M1/P21910 (IN - mid seat tray 1)
M1/P21261 (ADHESIVE - WINDSHIELD BOND	M1/P4006 (wheel casting coat shell)	M1/P1006 (H3.50 LTORSO - NULL SHELLS	M1/P21909 (IN - mid seat middle casing 1)
M1/P11011 (BELT BRKT JNT RIGID)	M1/P4000 (WHEEL BACK PLATE MOUNT)	M1/P1007 (H3.50 LTORSO - NULL SHELLS	M1/P21908 (IN-R-mid seat low casing 2)
M1/P11012 (BELT BRKT JNT RIGID)	M1/P20800 (STR WHL RACK CONNECT)	M1/P1008 (H3.50 LTORSO - NULL SHELLS	M1/P21907 (IN-L-mid seat low casing 1)
M1/P21088 (BIW -firewall brkt up - L - I)	M1/P20696 (SPOTWELDS HEXA PWR)	M1/P1009 (H3.50 UPPARML - NULL SHELL	M1/P21906 (IN - mid seat low cush frame 3)
M1/P21089 (BIW -firewall brkt up - R - I)	M1/P21206 (SPOTWELDS HEXA BIW)	M1/P1010 (H3.50 LOWARML - NULL SHELL	M1/P21905 (IN - mid seat low cush frame 2)
M1/P21090 (BIW -firewall mid brkt 1 up - I)	M1/P11002 (SHOULDER ANCHOR)	M1/P1011 (H3.50 HANDL - NULL SHELLS F	M1/P21904 (IN - mid seat low cush frame 1)
M1/P21091 (BIW -firewall mid brkt 2 up - I)	M1/P12348 (SETA LH FLOOR MOUNT)	M1/P1012 (H3.50 UPPARMR - NULL SHELL	M1/P21903 (IN - mid seat frame 4 - FR)
M1/P21058 (BIW - A pillar reinforcmen2 - L - I)	M1/P11000 (SEATBELT SHOULDER SLIPRIN	M1/P1013 (H3.50 LOWARMR - NULL SHELL	M1/P21902 (IN - mid seat frame 3 - FR)
M1/P21113 (BIW - A pillar reinforcmen2 - R - I)	M1/P11001 (SEATBELT RETRACTOR)	M1/P1014 (H3.50 HANDR - NULL SHELLS F	M1/P21901 (IN - mid seat frame 2 - FR)
M1/P21070 (BIW - A pillar bracket 1 - L - I)	M1/P12336 (SEAT_RRMECH_OUTBOARD)	M1/P1015 (H3.50 UPPLEGL - NULL SHELL	M1/P21900 (IN - mid seat frame 1 - FR)

Keyword Editor – KEY_IN

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



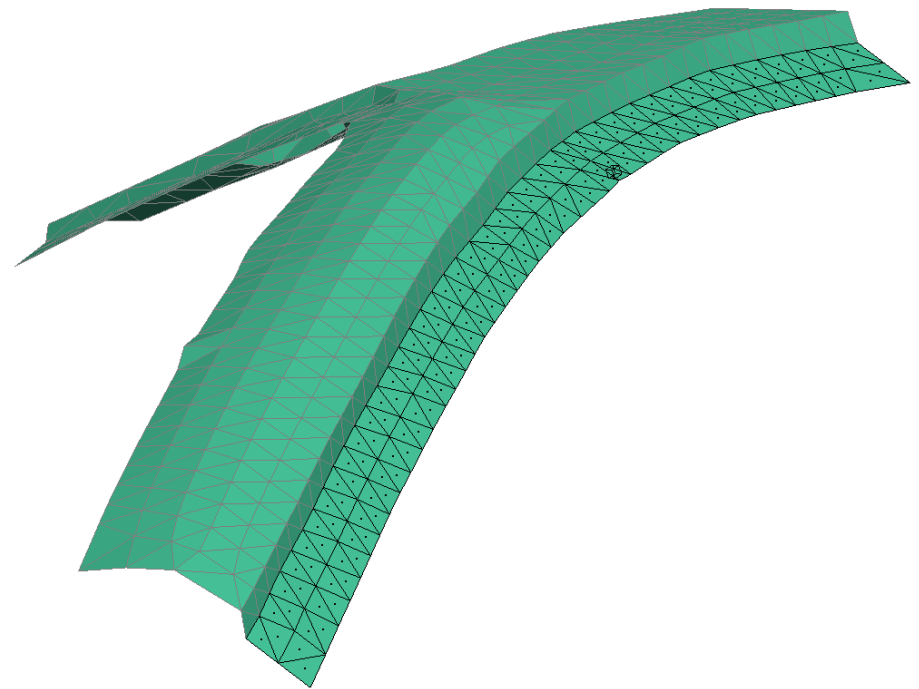
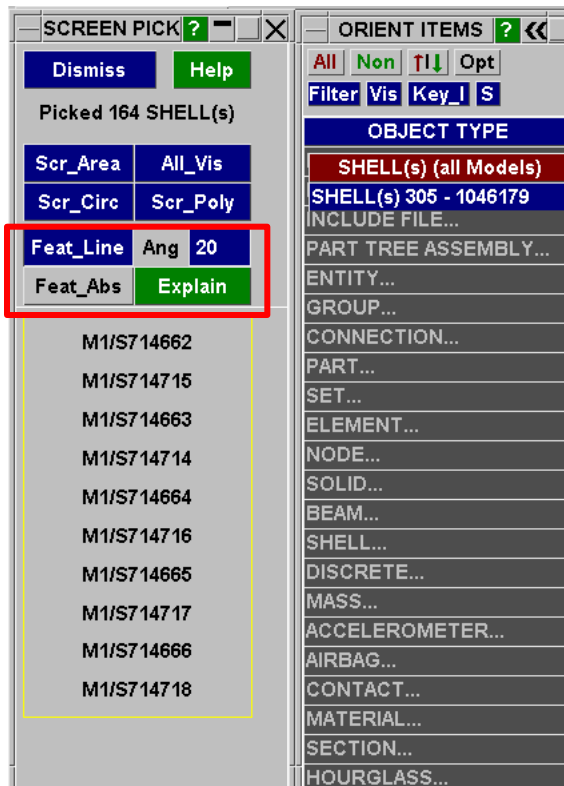
Select all parts with ID's between 150 and 159 inclusive



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

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



Options Blank ▾ PART (any type) ▾

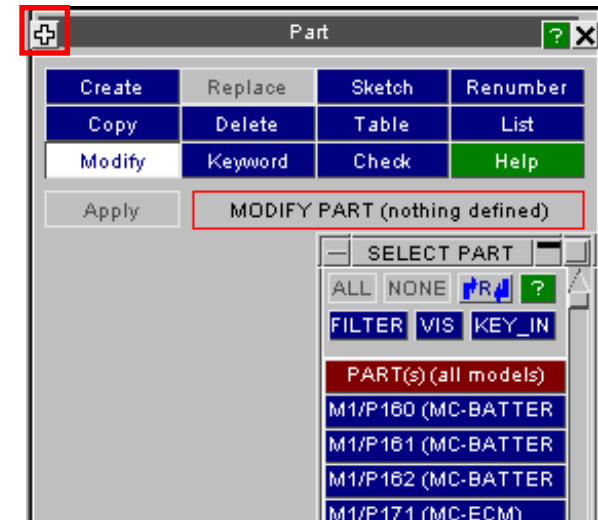
Cursor action = Quick-pick

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

Options Pick PART ▶

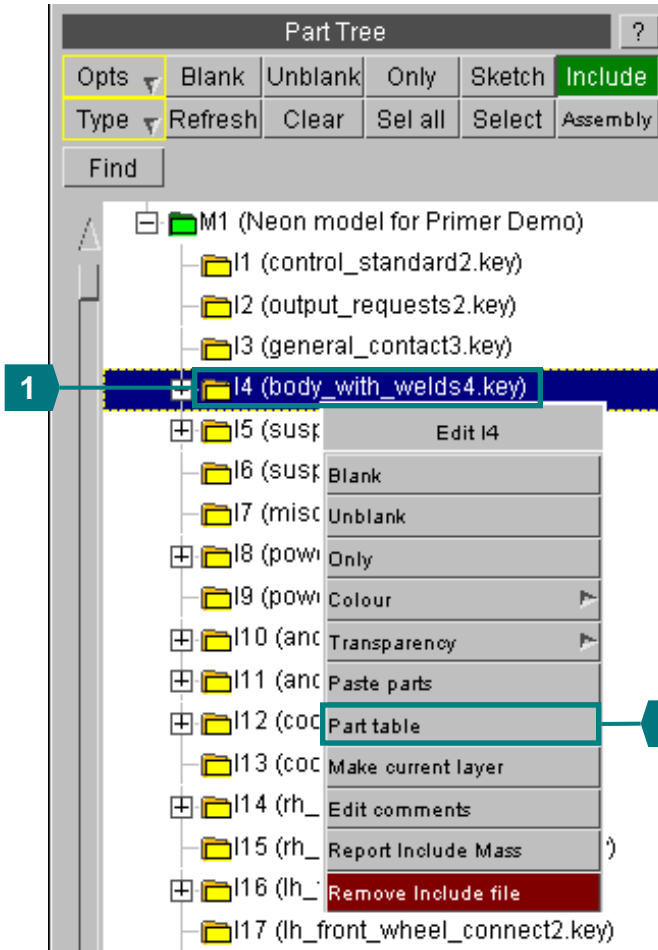
Cursor action = menu pick

- 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

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 **Part Mass**, **Added Mass**, **NIP** and **Elform**.
4. Turn off **Part Type** and **Section ID**.



3

PART TABLE											
Dismiss	View...	Refresh	Write...	Table Changes:	Undo	Apply	Mass in table: 0.217077	?			
Part ID	Model	HQ Coeff	<input checked="" type="checkbox"/> Part Mass	Include		ed Mass	NIP	Elform			
<input checked="" type="checkbox"/> Part ID	<input checked="" type="checkbox"/> Mat ID		Lumped Mass	Dismiss		43142]					
281	<input checked="" type="checkbox"/> Part title	Mat title	<input checked="" type="checkbox"/> Added Mass			207e-008	3	2			
282	Part type	Mat type	Added Mass %				3	2			
283	Section ID	Yield	C of G			323e-006	3	2			
284	Section title	Modulus	Inertia (XX,YY,Z			374e-006	3	2			
308	<input checked="" type="checkbox"/> Gauge	Density	Inertia (XY,XZ,Y			:569e-005	3	2			
309	<input checked="" type="checkbox"/> NIP	EOS ID	Blanking			:761e-005	3	2			
310	<input checked="" type="checkbox"/> Elform	Struct Mass	Colour			:752e-005	3	2			
311	HQ ID	Assign Mass	Transparency			:483e-005	3	2			
312	HQ Type	NS Mass	Style			:675e-005	3	2			
313	CH-A-PILLAR-BRKT1-R	2.159000 155	0.00036921	1.3138e-005	3	2					
314	CH-A-PILLAR-BRKT3-L	2.147000 156	0.00015730	1.28988e-006	3	2					
315	CH-A-PILLAR-BRKT3-R	2.147000 157	0.00015737	1.29027e-006	3	2					
316	CH-A-PILLAR-BRKT4-L	2.805000 158	0.00012169	0	3	2					
317	CH-A-PILLAR-BRKT4-R	2.805000 159	0.00012169	0	3	2					
318	CH-A-PILLAR-I-L	1.561000 160	0.00066424	3.44369e-006	3	2					
319	CH-A-PILLAR-I-R	1.561000 161	0.00066426	3.43556e-006	3	2					

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							
Dismiss View... Refresh Write...		Table Changes: Undo Apply		Mass in table: 0.217077 ?			
Part ID	Part title	Gauge	Mat ID	Part Mass [0.202763]	Added Mass [0.0143142]	NIP	Elform
353	CH-CBN-FLOORBRD-FT	0.705000	195	0.0131395	0.000394783	3	2
354	CH-CBN-FLOORBRD-RR	0.706000	196	0.0122001	9.26798e-005	3	2
355	CH-CBN-OUTER-L	0.829000	197	0.011437	0.00028267	3	2
416	CH-ROOF	0.702000	250	0.0068424	3.09549e-005	3	2
362	CH-CBN-SILL-B	1.701000	204	0.00637916	2.13682e-005	3	2
414	CH-RAILS-U-RR-R	1.916000	248	0.00516152	4.8488e-005	3	2
410	CH-RAILS-U-RR-L	1.916000	244	0.00515992	4.84855e-005	3	2
391	CH-RAILS-FT-R-I	1.895000	225	0.0038872	0.00245228	3	2
389	CH-RAILS-FT-L-I	1.895000	223	0.00382221	0.000903256	3	2
311	CH-A-PILLAR-B-O-R	1.611000	153	0.00351327	5.43483e-005	3	2
352	CH-CBN-FIREWALL	0.735000	194	0.00344807	0.000228272	3	2
310	CH-A-PILLAR-B-O-L	1.611000	152	0.00344365	5.47752e-005	3	2
381	CH-RAD-SUPPORT-B1	1.314000	215	0.00304176	0.000217485	3	2
347	CH-CBN-DECK-SPKR-SUPP	0.960000	189	0.00287323	1.84788e-005	3	2
610	CH-RAILS-U-RR-R-REINF	2.520000	295	0.00277413	7.5753e-005	3	2
609	CH-RAILS-U-RR-L-REINF	2.520000	294	0.0027737	7.80346e-005	3	2

Part Table

1. Next, click twice on the column heading “**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 “**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.

2

1

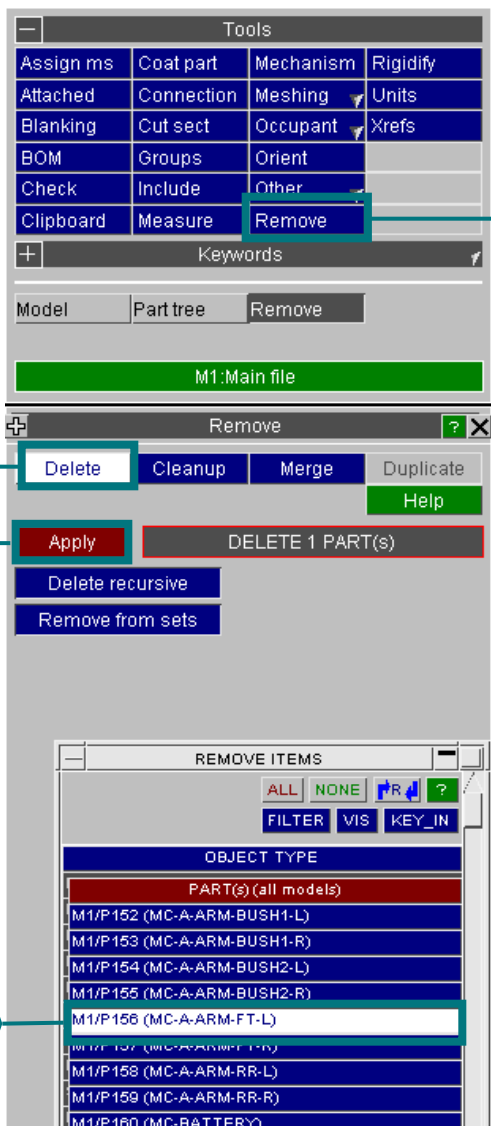
3

PART TABLE

Dismiss View... Refresh Write... Table Changes: Undo Apply Mass in table: 0.217077 ?

Part ID	Part title	Gauge	Mat ID	Part Mass [0.202763]	Added Mass [0.0143142]	NIP	Elform
616	CH-RAILS-U-RR-SUSP-MTS-2	4.500000	301	5.70972e-005	0	5	2
615	CH-RAILS-U-RR-SUSP-MTS-1	4.500000	300	5.70731e-005	0	5	2
612	CH-RAILS-U-RR-R-REINF2	3.700000	297	0.00022644	3.54961e-005	5	2
611	CH-RAILS-U-RR-L-REINF2	3.700000	296	0.000226448	3.54875e-005	5	2
317	CH-A-PILLAR-BRKT4-R	2.805000	159	0.000121698	0		
316	CH-A-PILLAR-BRKT4-L	2.805000	158	0.000121698	0		
614	CH-FENDER-SILL-ATT2-R	2.520000	299	0.000143157	0		
613	CH-FENDER-SILL-ATT2-L	2.520000	298	0.00014316	0		
610	CH-RAILS-U-RR-R-REINF	2.520000	295	0.00277413	7.5753e-005	3	2
609	CH-RAILS-U-RR-L-REINF	2.520000	294	0.0027737	7.80346e-005	3	2
603	CH-FENDER-SILL-ATT-R	2.500000	293	0.000413244	0	3	2
602	CH-FENDER-SILL-ATT-L	2.500000	292	0.000428892	6.59022e-007	3	2
427	CH-SHOCKHOUSE-REINR	2.365000	261	0.00116831	0.000413695	3	2
426	CH-SHOCKHOUSE-REINL	2.365000	260	0.00116968	0.000409606	3	2
340	CH-CBN-CTR-CSL-P1	2.209000	182	0.000533915	7.22135e-006	3	2
339	CH-CBN-CTR-CSL-DASH	2.203000	181	0.000374083	1.68848e-005	3	2

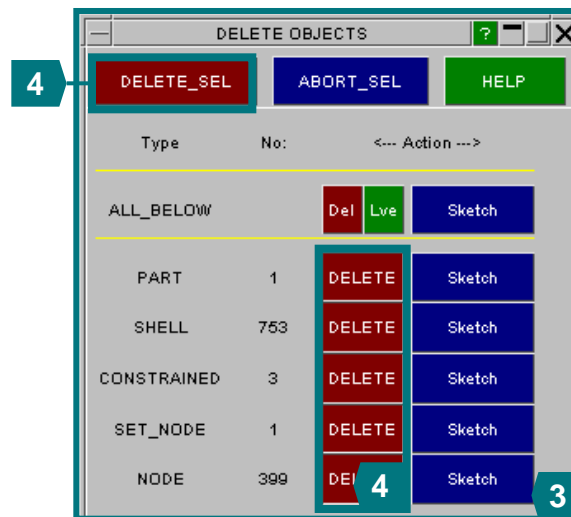
Change NIP
Sketch

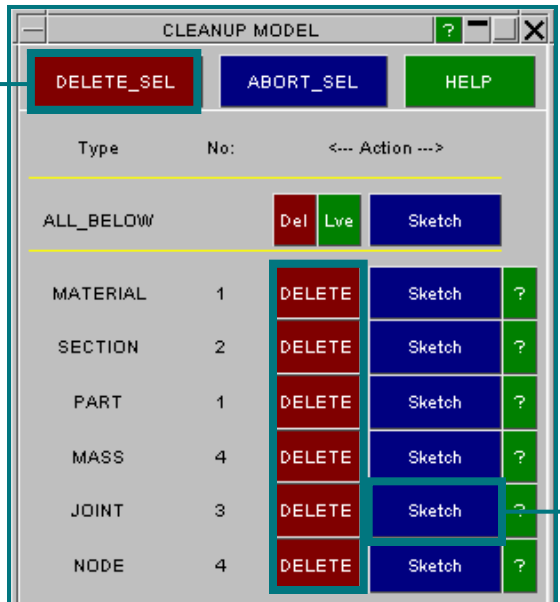
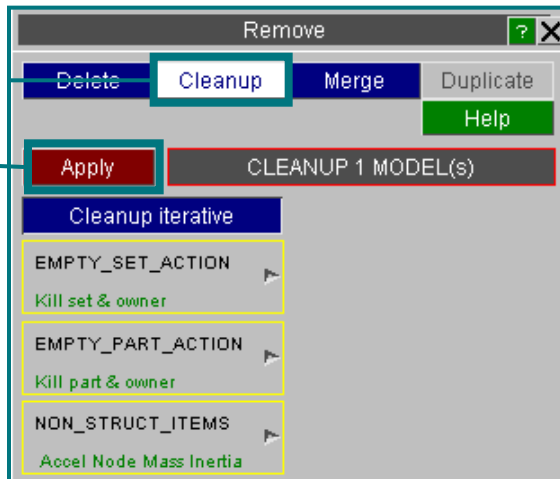


Remove

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**.



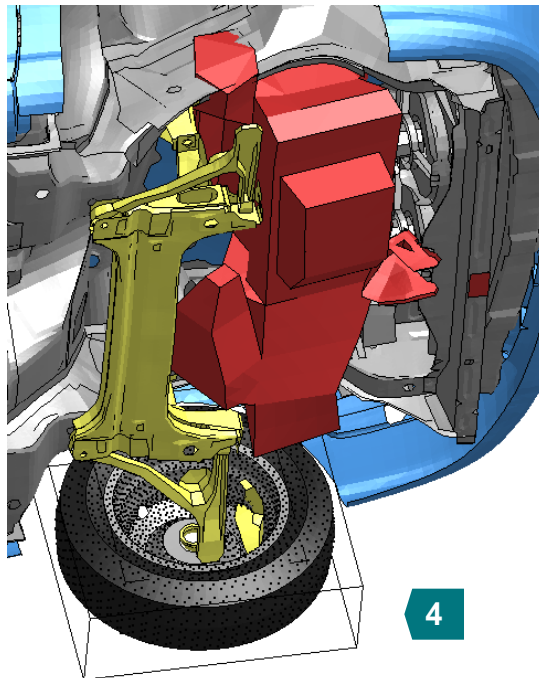
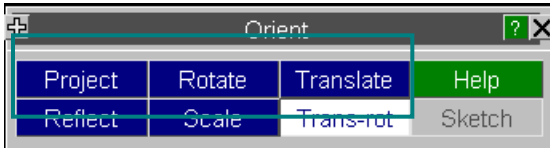


Remove (cont)

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**.

6 ways to Orient are available:

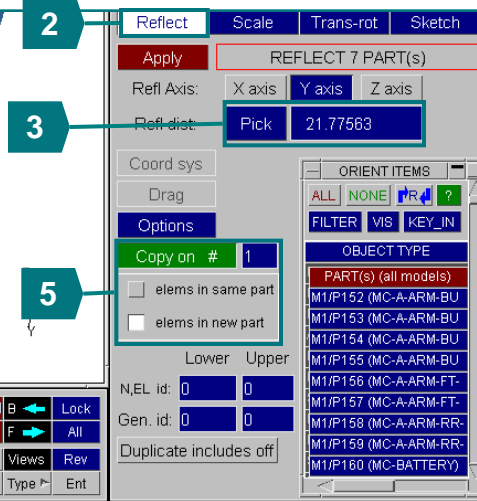


Orient

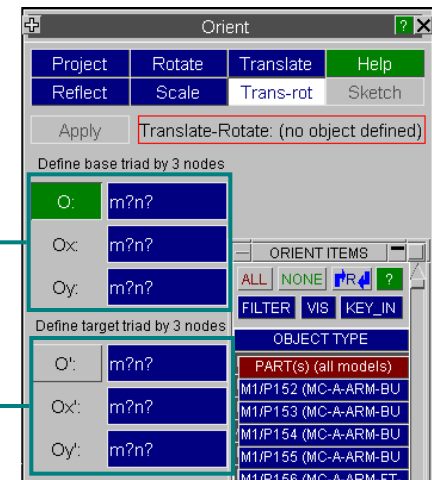
Using the **TOOLS – ORIENT** menu we can move any entity in the model to a different position and (optionally) make copies of the entity during the process

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, elems in new part** (some will be rigid)
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**



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

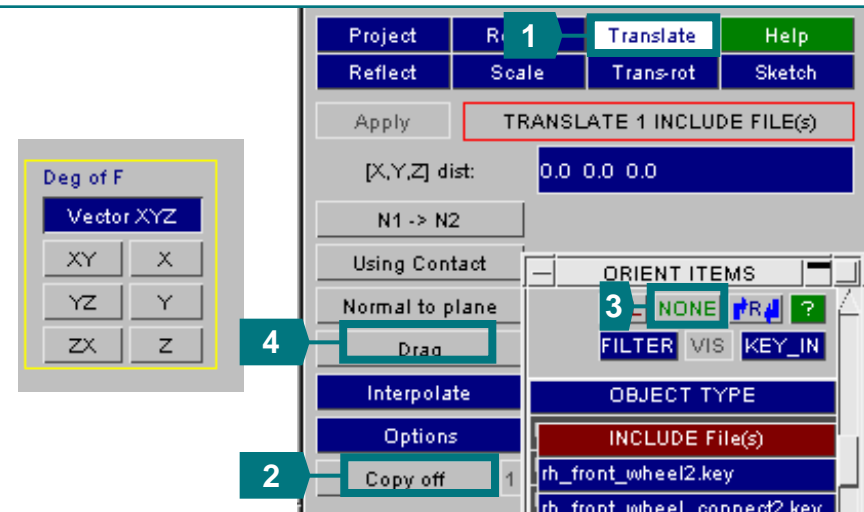
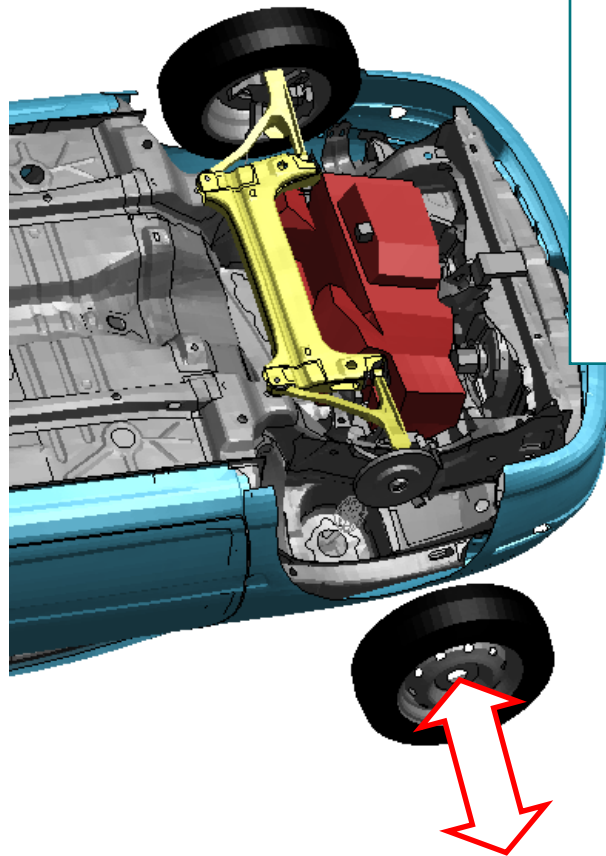


Orient

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

The screenshot displays the Oasys PRIMER software interface. The main window shows a 3D model of a blue car. The top menu bar includes File, Keywords, Tools, Display, Images, Viewing, Options, and a 'Pick NODE' dropdown. On the right side, there are several docked menus: 'Tools' (Assign ms, Attached, Blanking, BOM, Check, Clipboard, Coat part, Connection, Cut sect, Groups, Include, Measure, Mechanism, Meshing, Occupant, Orient, Other, Rigidify, Units, Xrefs, Remove), 'Keywords' (AIRBAG, ALE, BOUND, CONSTR, CONTACT, CONTROL, DAMPING, DATABS, DEFINE, DEF_2_R6, ELEMENT, EOS, HOURGL, INITIAL, INTEGRN, INTRFCE, LOAD, MAT, NODE, N_TFRM, PART, P_MODES, PERTURB, RAIL, RIGIDWL, SECTION, SET, PARAM), 'Model' (Model, Part tree, Part, Attached), 'CONTACT' (CONTACT, Measure), and 'Measure' (Point to Point, Point angle, Node to Node, Node angle, Nodal coord, Help). The 'Measure' menu is currently active, showing a 'Node to Node - pick nodes' dialog with 'Done' and 'Reject last' buttons. The bottom of the interface features a command line with 'Primer >' prompts and a toolbar with various icons for manual, stop, tidy, vector plot, init vels, AC, zoom, CN, B, F, Views, Type, Lock, All, Rev, and Ent.

Tabs

- 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.
- Clicking on a Tab brings that menu to the front.
- 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.

The screenshot shows the 'MODIFY PART M1/P156' dialog box with the 'VIEW_XREFS' button highlighted. To the left is a tree view of the model hierarchy. To the right is the 'Cross reference viewer' showing two panels: 'Referred to by' and 'Refers to'. The 'Referred to by' panel lists entities like '156 (MC-A-ARM-FT-L)', 'SHELL', and 'CONSTRAINED'. The 'Refers to' panel lists '156 (MC-A-ARM-FT-L)', 'MATERIAL', and 'SECTION'. Below the panels are two callouts: 'Other entities that refer to this one' and 'Entities referred to by this one'.

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

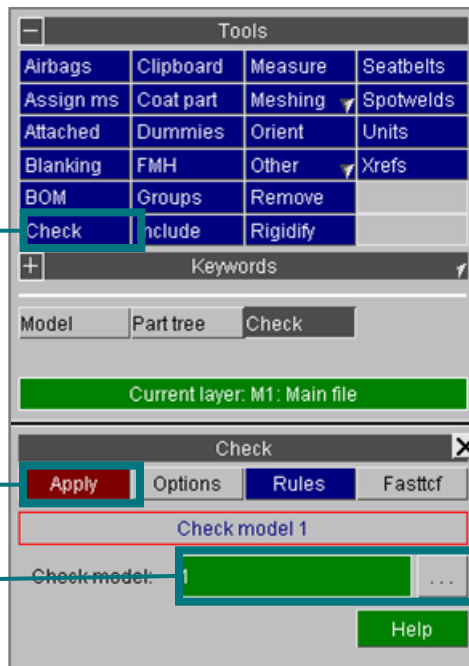
4. The right menu contains entities that this part refers to – right click & open – we can see that only 1 parts references this material

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



3

CHECK DYNA3D MODEL

DISMISS RECHECK AUTOFIX TREE HELP

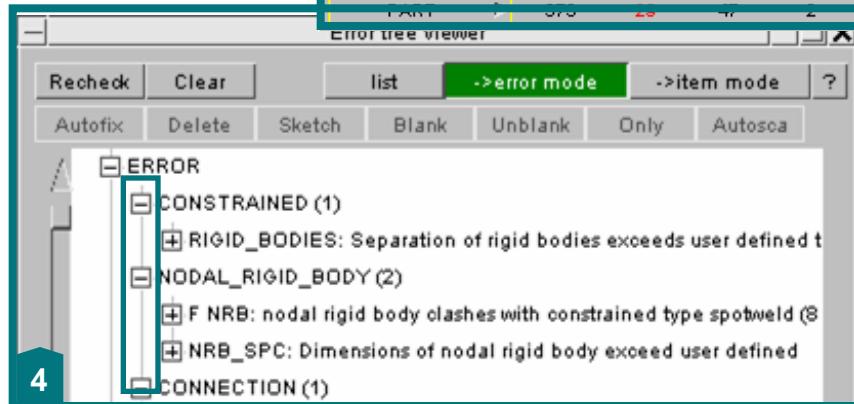
Check (model 1)

Total no		427	4180	2609
Entity type	(No.)	# errors	# warn	# fixable
AIRBAG	4	0	0	0
CONSTRAINED	1391	18	0	8
CONNECTION	2588	0	2588	2588
CONTACT	3	1	0	0
CONTROL	10	0	1	1
DATABASE	27	0	0	0
DEFINE	19	0	0	0
ELEMENT	537779	364	1450	1
HOURGLASS	3	0	0	0
INCLUDE FILE	3	0	0	0
MATERIAL	321	12	92	7
NODE	296419	3	2	2

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 **[-]**



Read the descriptions of the different errors...

- ERROR
 - CONstrained (1)
 - RIGID_BODIES: Separation of rigid bodies exceeds user defined toler **User-defined check to catch possible modelling error**
 - NODAL RIGID BODY (2)
 - F NRB: nodal rigid body clashes with constrained type spotweld (8) **"multiple constraint" error**
 - NRB_SPC: Dimensions of nodal rigid body exceed user defined maxit **User-defined check to catch possible modelling error**
 - CONNECTION (1)
 - Connection is invalid (77) **Faults with connections can be fixed using Connections Table**
 - CONTACT (1)
 - Constrained type SPOTWELD contact does not allow rigid parts (1) **"multiple constraint" error**
 - DEFINE_CURVE (1)
 - Load-curve referenced but not defined (1) **Missing data – will stop LS-DYNA**
 - BEAM (1)
 - spotweld is too close to another on same part (351) **(Spotweld beam) quality checks**
 - MATERIAL (3)
 - MAT_24/123: <LCSS> curves starts at yield stress $Y \leq 0.0$ (4)
 - MAT_24/123: <LCSS> curve does not begin at strain $X = 0.0$ (6)
 - MAT_24/123: table curves cross at strain $< EPPF$ (1)
 - NODE (2)
 - node on rigid part clashes with constrained type spotweld (3) **"multiple constraint" error**
 - F node is not in same include as element (2)

Recheck Clear show tags list ->error mode
Autofix Delete Sketch Blank Unblank Only Autosca

Read the descriptions of the different warnings...

ERROR

WARNING

CONTROL (1)

F SHELL: Sorting flag for degenerate quads <itrist/esort> is not = 1 (1)

Recommended parameter is not the default

BEAM (3)

(spotweld) beam length is greater than maximum allowed size (1014)

(Spotweld beam) element quality checks

Beam is part of a spotweld which is longer than maximum allowed length (361)

(spotweld) beam length is less than minimum allowed size (74)

SHELL (1)

F Duplicate Shell - overlaps existing shell of same part (1)

MATERIAL (2)

F MAT_24/123: yield stress from <LCSS> curve over rules SIGY (6)

Material type 24 - warnings

MAT_24/123: curve discretisation may lose data (86)

PART (4)

elements of rigid part are not continuously meshed (32)

Discontinuous rigid body – could be a mistake?

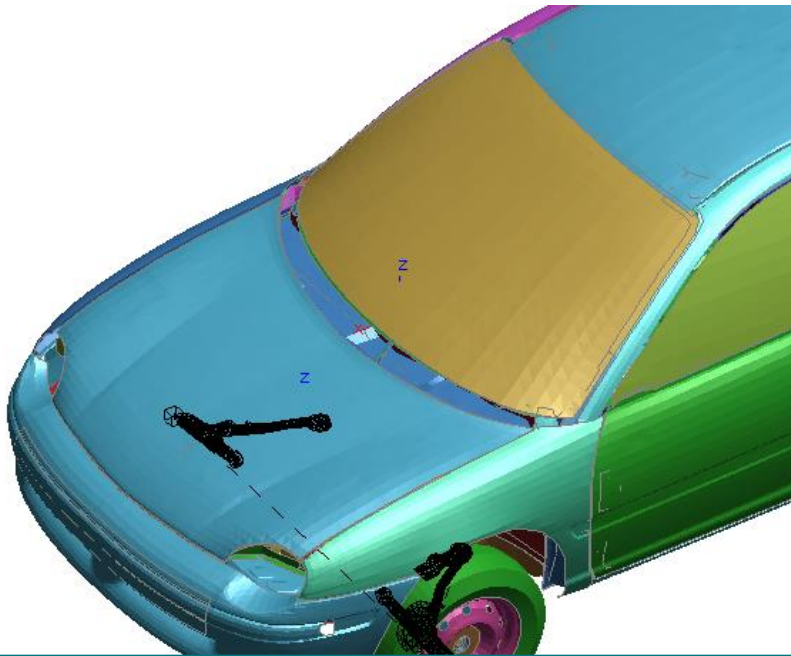
F Part contains elements in different include file (1)

Model badly organised

%age added mass on part exceeds allowed percentage (12)

%age added mass on spotweld part exceeds allowed percentage (2)

User-defined quality check – mass-scaling



Checking with the Error Tree Viewer:

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

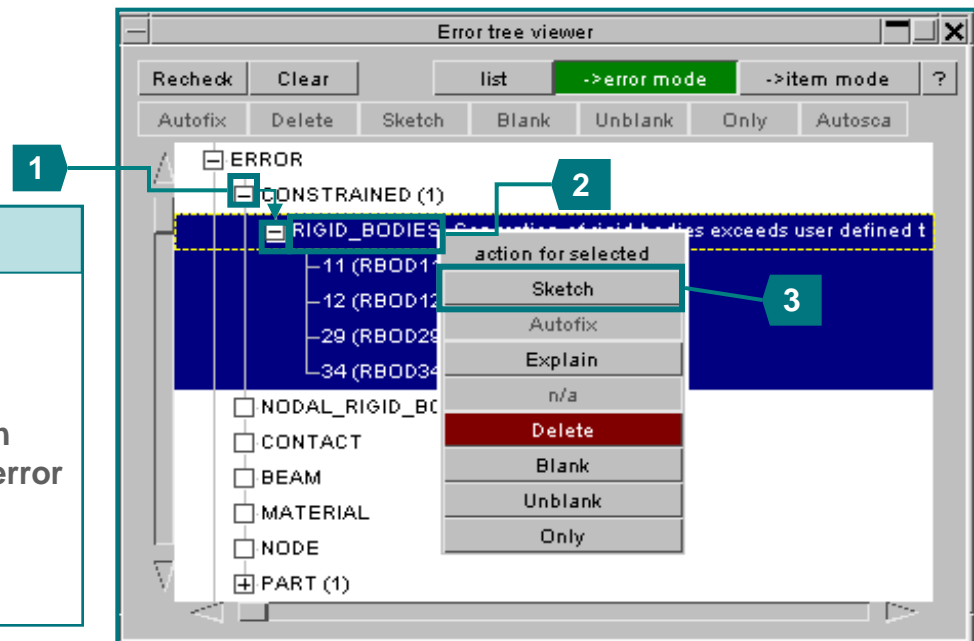
This is a *user-defined check* for the distance between merged rigid bodies – it could show up a modelling error that the wrong rigid bodies were merged.

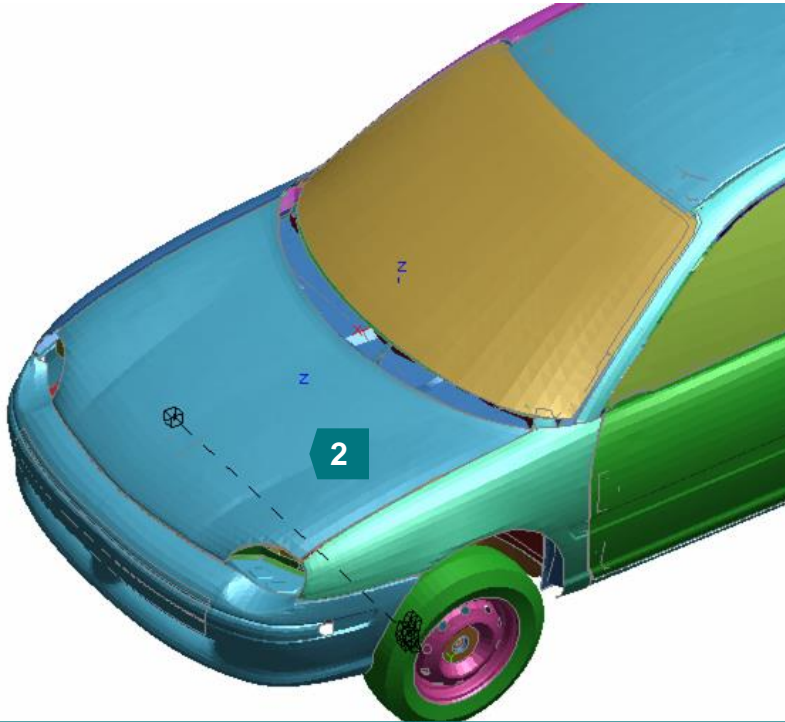
Later we will learn how to set preferences for user-defined error checks.

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*.

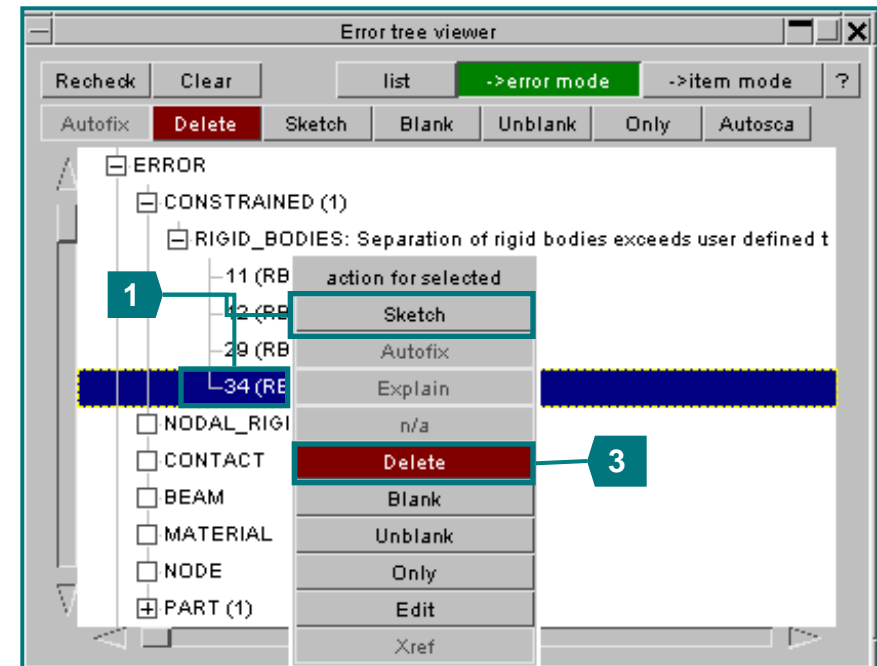
Some of these may not be errors – it depends on user defined values – you need to consider each one separately.





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**.



MODIFY MATERIAL M1/MAT226

Include: I4 body_with_welds4.key

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

Type: MAT_024: PIECEWISE_LINEAR_PLASTICITY

Title: <No material name given>

Row\Col	1	2	3	4
1	<Label>	RO	F	E
2	226	7.89E-9	210000.0	0.3
3	C	F	P	LCSS
4	0.0	0.0	320057	0
5	EPS1	F	EPS2	F
6			EPS3	F
7			EPS4	F

MODIFY DEFINE_TABLE M1/TABL320057 (for MATERIAL 2)

Include: I7 misc_materials3.key

Modify DEFINE_TABLE 320057 (model 1)

Stress-strain curve plot showing Stress (Y-axis, 0 to 350) vs Strain x 1.0E-3 (X-axis, 0 to 450). The plot shows three curves: a red curve (highest stress), a green curve (middle stress), and a yellow curve (lowest stress). The curves cross at high strain values.

More Checking & Fixing:

- Right-click on the last **MAT_24** error. Click on **Edit**. This will open the **MODIFY MATERIAL** menu for MAT 226
- 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.

Error tree viewer

Recheck Clear

Autofix Delete Sketch

ERROR

CONSTRAINED

NODAL_RIGID_BODY

CONTACT

BEAM

MATERIAL (3)

MAT_24/123: 226 0

MAT_24/123: 226 0

MAT_24/123: 226 0

226 0

NODE

PART

WARNING

MODIFY MATERIAL M1/MAT226

ABORT_MODIFY RESTORE_ORIGINAL HELP

UPDATE_MATERIAL COPY_EXISTING SKETCH

VIEW_XREFS CHECK_DEFN 4

Include: 14 body_with_welds4.key

Modify material M1/MAT226

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

Type: MAT_024: PIECEWISE_LINEAR_PLASTICITY

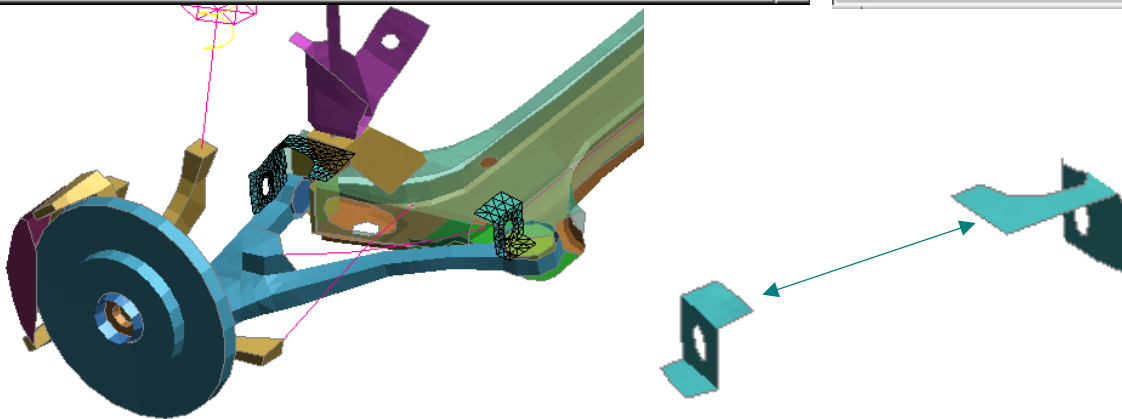
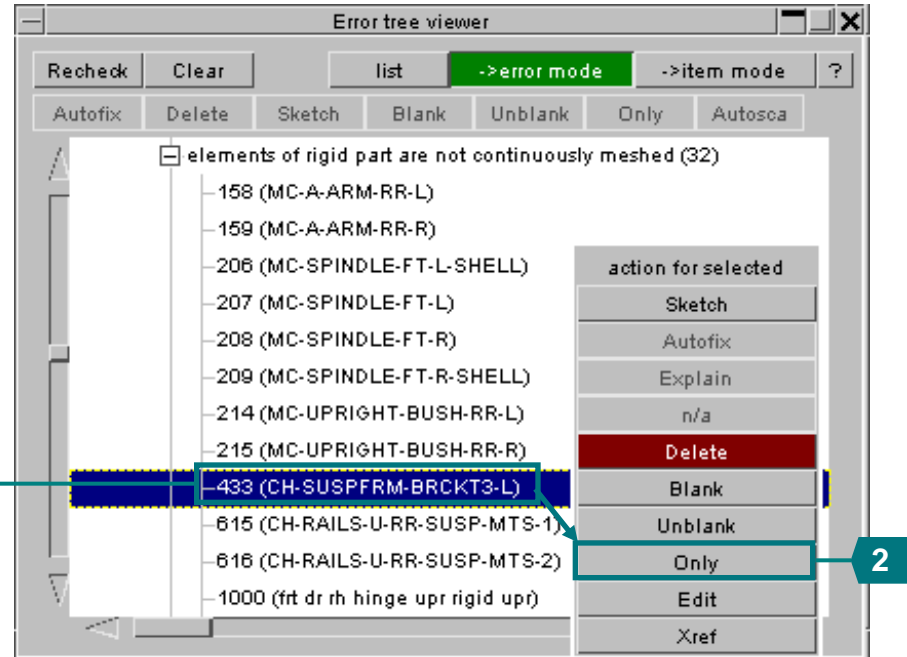
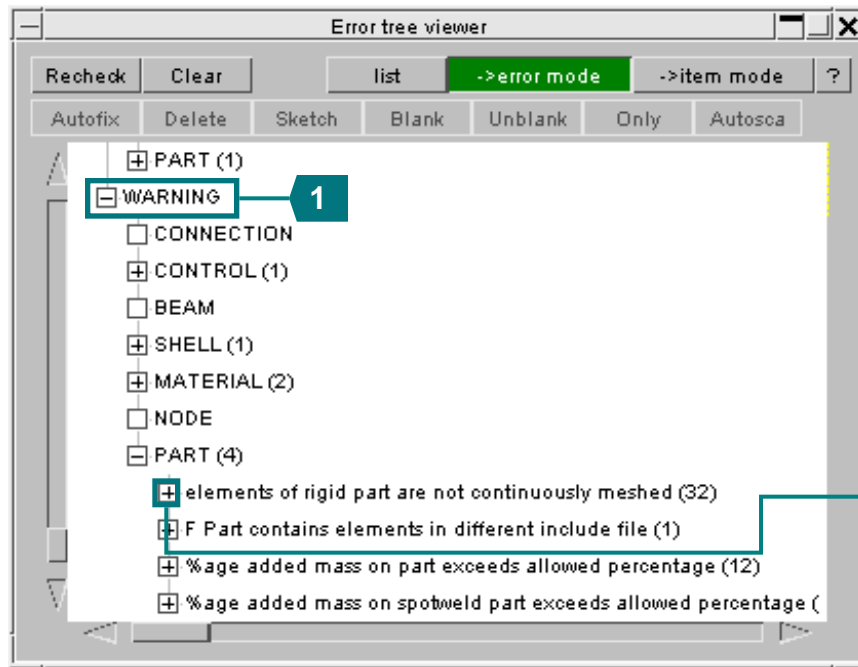
Title: <No material name given> 5

IMPORT... From database

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 +LC	LCSR +LC	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

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_DEFN** button
5. Erase the **SIGY**=400 – this is ignored by Dyna when the stress-strain curves start at a different value.
CHECK_DEFN again – should be no errors for MAT 226

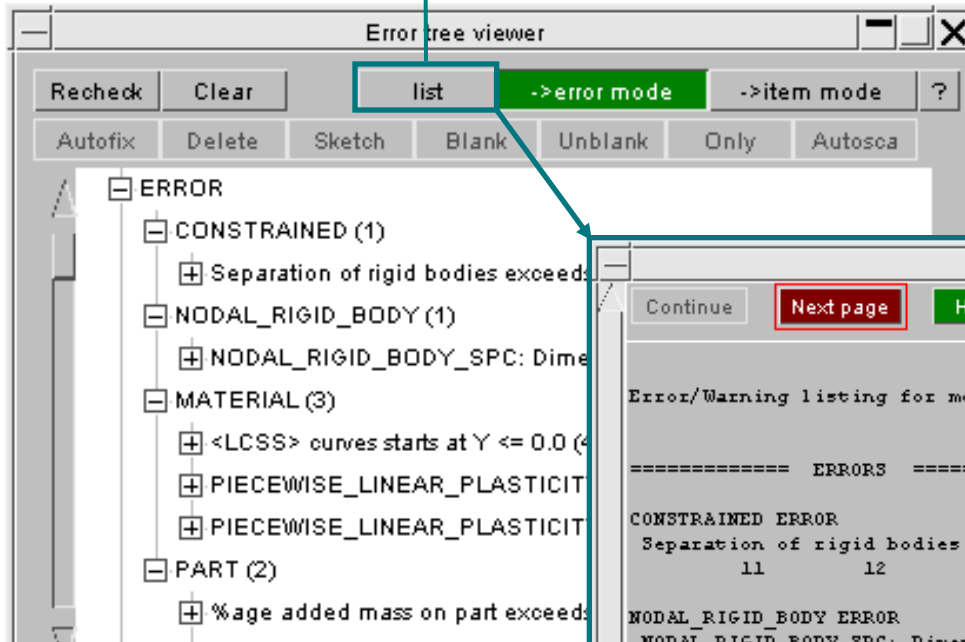


More Checking:

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.*

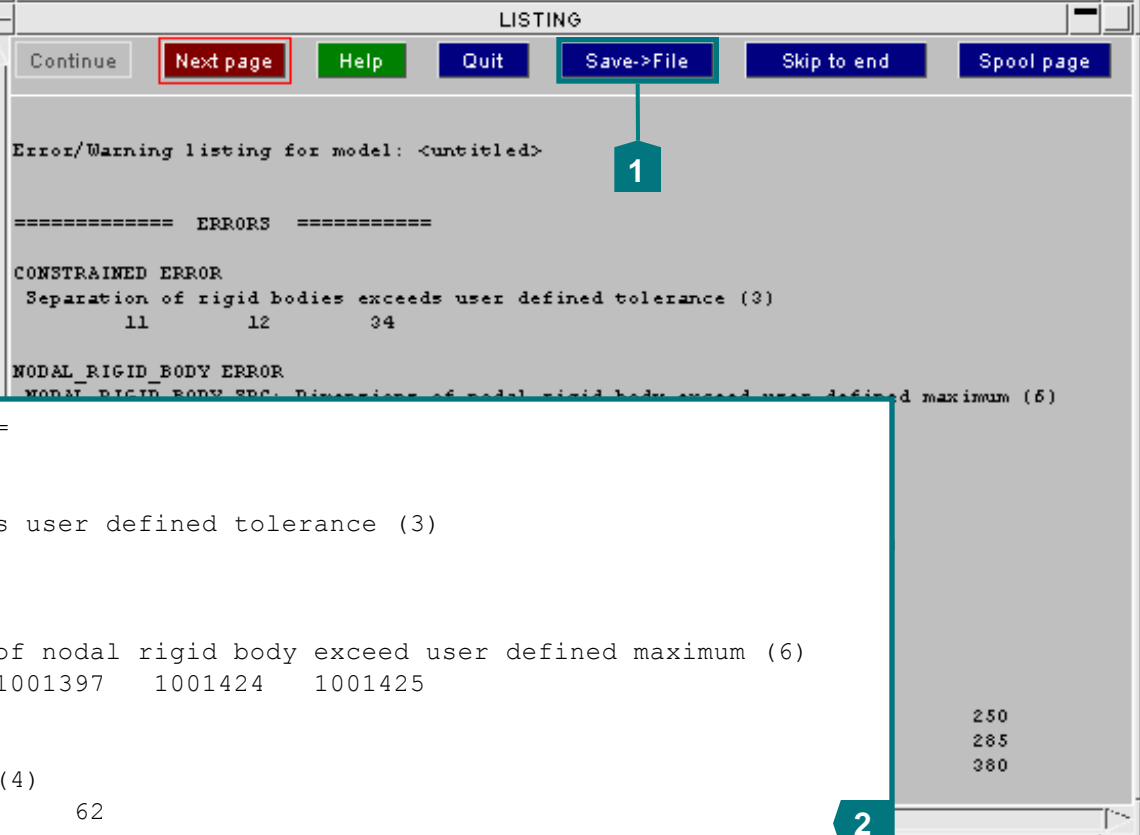
Writing an error summary file

1



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.*



```
$ ===== ERRORS =====
$
$ CONSTRAINED ERROR
$ Separation of rigid bodies exceeds user defined tolerance (3)
$      11      12      34
$
$ NODAL_RIGID_BODY ERROR
$ NODAL_RIGID_BODY_SPC: Dimensions of nodal rigid body exceed user defined maximum (6)
$ 1001388 1001389 1001396 1001397 1001424 1001425
$
$ MATERIAL ERROR
$ <LCSS> curves starts at Y <= 0.0 (4)
$      59      60      61      62
```

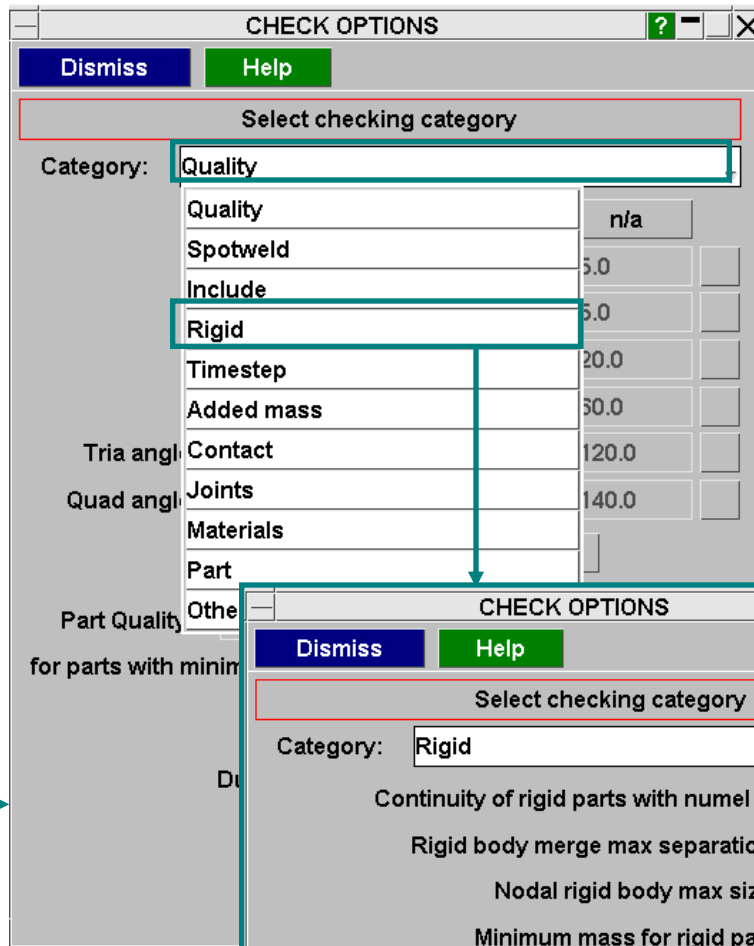
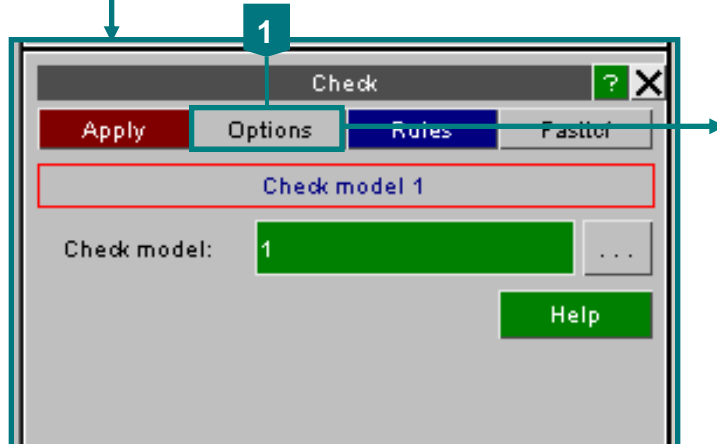
2

User-defined checking options

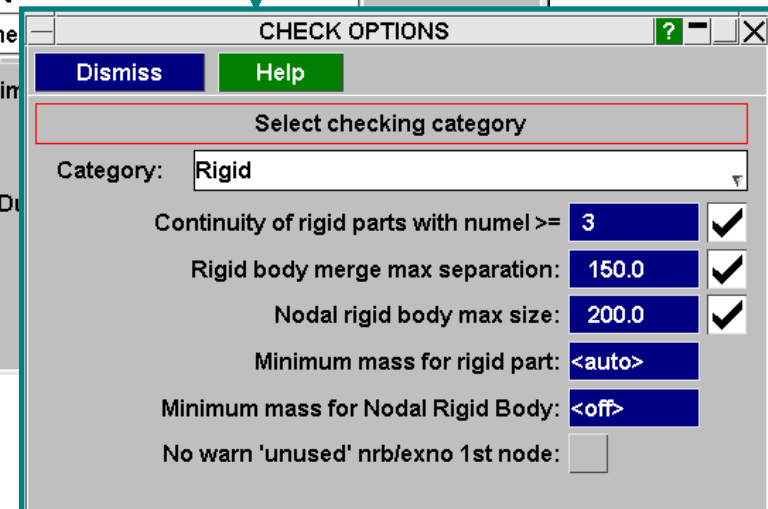
User Defined Options:

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*

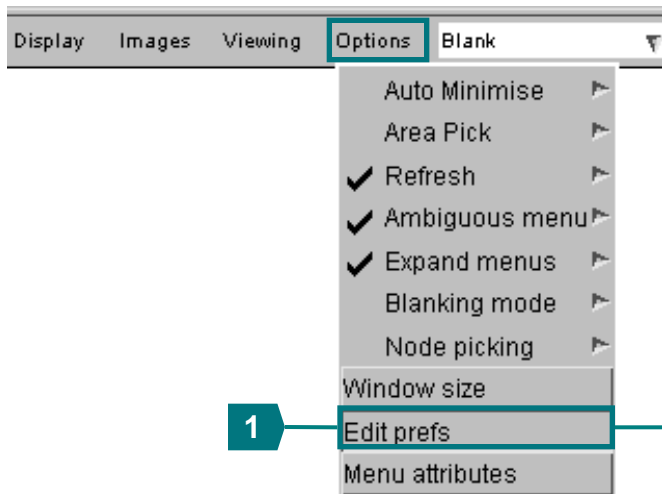
Airbags	Clipboard	Measure	Seatbelts
Assign ms	Coat part	Meshing	Spotwelds
Attached	Dummies	Orient	Units
Blanking	FMH	Other	Xrefs
BOM	Groups	Remove	
Check	Include	Rigidify	



Use drop-down to change category

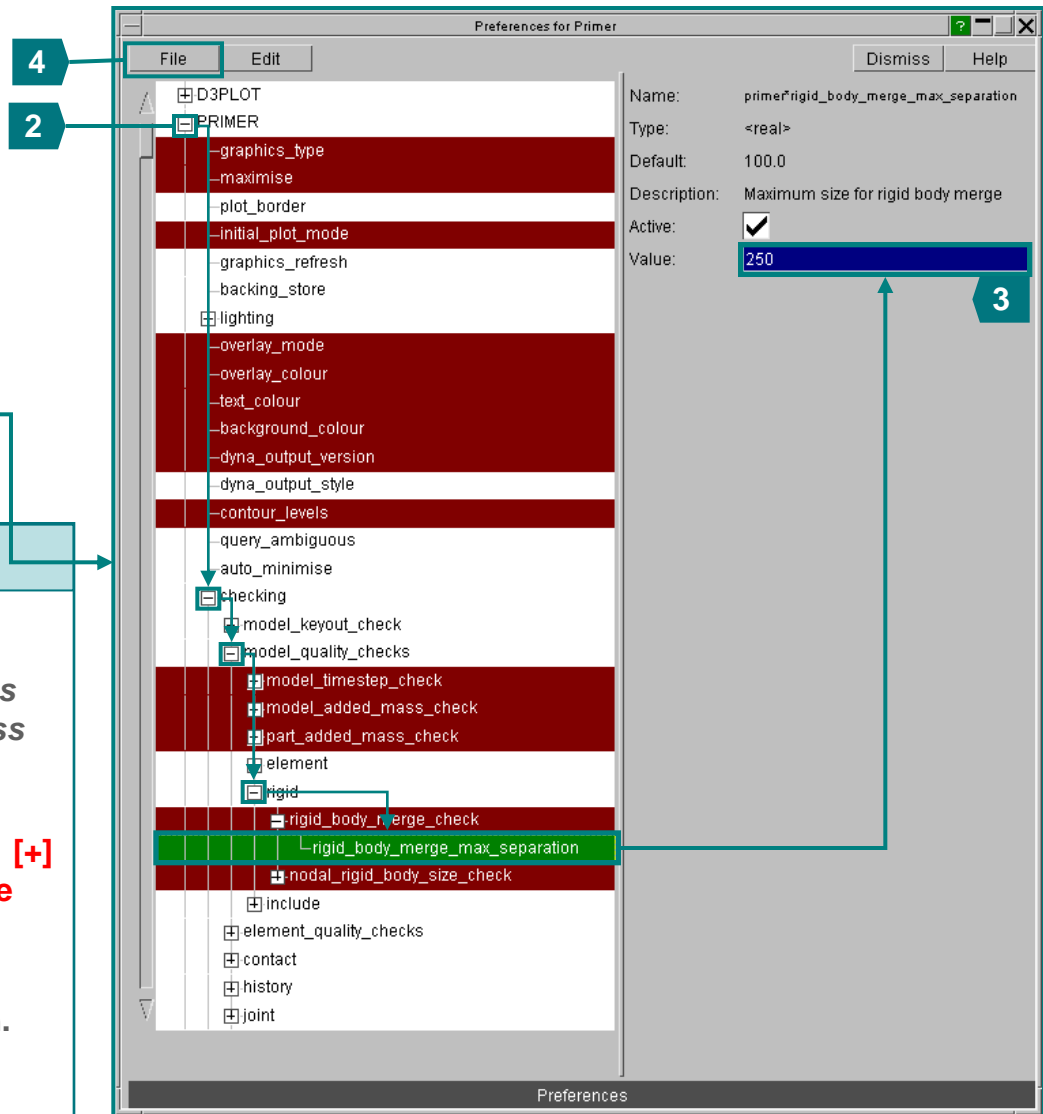


User-defined checking options



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.*



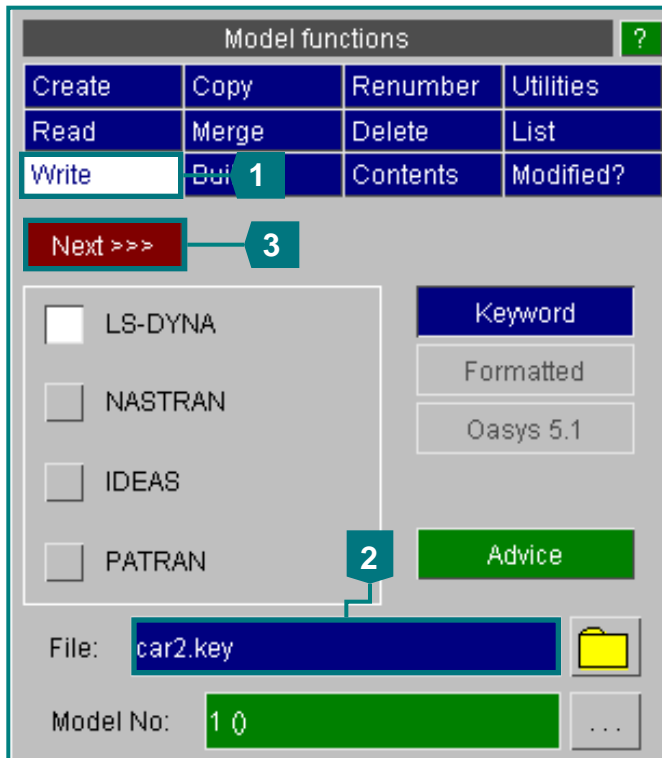
- PRIMER has a further capability for customising error checking.
- A user-written file can change the status of selected errors, and add extra messages which will appear in the error log file. The user's scripts can then scan the error file for these messages and take action accordingly.

Example file:

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

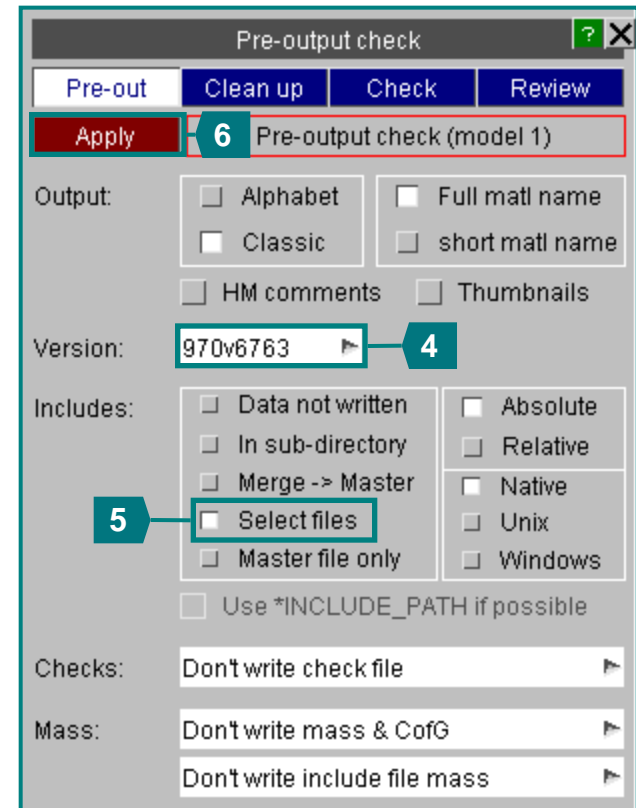
↑
Error tags – identifying the
selected error. The tags can be
made visible in the error tree.

Writing out the corrected model

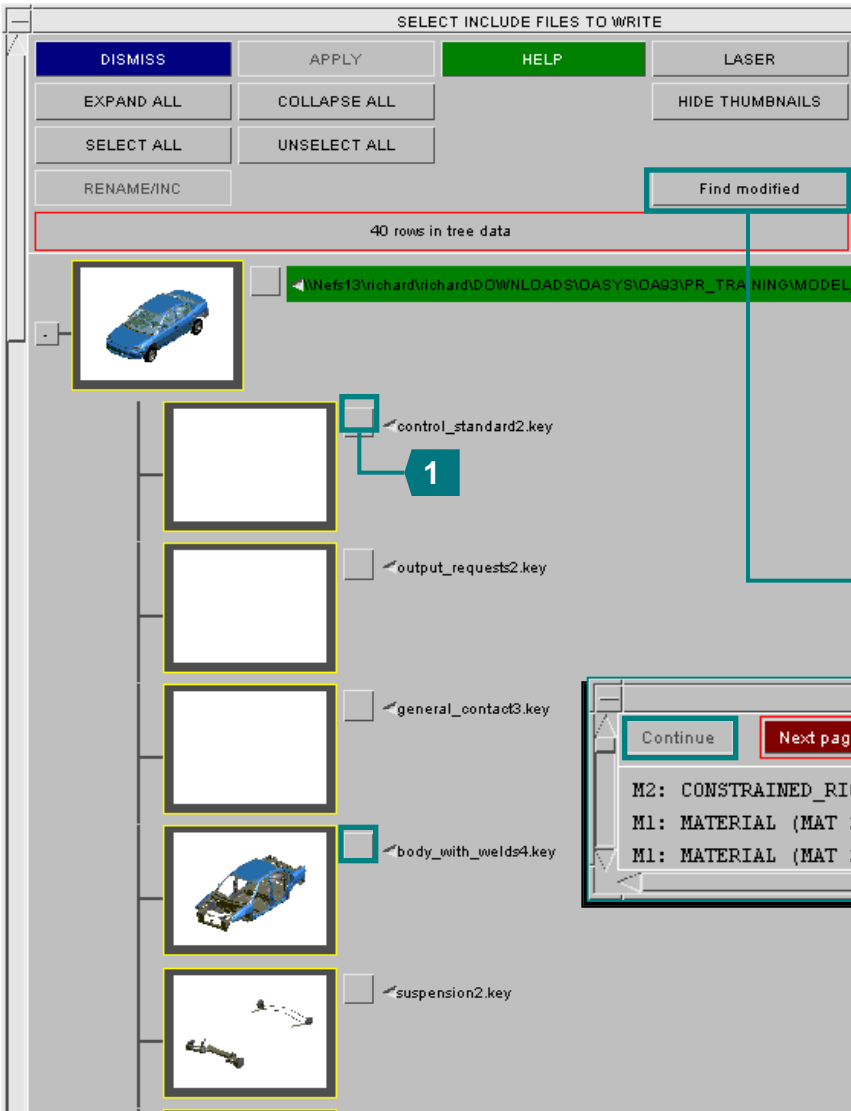


Model Write:

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**

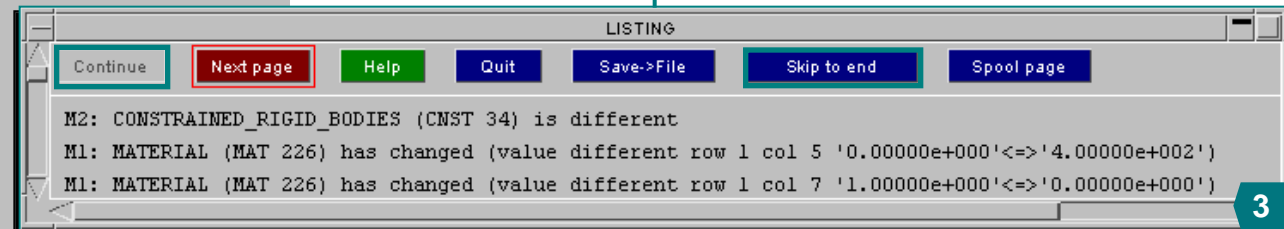


Writing out the corrected model



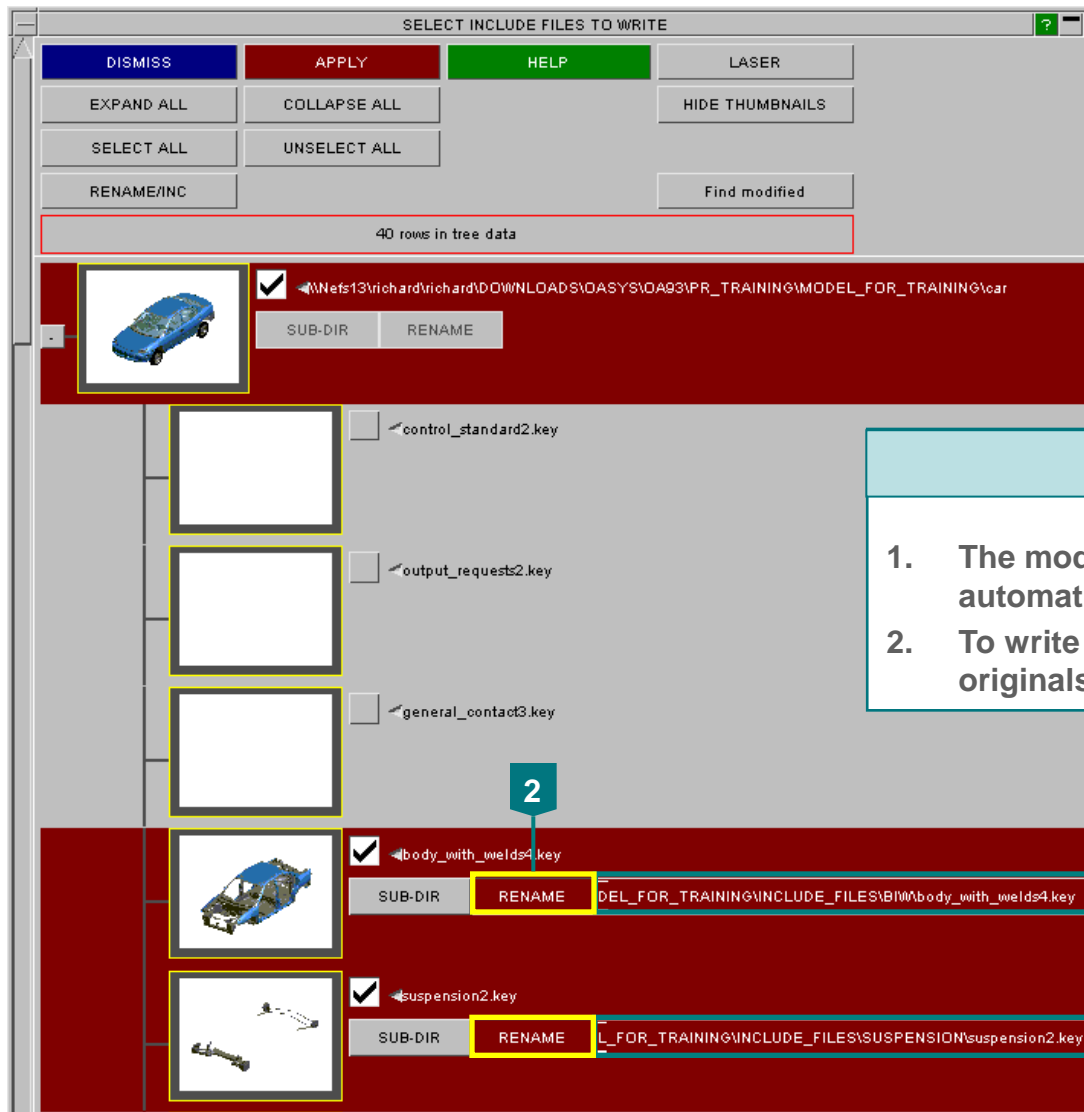
Model Write contd.:

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



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

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

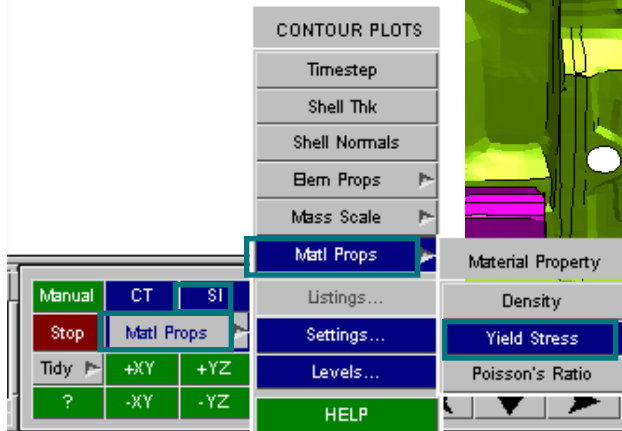
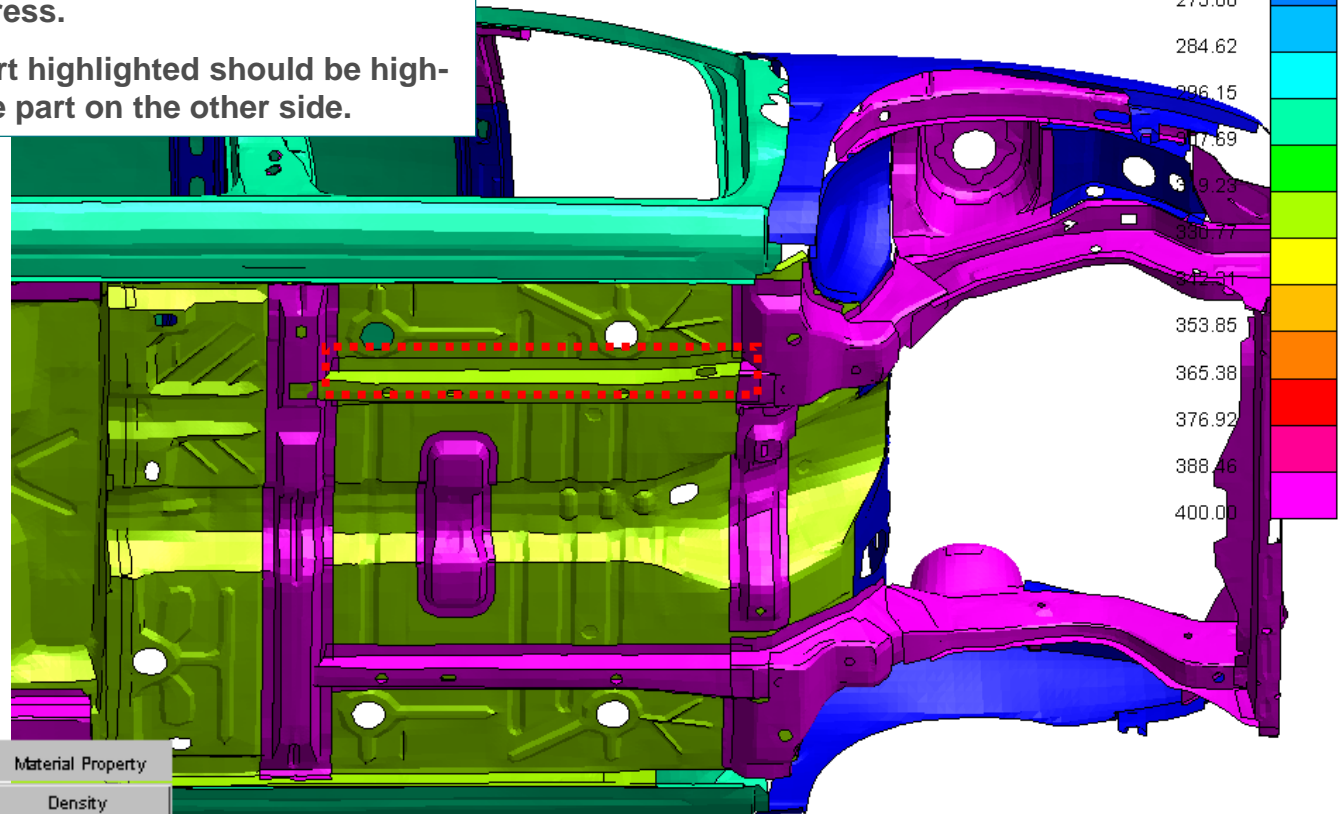
Visual Checking - Contouring

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.

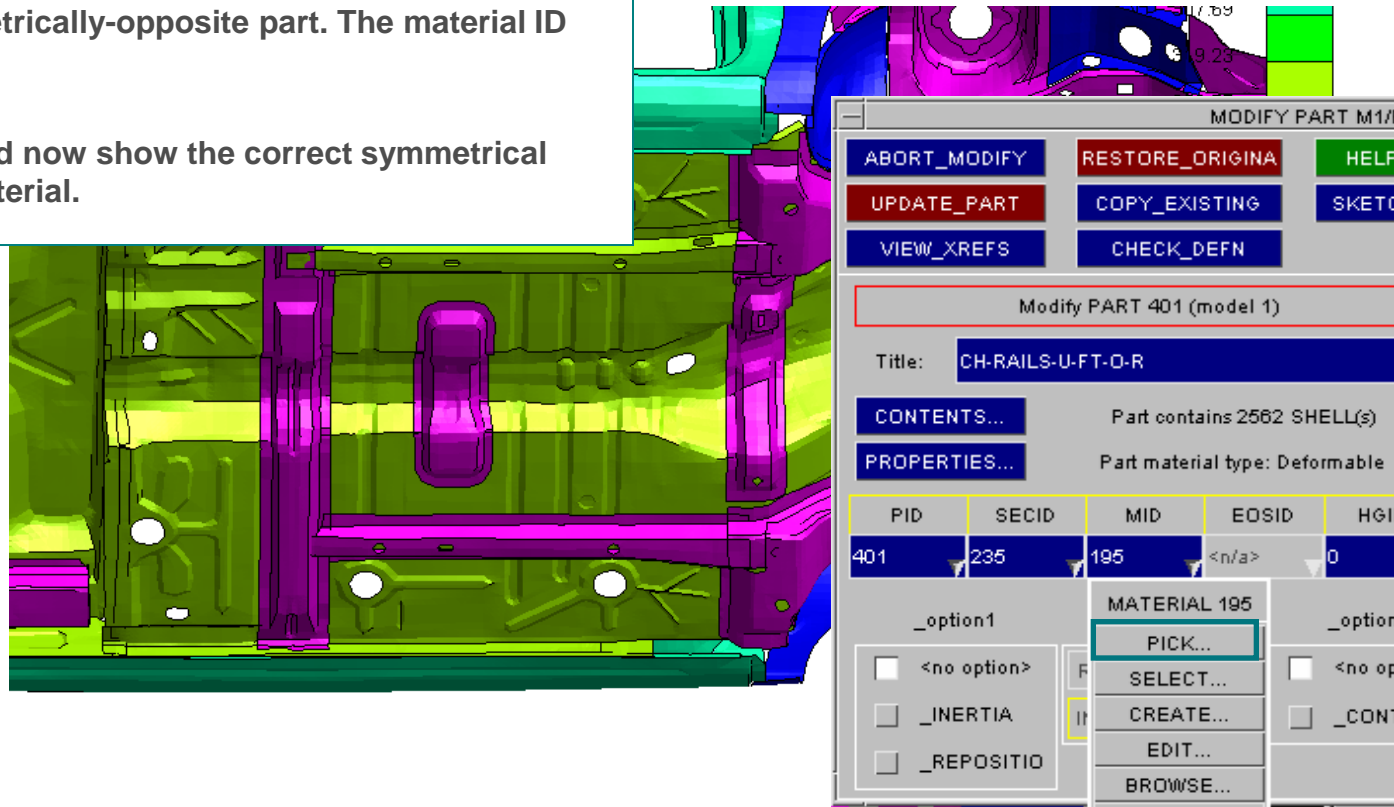
Can you see the error? The part highlighted should be high-strength steel – purple, like the part on the other side.



Visual Checking - Contouring

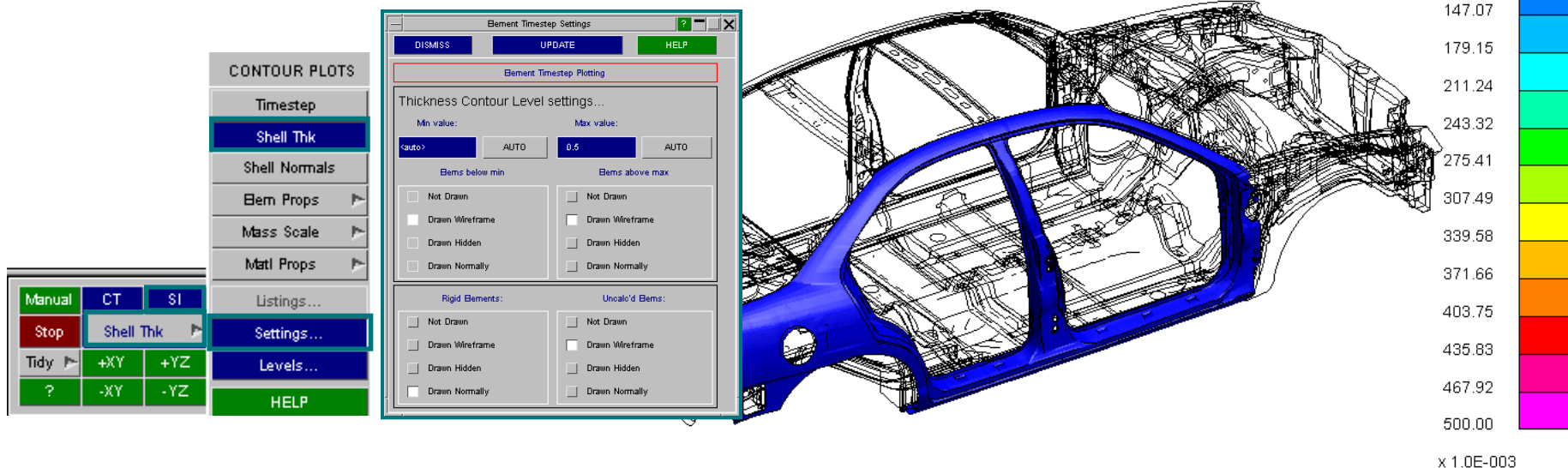
Edit PART

- 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.



Thickness contouring:

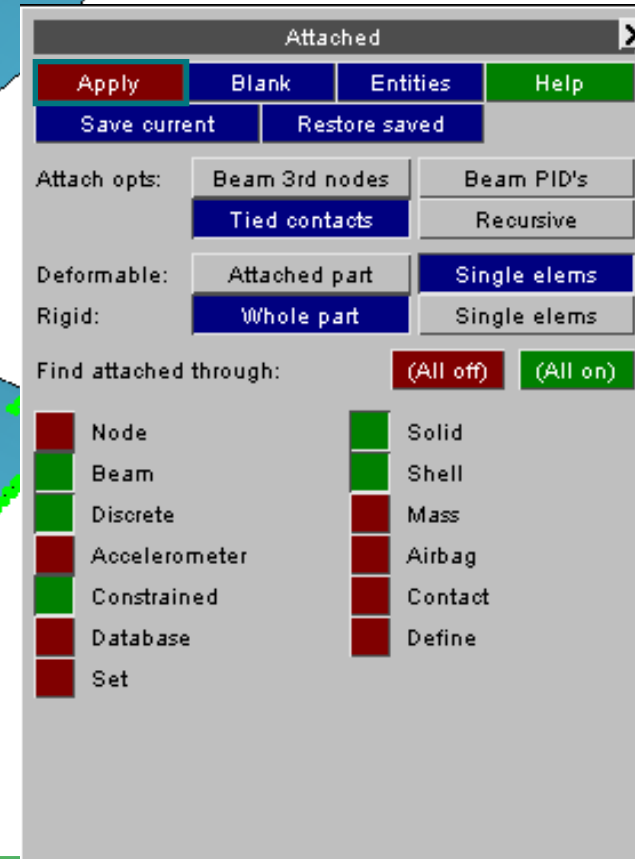
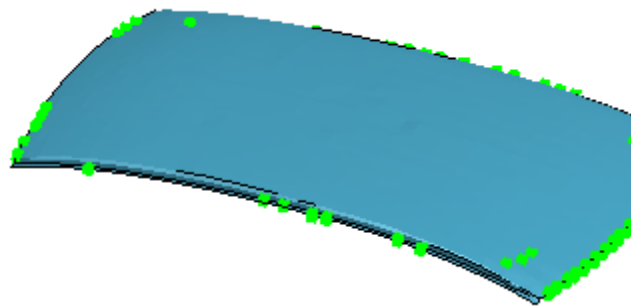
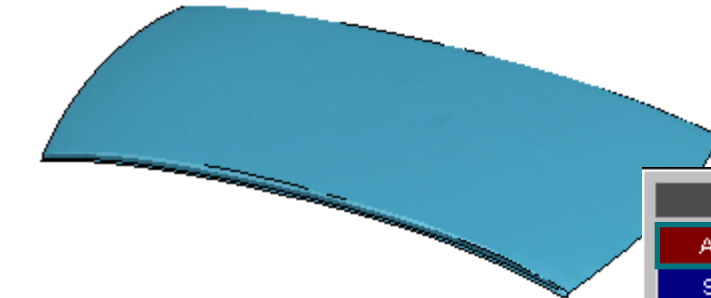
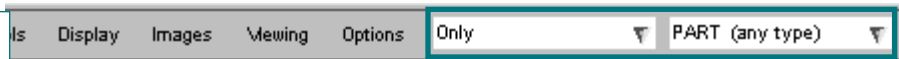
1. Ensure that only the body structure is visible.
2. Select component Shell Thk, do an SI plot
3. Note the contour levels – somewhere there is a panel with a thickness of only 0.083mm – this is probably an error.
4. To find it, set the contour levels: press Settings..., enter a contour maximum value 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.

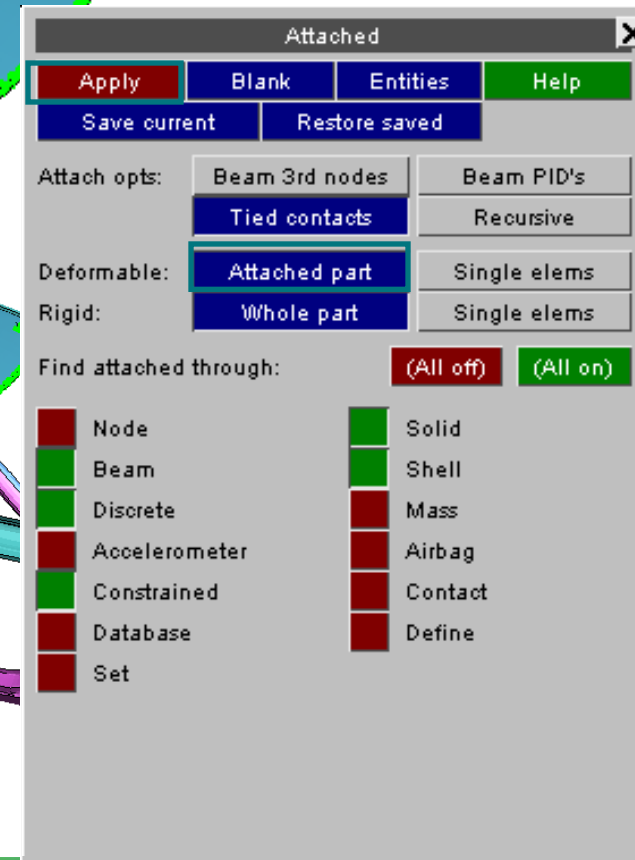
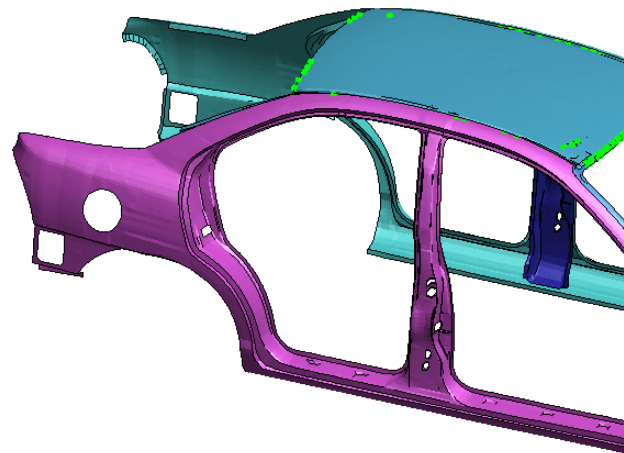
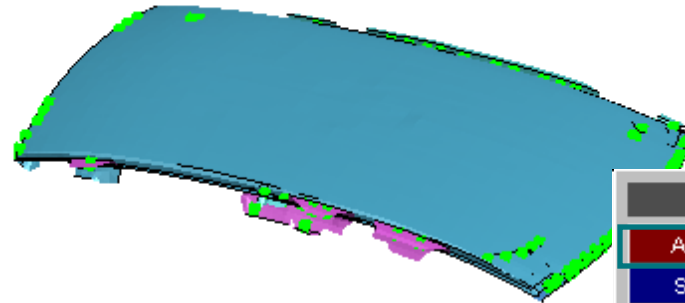
Under Tools, press **ATTACHED**. In the attached menu, press **APPLY**. PRIMER finds and displays any entities joined to the entities already displayed – in this case, mesh-independent spotwelds that are joined by a tied contact. PRIMER uses the same thickness tolerances etc as LS-DYNA to decide which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

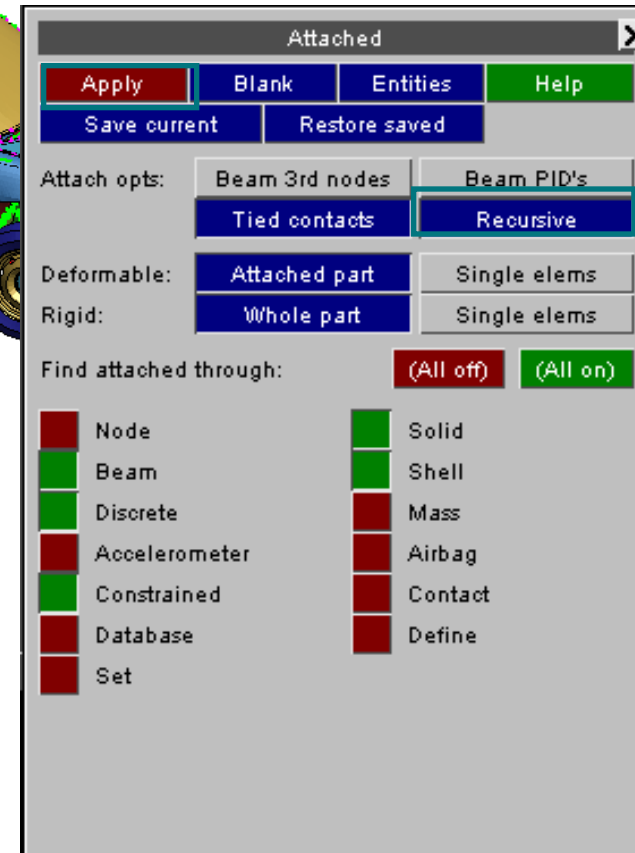
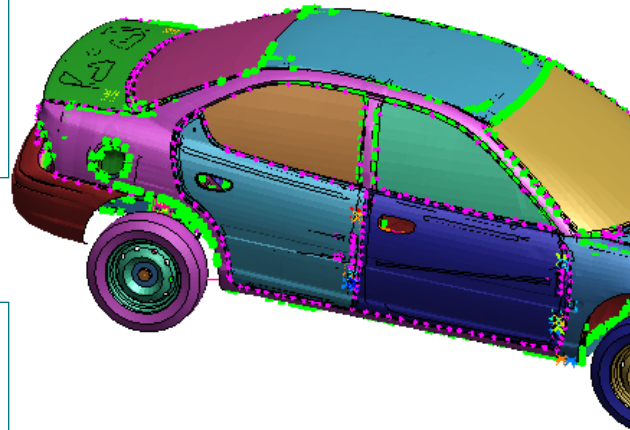


Visual checking – Find Attached



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.

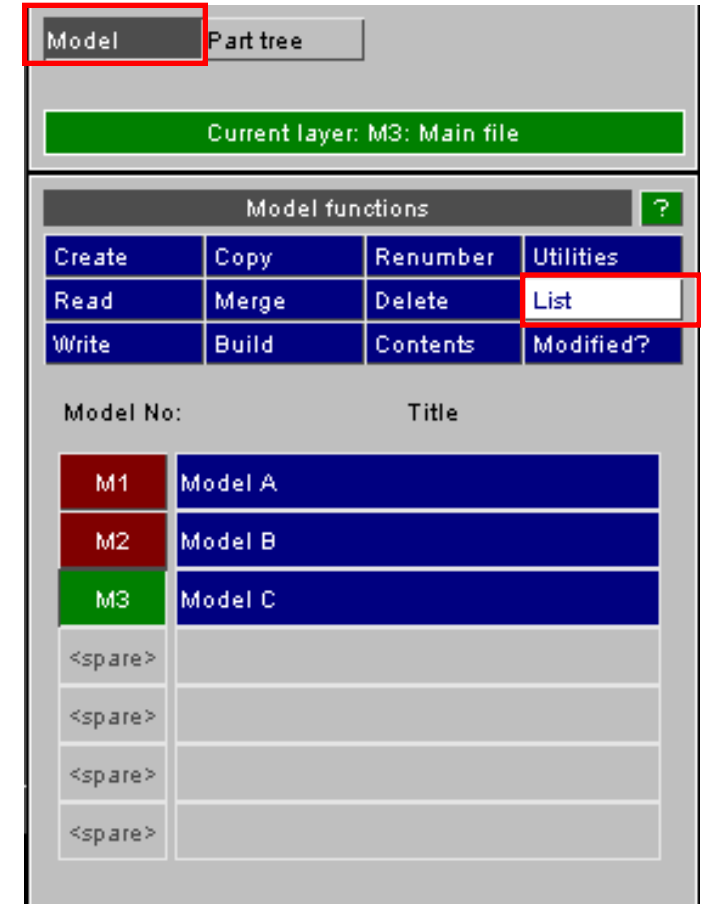


Working with multiple models and merging models

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.

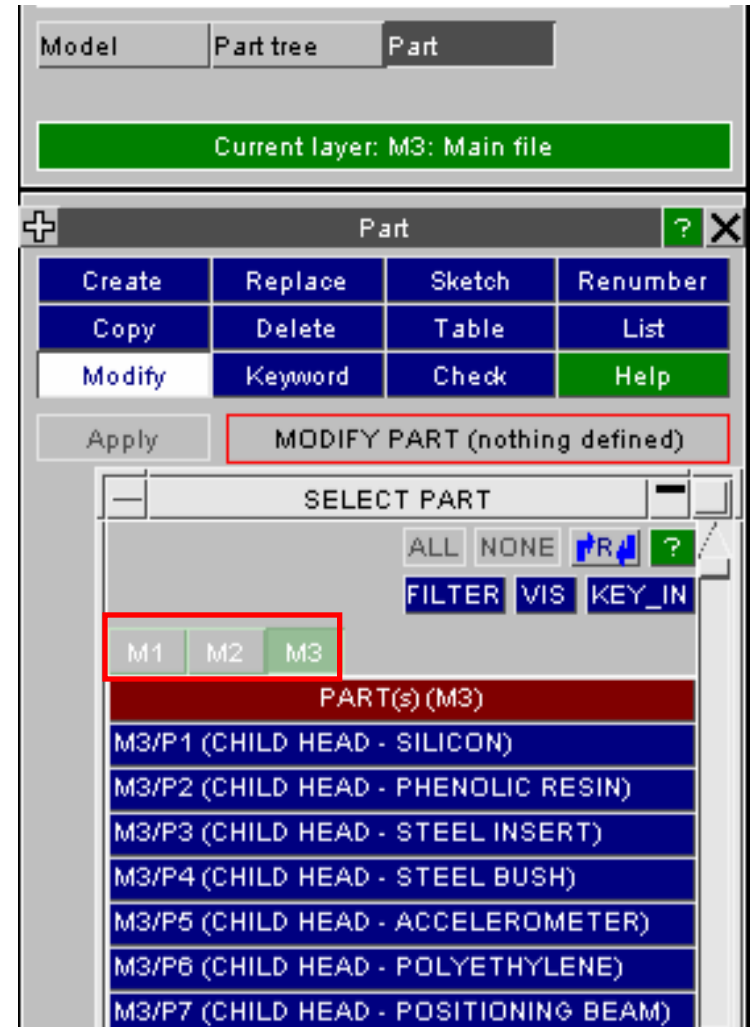


Working with Multiple Models

Go to Keyword=>Part=>Modify.

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

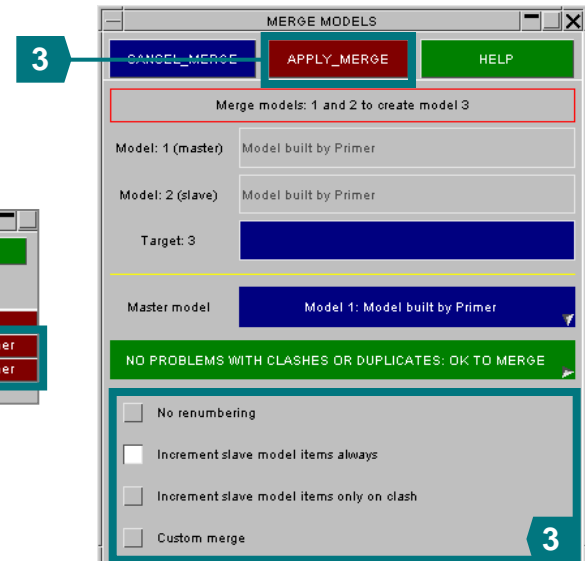
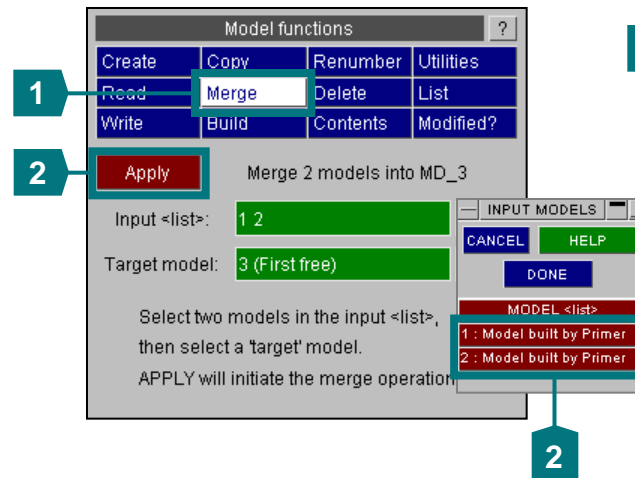
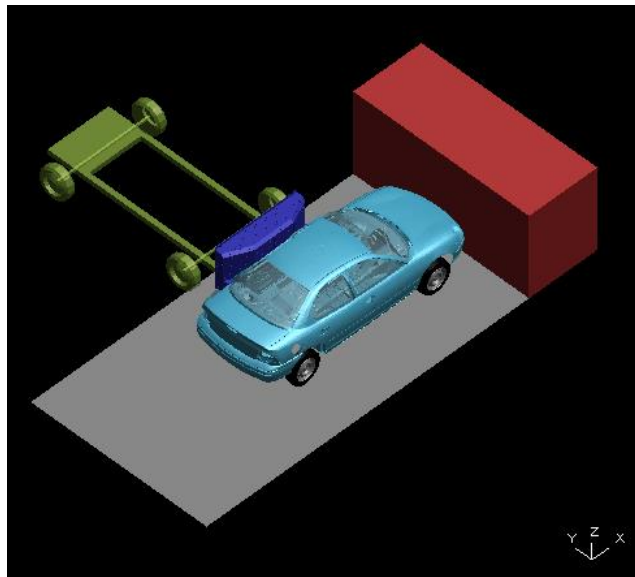
Try clicking the tabs, see the effect on the list of parts offered.



Merging Models

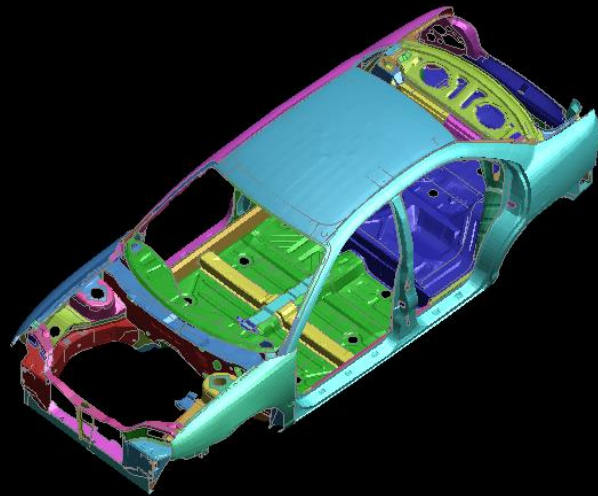
Multiple models can be read into PRIMER and kept separate unless the user merges them.

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.

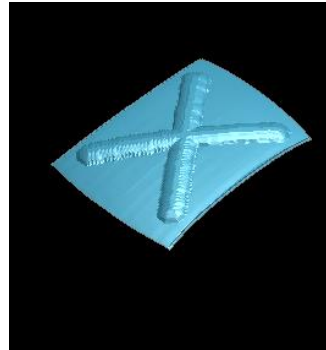


Replace Part

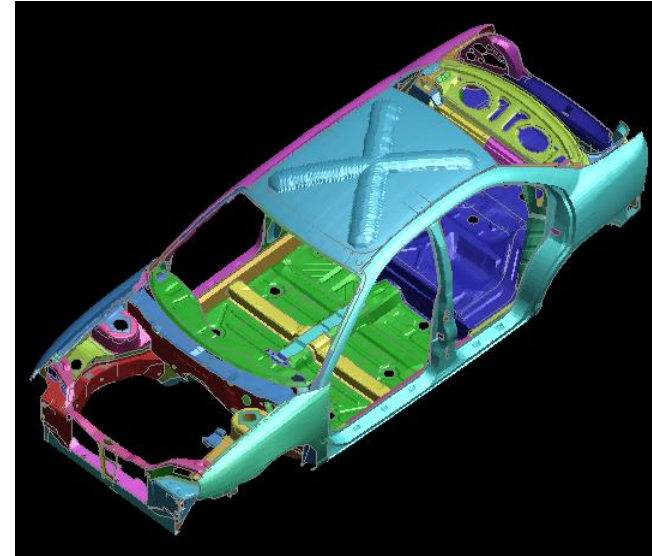
Replace Part



+



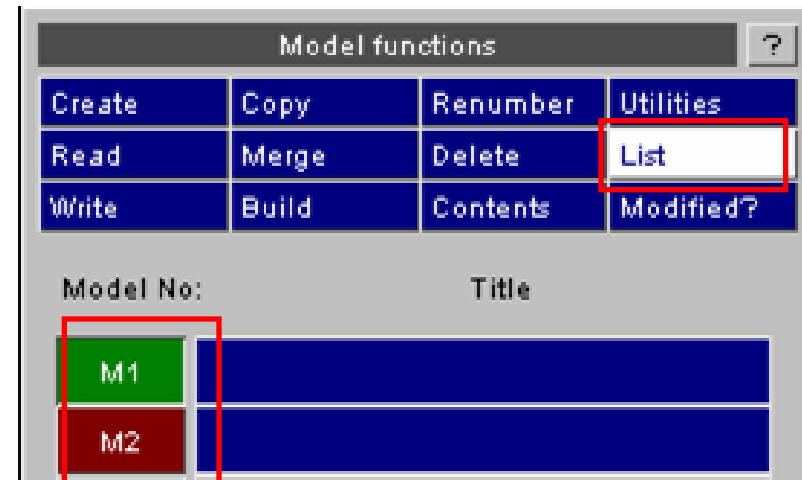
=



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.



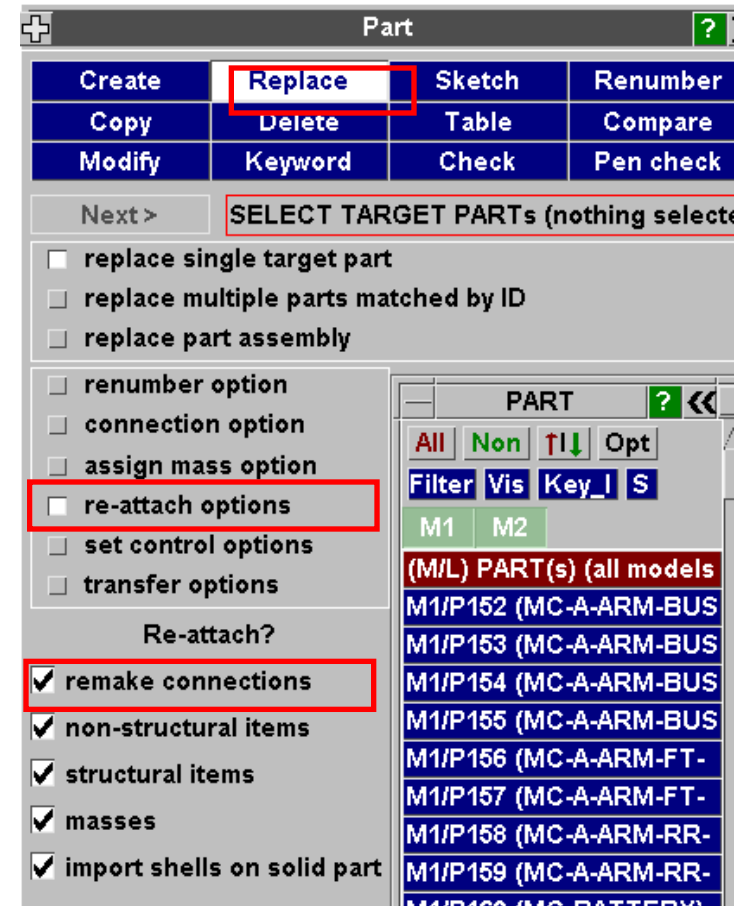
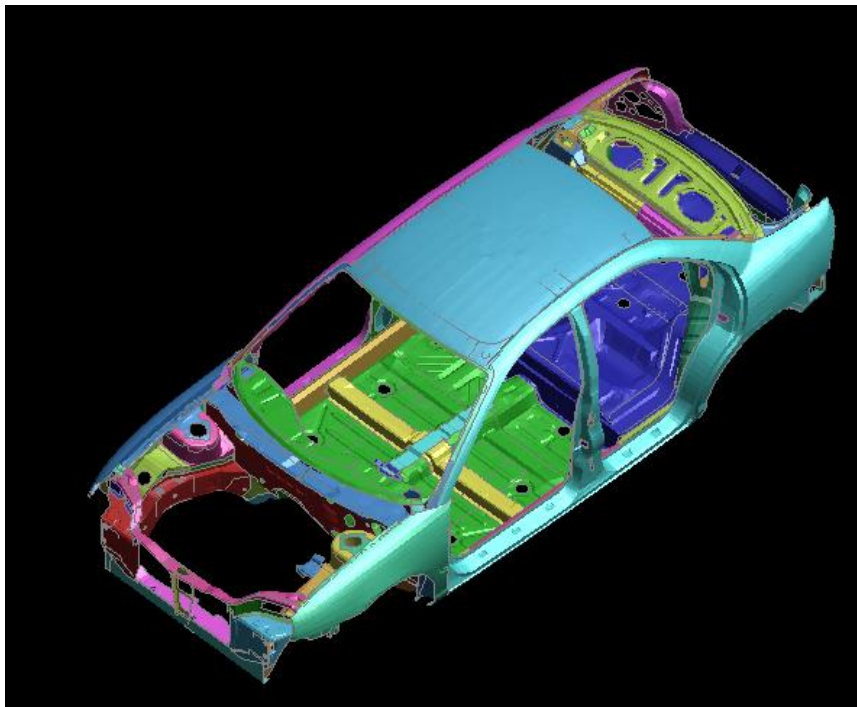
Replace Part

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 Apply.

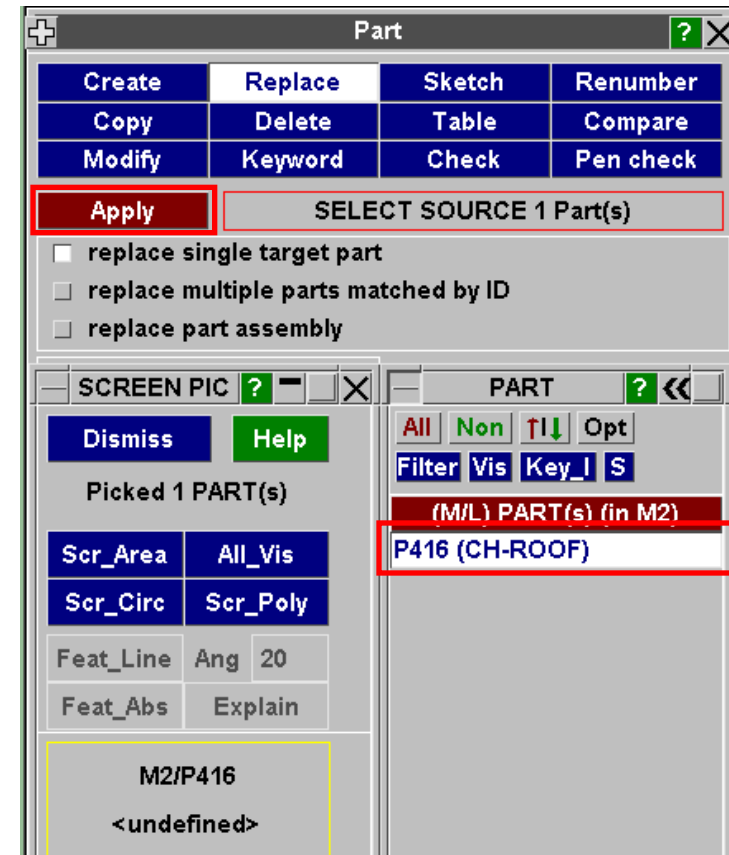
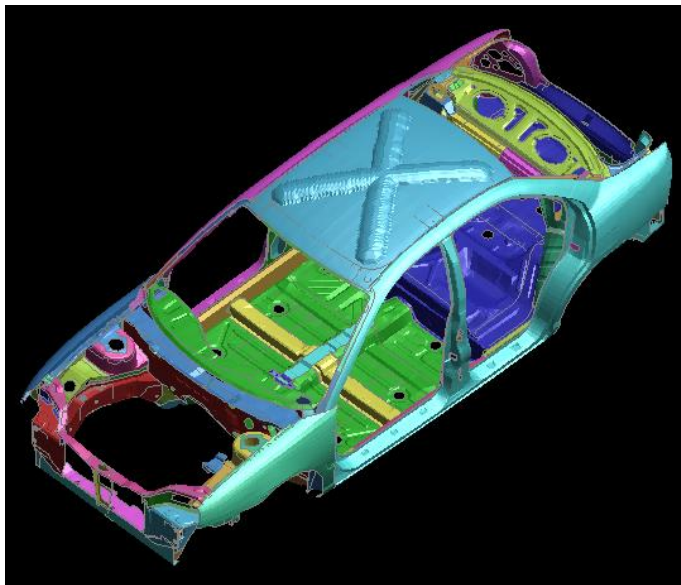
The new roof is now included in the car model.

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

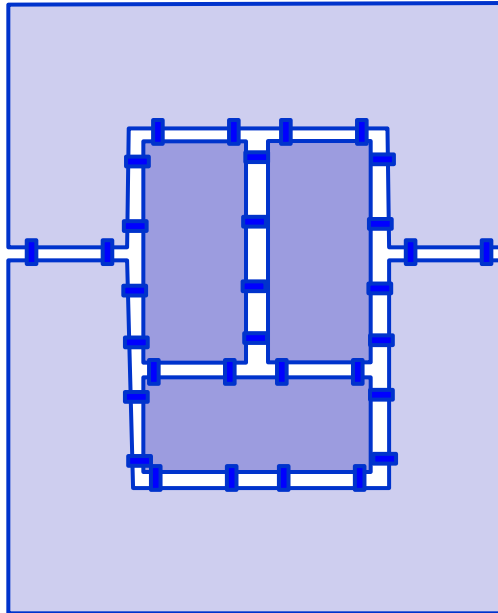
Delete Model 2 (the separate roof panel) from PRIMER.

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

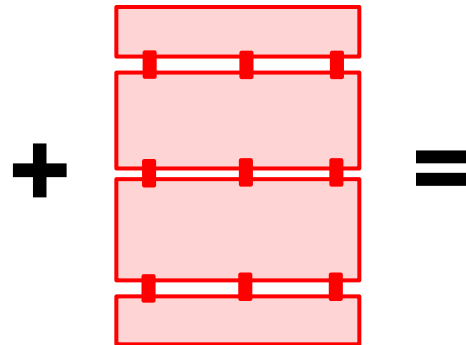
Multiple parts may be replaced in one operation, provided that the Part IDs match.



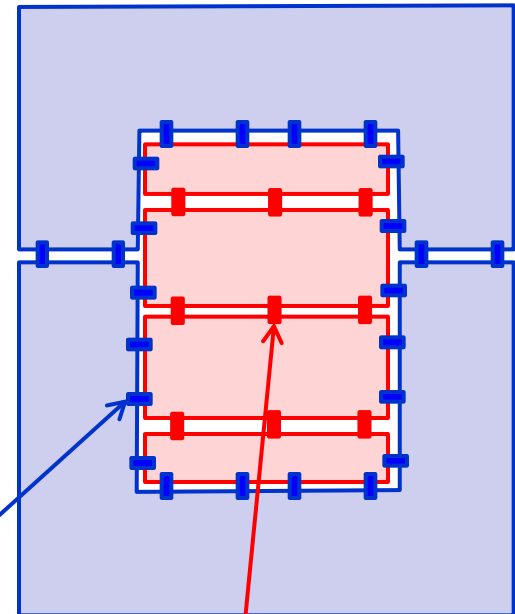
Old model with spotwelds



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



Old model with spotwelds

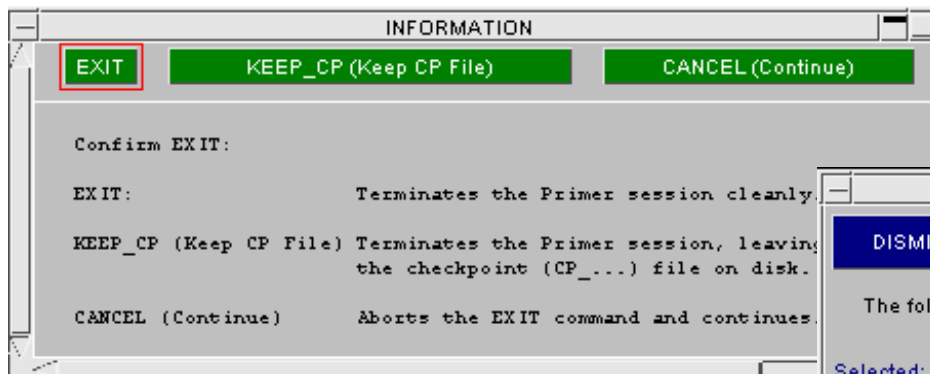


Spotwelds "around the edges" of the assembly are taken from the old model, and re-connected.

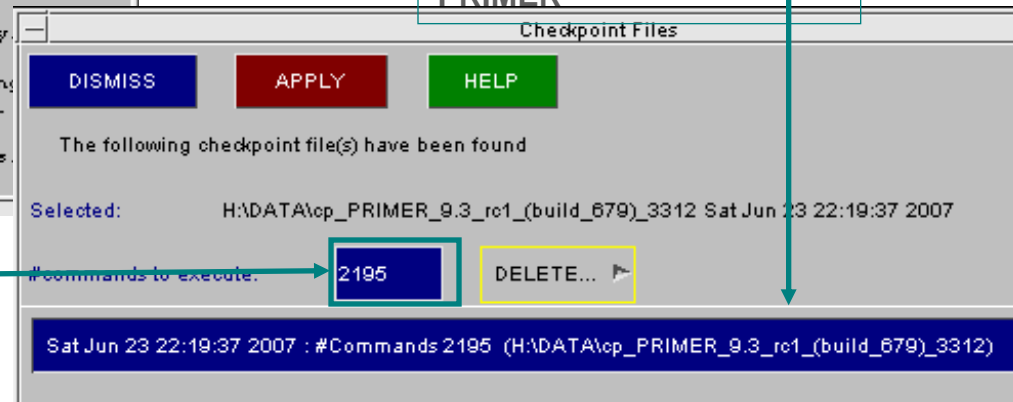
Spotwelds "inside" the assembly are taken from the new mesh

Checkpoint files, command files, and macros

- 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.



To avoid repeating the same final steps, reduce the number of commands to be repeated



To replay the cp file, click on it when starting PRIMER

- 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
```


- 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.

Tools			
Assign ms	Coat	Measure	Remove
Attached	Connection	Mechanism	Rigidify
Blanking	Cut sect	Meshing	Script
BOM	Groups	Occupant	Units
Check	Include	Orient	Xrefs
Clipboard	Macro	Other	

```

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).



Connections

Create	Find con	Read	Check
Table	Find unc	Write	From FE
Delete	List	Contact	Help
Apply	Choose spotweld mode		

HELP BOX

OK Manual Using help.. Using windows..

SPOTWELDING

=====

This panel allows you to make, manipulate, check and delete spotweld beams. Unlike 'constrained' spotwelds which are mesh dependent, spotweld beams are independent of the mesh. This makes them much easier to use as panel meshes do not have to line up exactly to be spotwelded.

The options that are available are:

CREATE: Create new spotweld beam elements.
TABLE: Modify/view spotwelds using a table.
DELETE: Delete spotweld(s).
FIND CONNECTED: Find any spotwelds that are connected to a part (or parts).
FIND UNCONNECTED: Find the parts in the model that are not connected by spotwelds.
LIST: Give summary or detailed information for spotweld(s).
READ: Read spotwelds from a file.
WRITE: Write spotwelds in the model out to a spotweld file.
CONTACT: Create or update the spotweld contact.

If your model does not contain any spotweld beams the only options that are valid are CREATE and READ.

Manual CT SI Vect plot Li Hi Sh B Lock
Stop Timestep Init Vels AC Zoom CN F All
Tidy +XY +YZ +XZ +ISO -XY -YZ -XZ -ISO Views Rev
? Typ Ent

When you need help...

File Keywords Tools Display Images Viewing Options Help Blank PART (any type)

- Presentations describing the new features of PRIMER are available from Help=>What's New.

User Manual

Support

What's New

About...

primer94.pdf

Primer 9.4 – Summary (1)

- Help menu – What's new
- LS971 R4.2.1 compatibility
- Recordable/editable macros
- Part Replace
 - Multiple parts in one operation
 - Improved re-attachment of other data
- Seatbelt fitting improvements
 - Belt path twist control
 - Easier set-up of retractors, slippers, etc
 - 2D belt elements
- Dummy positioning with contact
- Connections
 - Spotwelds: find lines, change pitch
 - Group by title
 - Adhesive "From FE" function
 - Create adhesive and MIG by node-node on free edge
 - Bolts: pick edge of hole
 - 2-ended bolts with deformable beam
 - Joint-type connections

Oasys

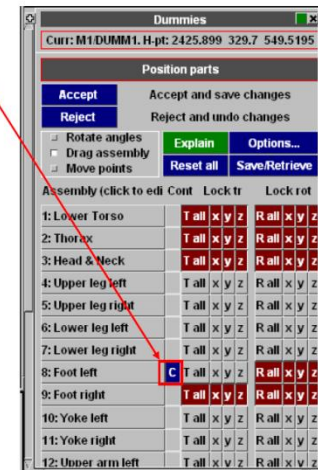
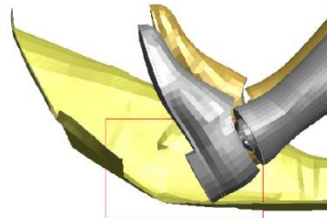
LS-DYNA ENVIRONMENT

[Back to Contents](#)

Dummy positioning with contact

During positioning, contact checking can be switched on/off.

This prevents dummy parts from penetrating the parts in the contact part set, in "drag assembly" and in "move points" modes.



Oasys

LS-DYNA ENVIRONMENT

[Back to Contents](#)

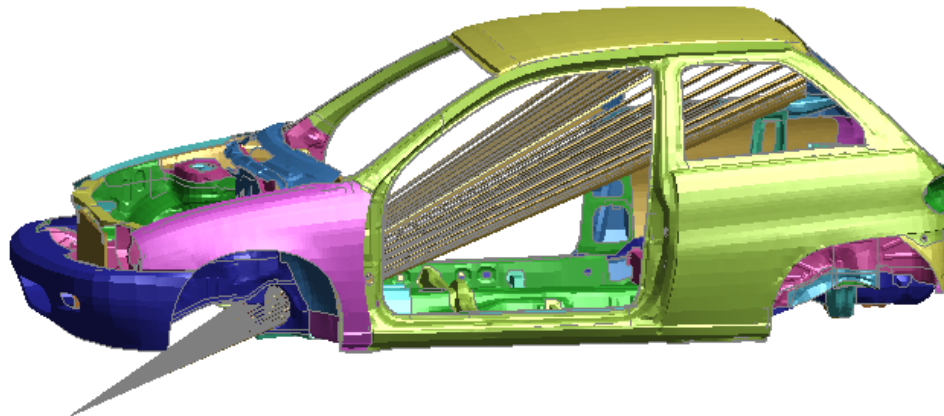
Slide 24

ARUP

Working with INCLUDE files

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).



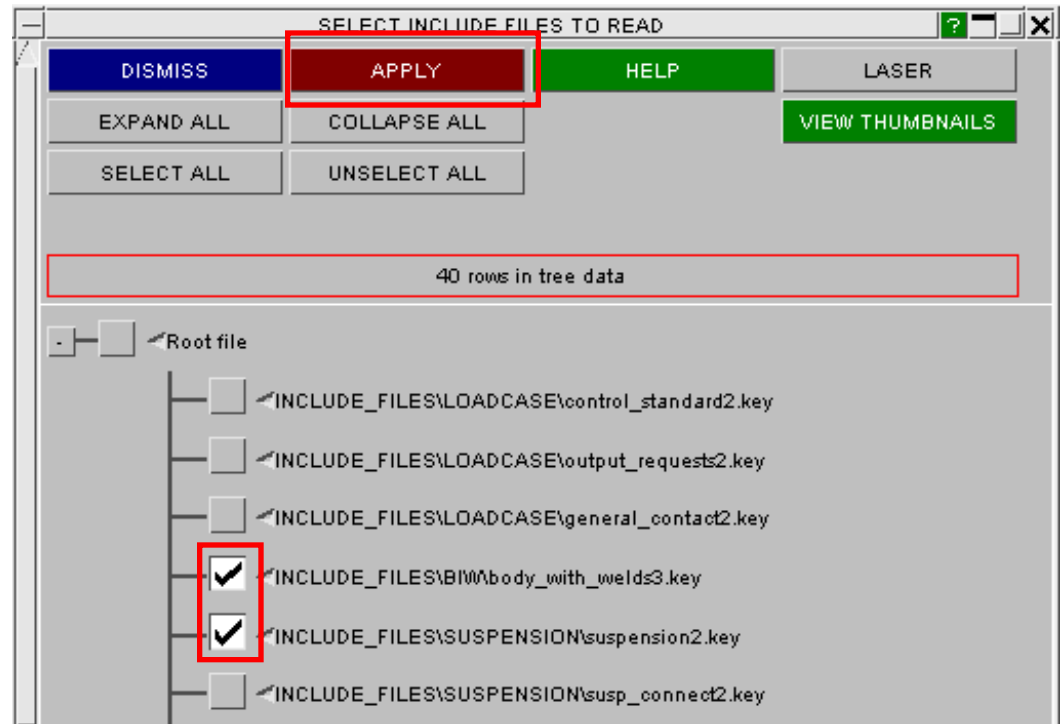
- Define a numbering scheme for each INCLUDE file. This will minimise the chance of clashes when building models, and allow large teams to work in parallel on the different INCLUDE files.
- This is a very simplified example!

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

Scan and Quick-scan

Include files may be selected on reading a model.

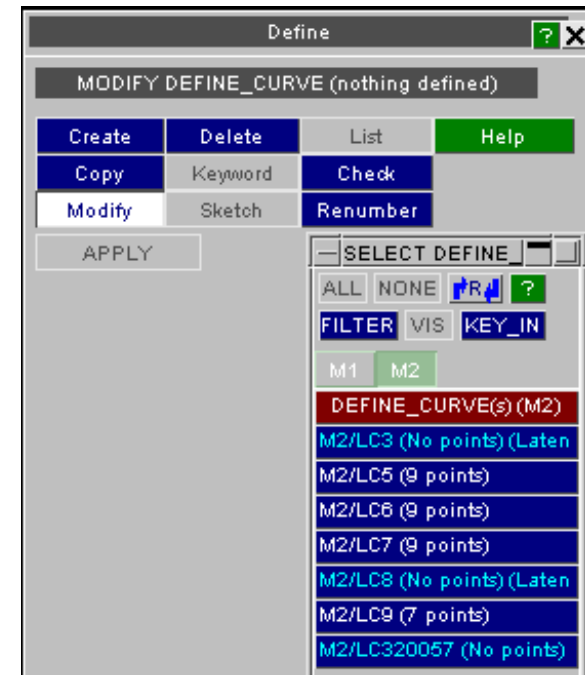
1. In the Model menu, click **Read**. Browse for demo_car4.key. Press **Quick scan**.
2. Click on the body and suspension INCLUDE files. Press **Apply**.
3. PRIMER reads in only the selected INCLUDE files – this will be Model 2.



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.

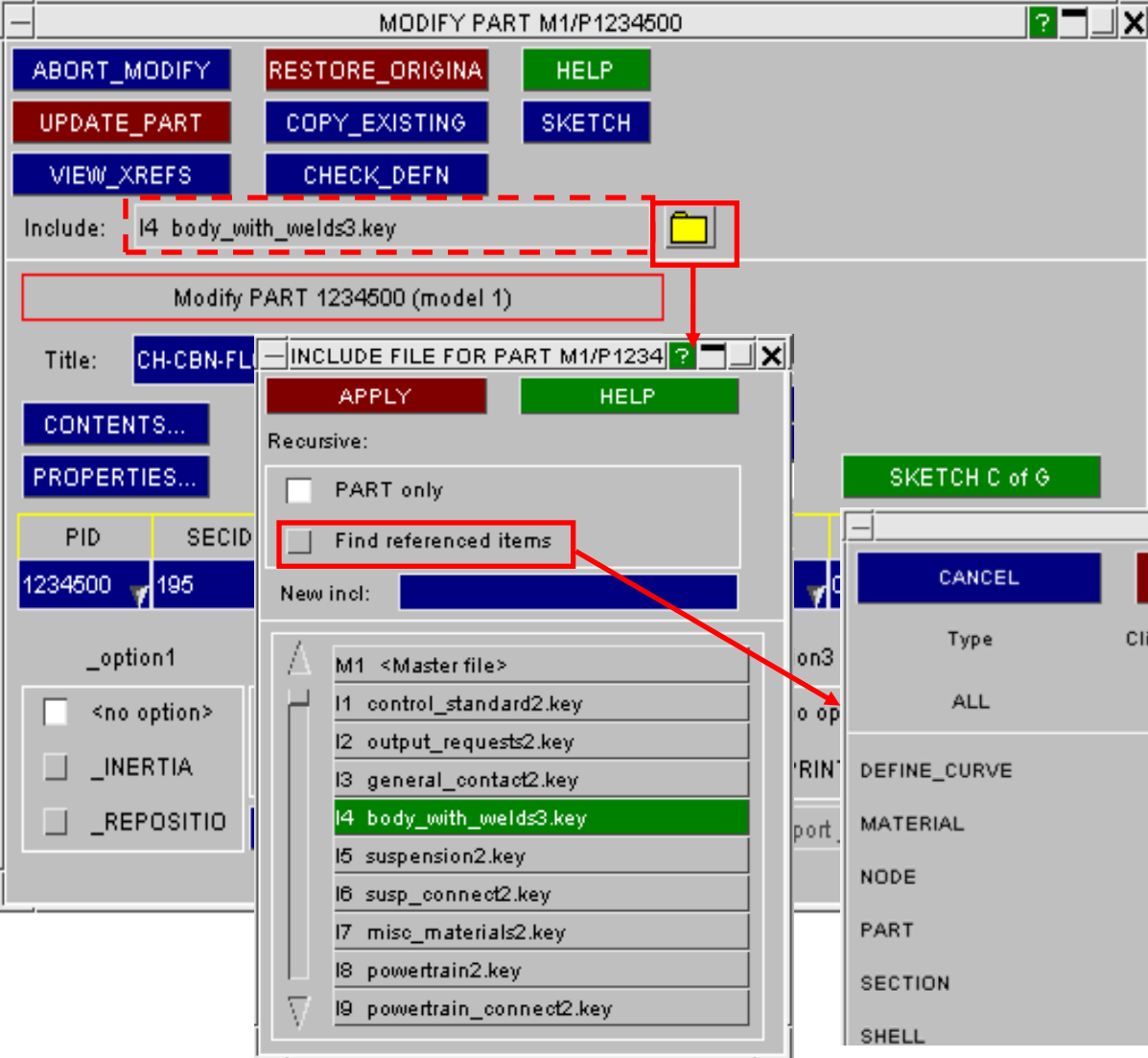
Now delete all models in PRIMER except Model M1.



- 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



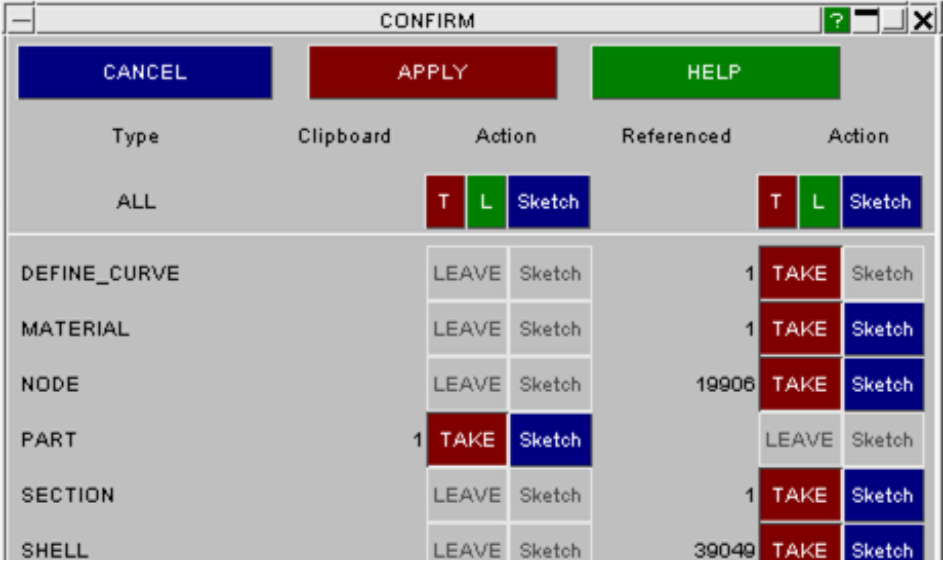
INCL file location



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.

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.

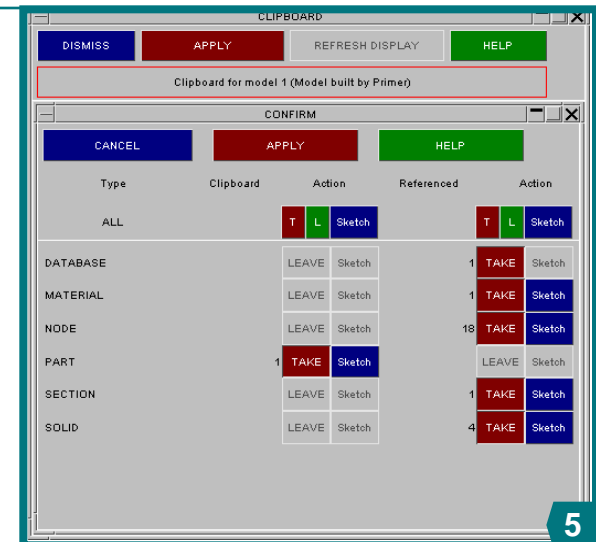
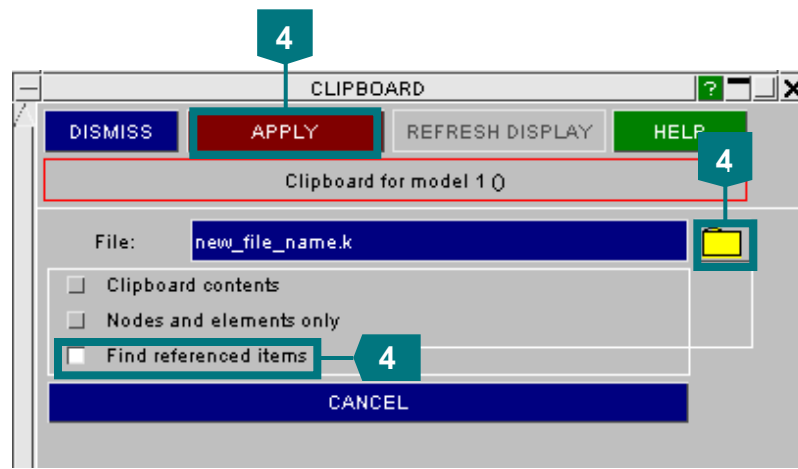
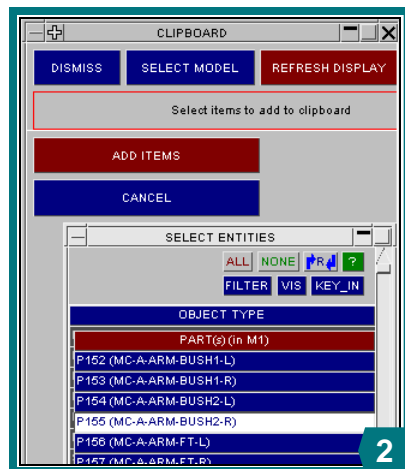
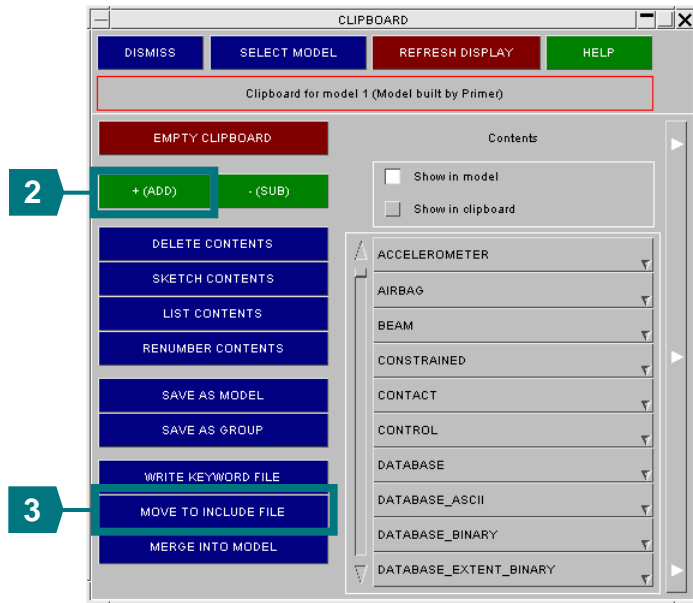


Moving entities to a different Include file

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.



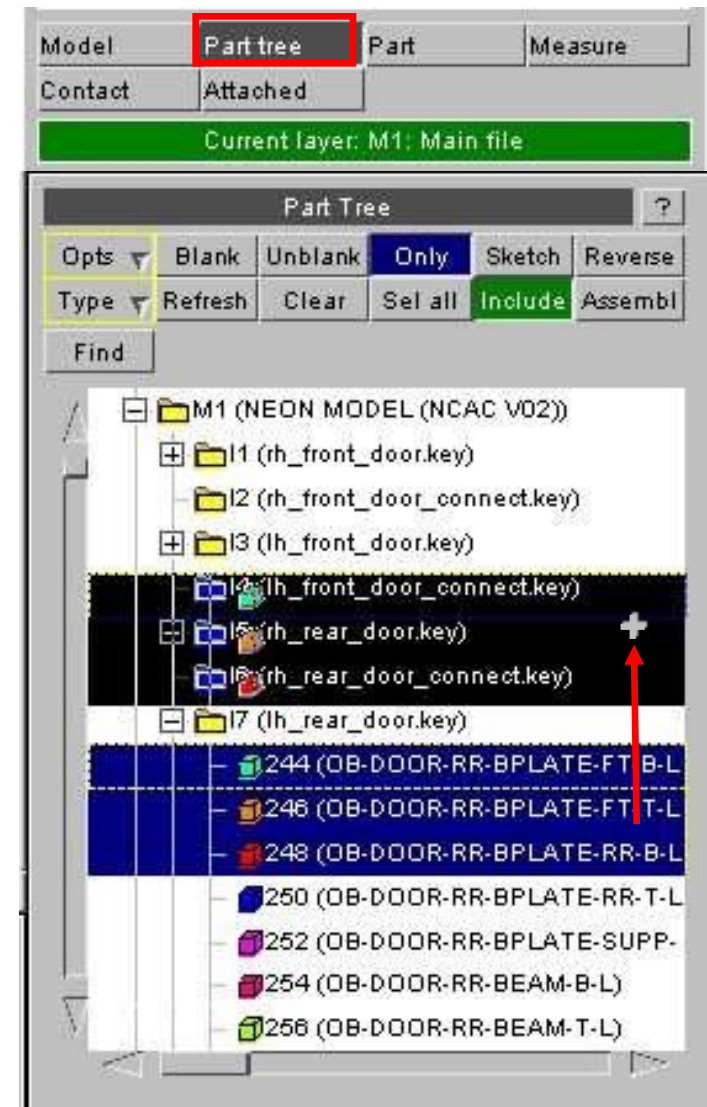
Splitting a model into INCLUDE files

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.

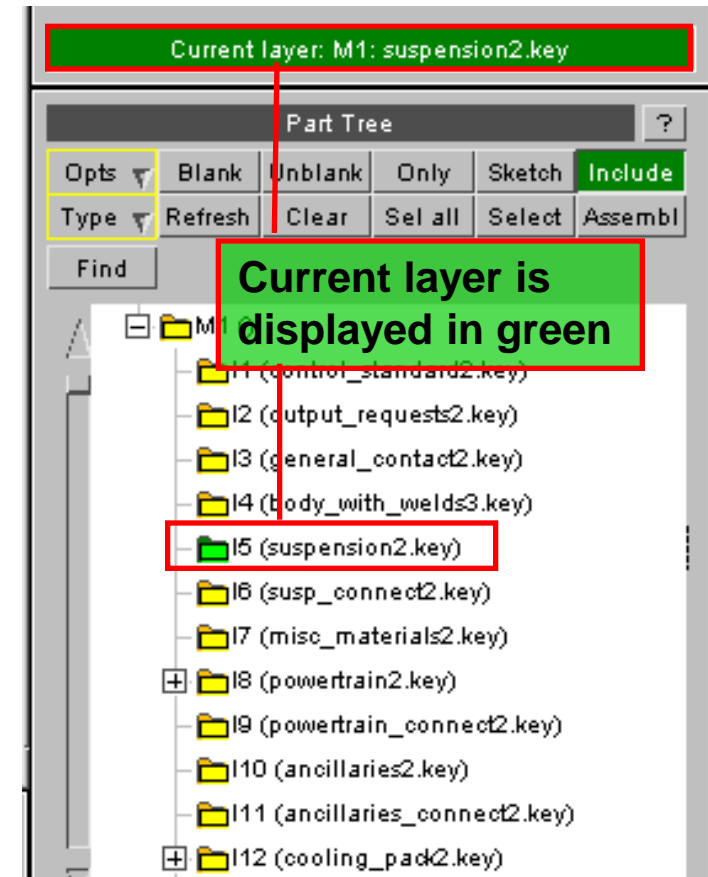
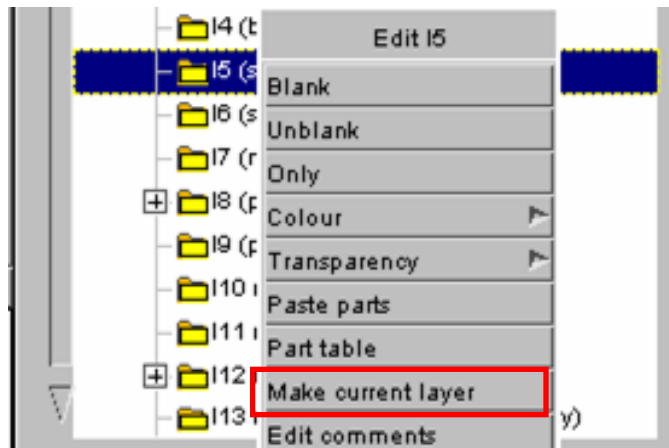
Alternatively, select as before => right-click => Cut => select INCLUDE file, => right-click => Paste.

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

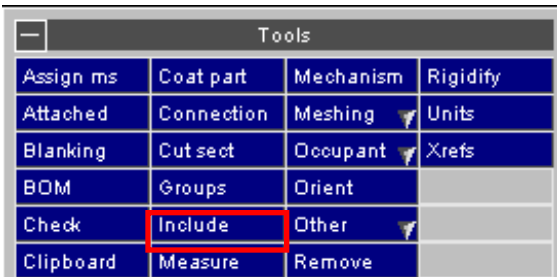


INCLUDE file for newly created entities

- 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.



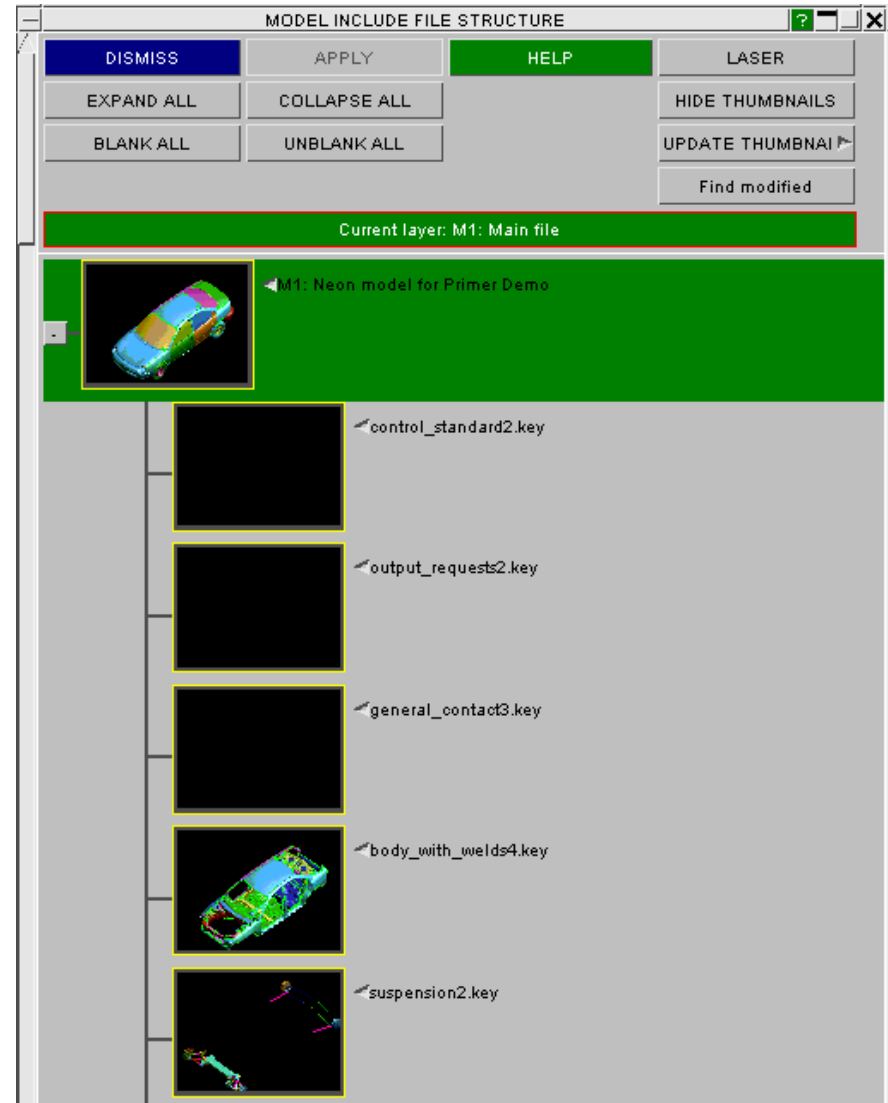
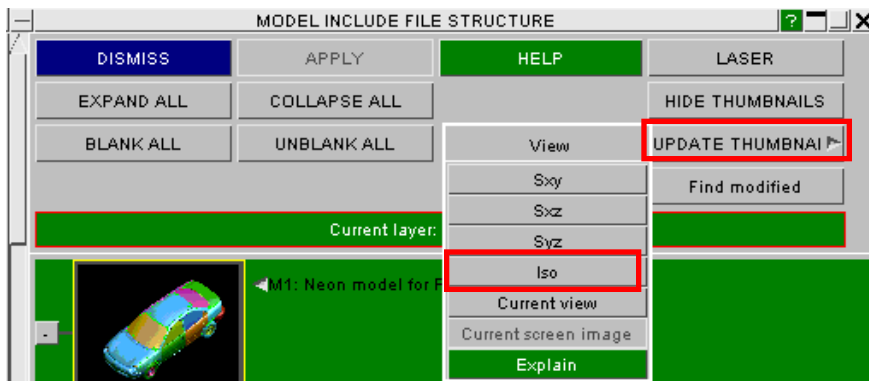
The INCLUDE menu



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 Update Thumbnails, choose ISO view. PRIMER creates a thumbnail for each INCLUDE file.

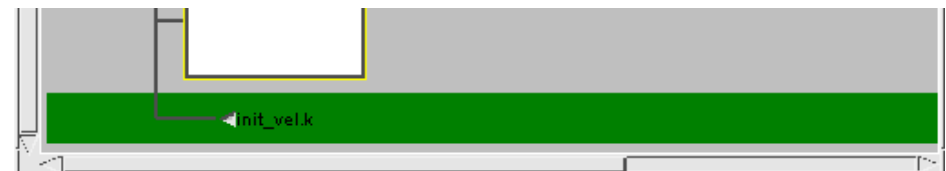
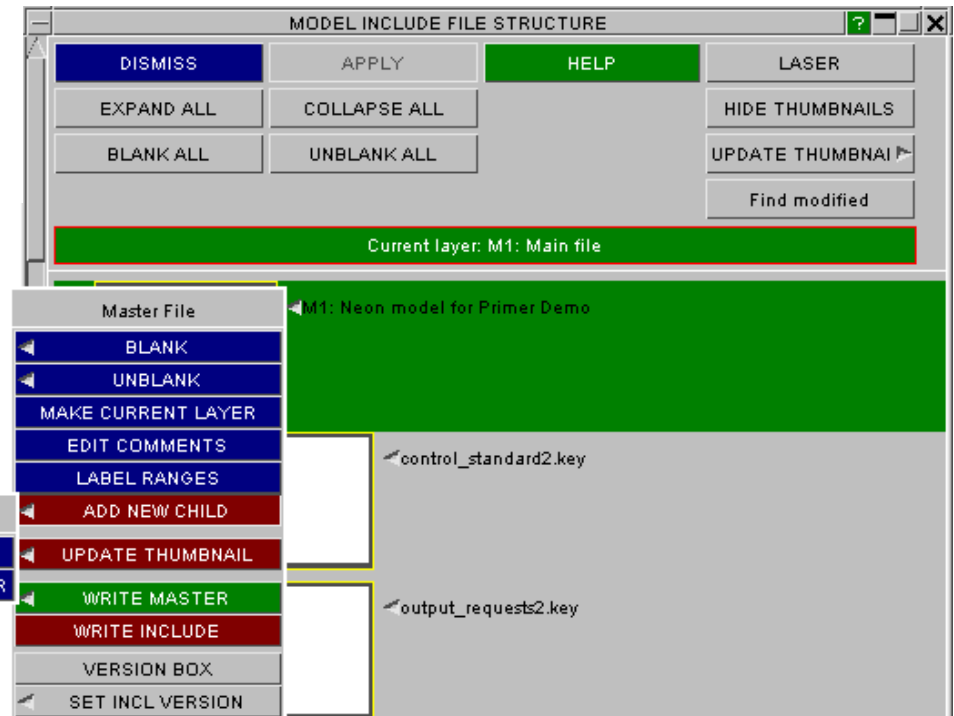
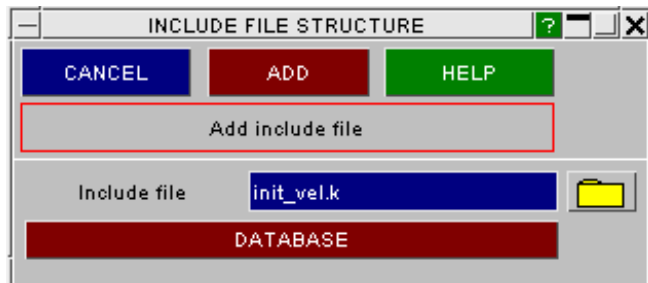


Adding a new INCLUDE file

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).

Add child
*INCLUDE
*INCLUDE_TRANSFOR



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.

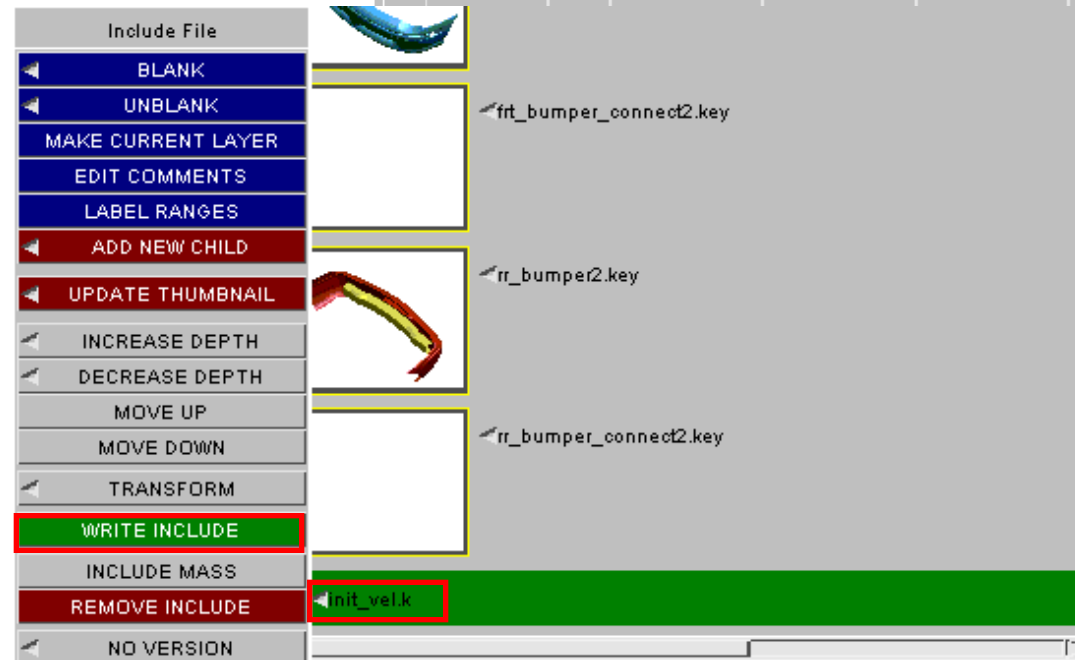
Using the Include menu, right-click off init_vel.k, WRITE INCLUDE. Use a text editor to check the contents of the file written out.

Keyword: M1/INITIAL_VELOCITY

CANCEL RESET_ALL HELP
UPDATE CHECK_ALL SKETCH_ALL

Keyword M1 INITIAL_VELOCITY (0/0 mod)

#	Options...	Incl	NSID S_NO	NSIDEX S_NO	BOXID BOX	IRIGID S_PT	
			VX F	VY F	VZ F	VXR F	VYR F
			VXE F	VYE F	VZE F	VXRE F	VYRE F
			0	0	0	0	
			13500.0	0.0	0.0	0.0	0.0



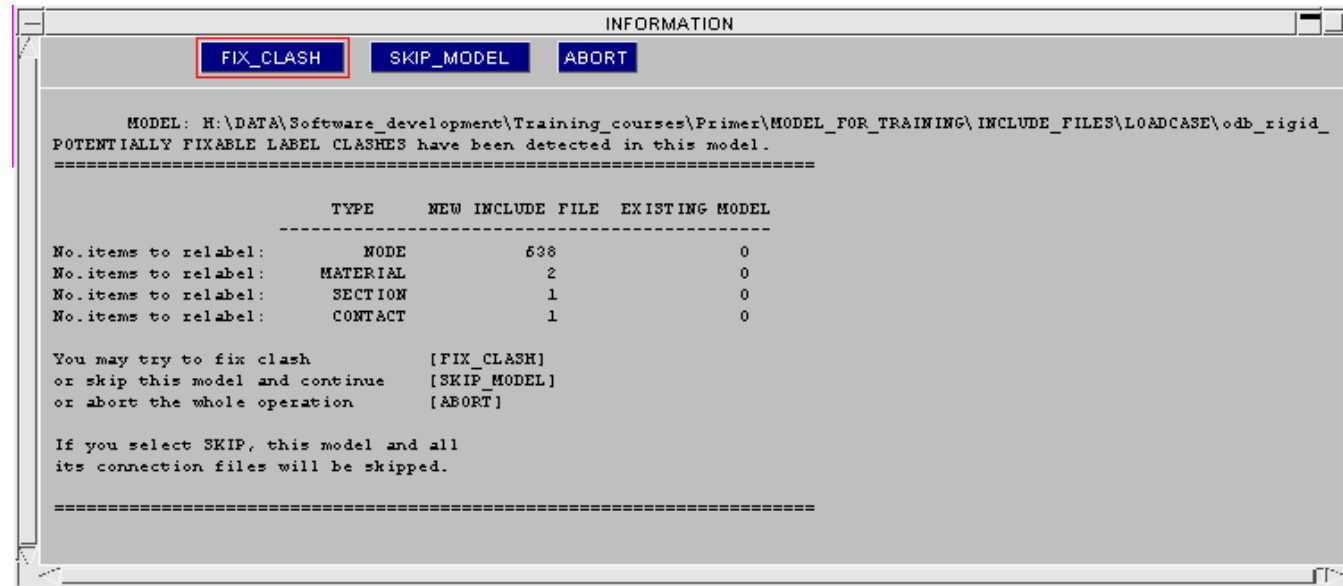
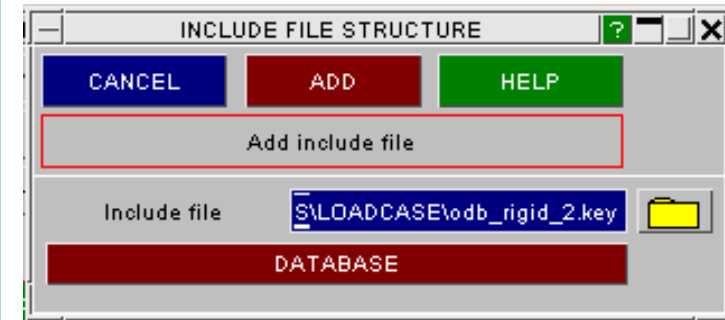
Adding an existing INCLUDE

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.

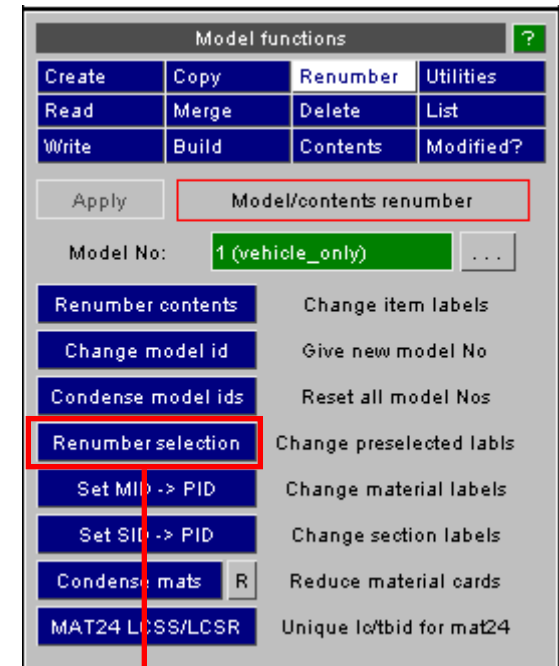
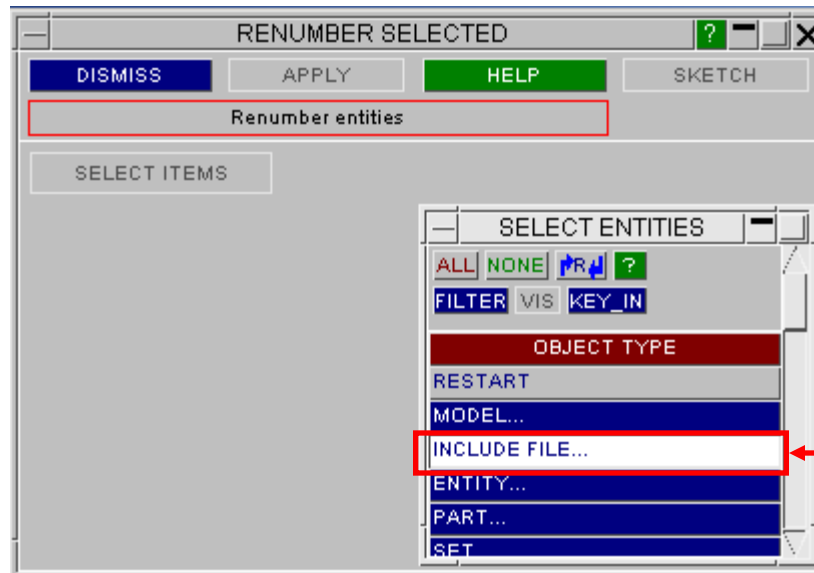
When adding INCLUDE files by this method, PRIMER is able to resolve label clashes. Press FIX_CLASH – we can adjust the numbering later.

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 HELP SKETCH

Renumber entities

Type	Renumber		low:high	start at	offset	Information	Action
ALL TYPES	YES	NO	id low:high (num sel)	1000000		LIST STATUS	GLOBAL ACTION
SHELL	YES	NO	1000000:1013207 (13208/	1000000		CLASH	Do not renumber
DISCRETE	YES	NO	1000000:1000007 (8/8)	1000000		NO CLASH	No action needed
MASS	YES	NO	1000000:1000035 (36/40)	1000000		NO CLASH	No action needed
SET_NODE	YES	NO	1000000:1000021 (22/121)	1000000		NO CLASH	No action needed
PART	YES	NO	1000000:1000044 (45/374)	1000000		NO CLASH	No action needed
MATERIAL	YES	NO	1000000:1000044 (45/321)	1000000		NO CLASH	No action needed
SECTION	YES	NO	1000000:1000044 (45/324)	1000000		NO CLASH	No action needed
JOINT	YES	NO	1000000:1000009 (10/10)	1000000		NO CLASH	No action needed
NODAL_RIGID_B	YES	NO	1000000:1000011 (12/103)	1000000		NO CLASH	No action needed
SPOTWELD	YES	NO	1000000:1000154 (155/12	1000000		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

Renumber include file ranges for model 1: demo_car4.key

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

Options Retain original order

File name	Range unique	In range?	General		no/el/nset/nrb/cwld/		Auto all
			Start	End	Start	End	
demo_car4.key	INACTIVE	INACTIVE	0	0	0	0	Auto
control_standard2.	INACTIVE	INACTIVE	0	0	0	0	Auto
output_requests2.	INACTIVE	INACTIVE	0	0	0	0	Auto
general_contact3.k	INACTIVE	INACTIVE	0	0	0	0	Auto
body_with_welds4.	INACTIVE	INACTIVE	0	0	0	0	Auto
suspension2.key	INACTIVE	INACTIVE	0	0	0	0	Auto
susp_connect2.ke	INACTIVE	INACTIVE	0	0	0	0	Auto
misc_materials3.ke	INACTIVE	INACTIVE	0	0	0	0	Auto
powertrain2.key	INACTIVE	INACTIVE	0	0	0	0	Auto
powertrain_conne	INACTIVE	INACTIVE	0	0	0	0	Auto
ancillaries2.key	INACTIVE	INACTIVE	0	0	0	0	Auto

Model functions

Create	Copy	Renumber	Utilities
Read	Merge	Delete	List
Write	Build	Contents	Modified?

Apply Model/contents renumber

Model No: 1 (Neon model for Primer D)

Renumber contents	Change item labels
Change model id	Give new model No
Condense model ids	Reset all model Nos
Renumber selection	Change preselected labls
Set MID -> PID	Change material labels
MAT24 LCSS/LCSR	Unique lc/tbid for mat24
Condense mats	Reduce material cards
Set SID -> PID	Change section labels
Renumber includes	Renumber include ranges
Declash labels	Declash Elements/Sets/Mat

Renumber include file ranges for model 1: demo.key

Apply Read csv Write csv Copy no/el/inset/nrb/cwld/hswa ranges to ge

Options Retain original order

File name	Range unique	In range?	General		no/el/inset/nrb/cwld/		Auto all
			Start	End	Start	End	
demo.key	INACTIVE	INACTIVE	0	0	0	0	Auto
control_04_comme	INACTIVE	INACTIVE	0	0	0	0	Auto
taurus_rrdoor_07.k	OK	YES	20000	20099	2000000	2000999	Auto
taurus_biw16.key	OK	NO	21000	21499	2581000	3080999	Auto
taurus_frdoors08.k	OK	YES	20100	20199	2001000	2100999	Auto
taurus_fascia07.ke	OK	YES	20200	20299	2101000	2200999	Auto
taurus_hood06.key	OK	YES	20300	20399	2201000	2250999	Auto
taurus_ancils09.ke	OK	YES	20400	20499	2251000	2350999	Auto
taurus_pwr11.key	OK	YES	20500	20699	2351000	2550999	Auto
seatD_09.key	OK	YES	12000	12999	1200000	1399999	Auto
belt_07.key	OK	YES	11000	11499	1100000	1149999	Auto

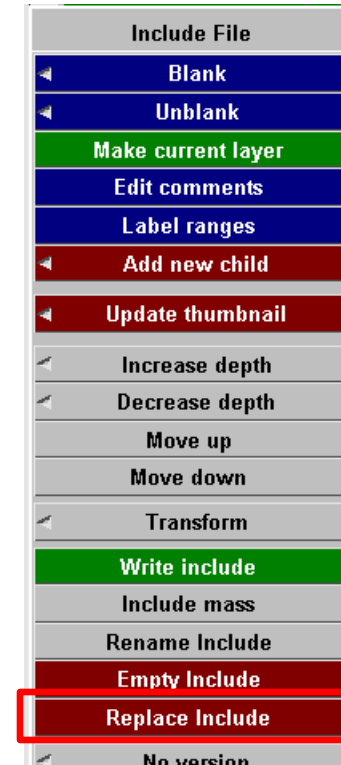
- Two ranges can be set for each include
 - One range for nodes/elements + some other types
 - One range for all other labelled entities.
 - Both ranges can be the same.
- Label ranges can typed in, or imported from a CSV file.
- Label ranges are saved as comments to each include file.
- The panel will 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.

Replacing an INCLUDE file

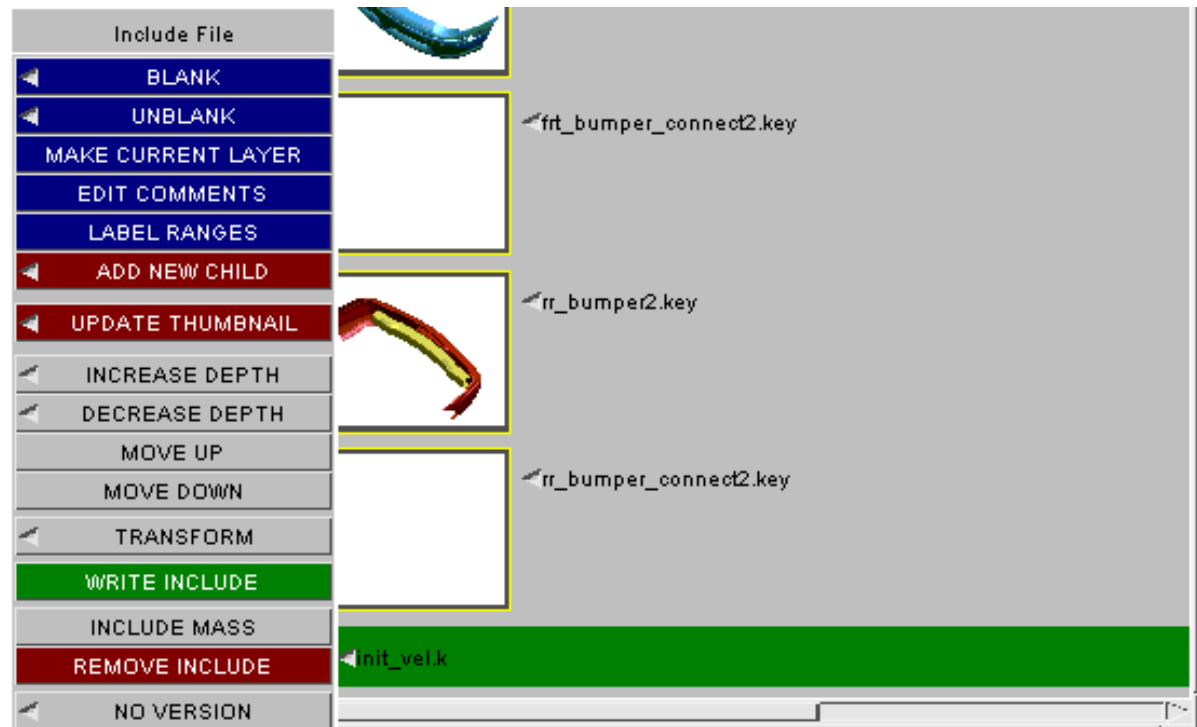
As an example, we will now replace this barrier model with a different one that has already been renumbered and positioned.

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

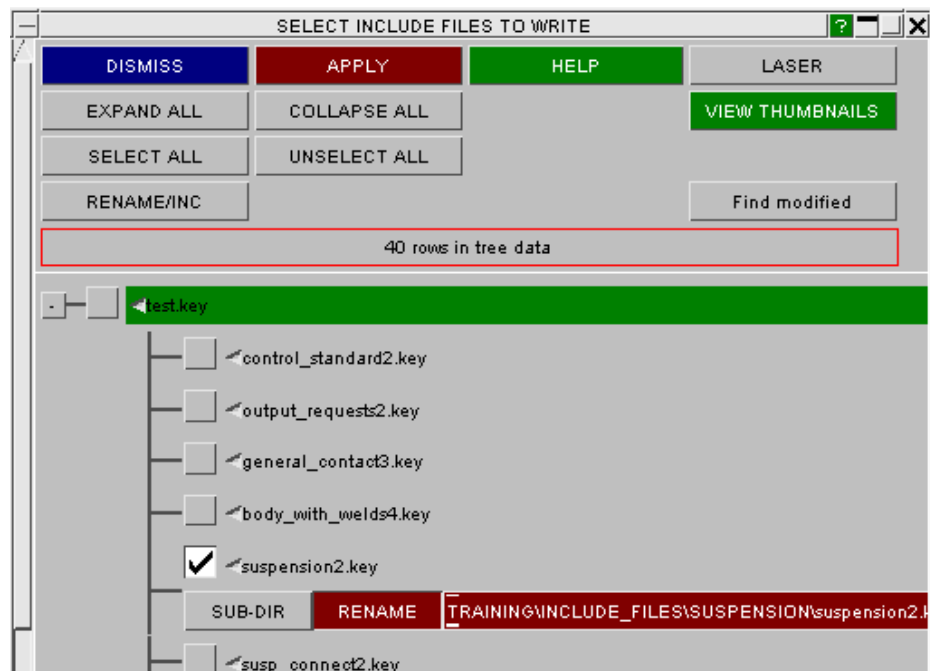
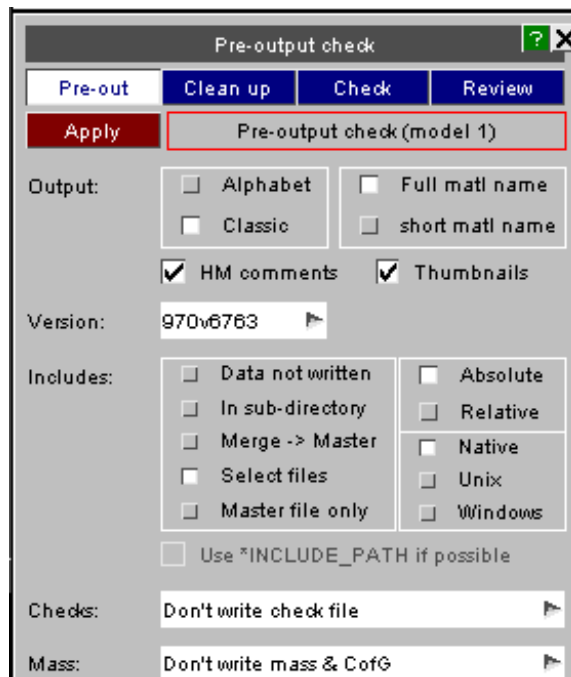
Browse for INCLUDE_FILES/LOADCASE/odb_rigid_moved.key



- 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

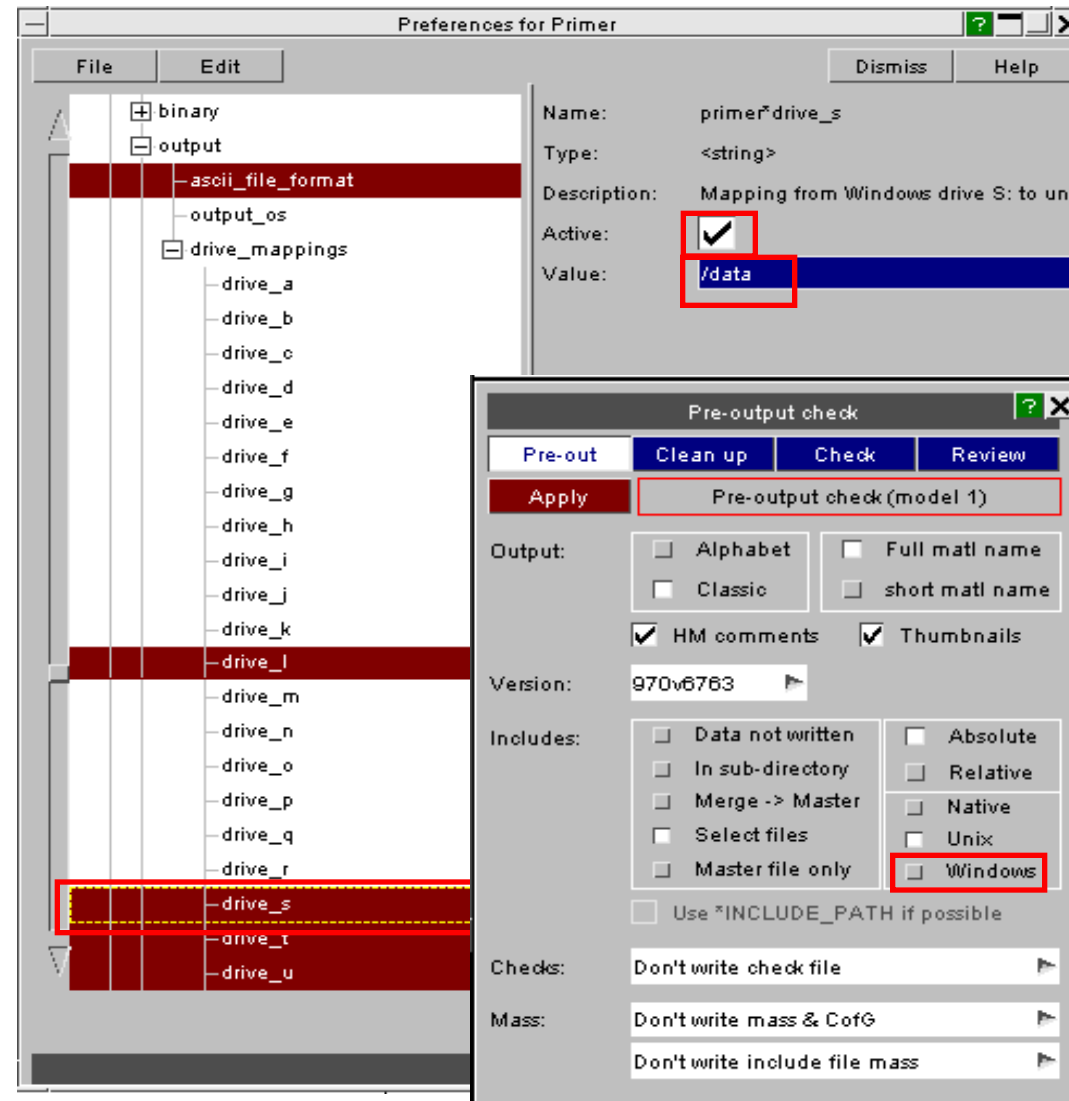


- We have already seen how to write an INCLUDE file from the Include menu.
- Now try Model=>Write, opt for Select Files.
- Check the box for suspension2.key (you can check as many INCLUDE files as you wish).
- Option SUB-DIR means that PRIMER will create a new subdirectory, INCL, INCL_1, INCL_2 etc in 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 filoe 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



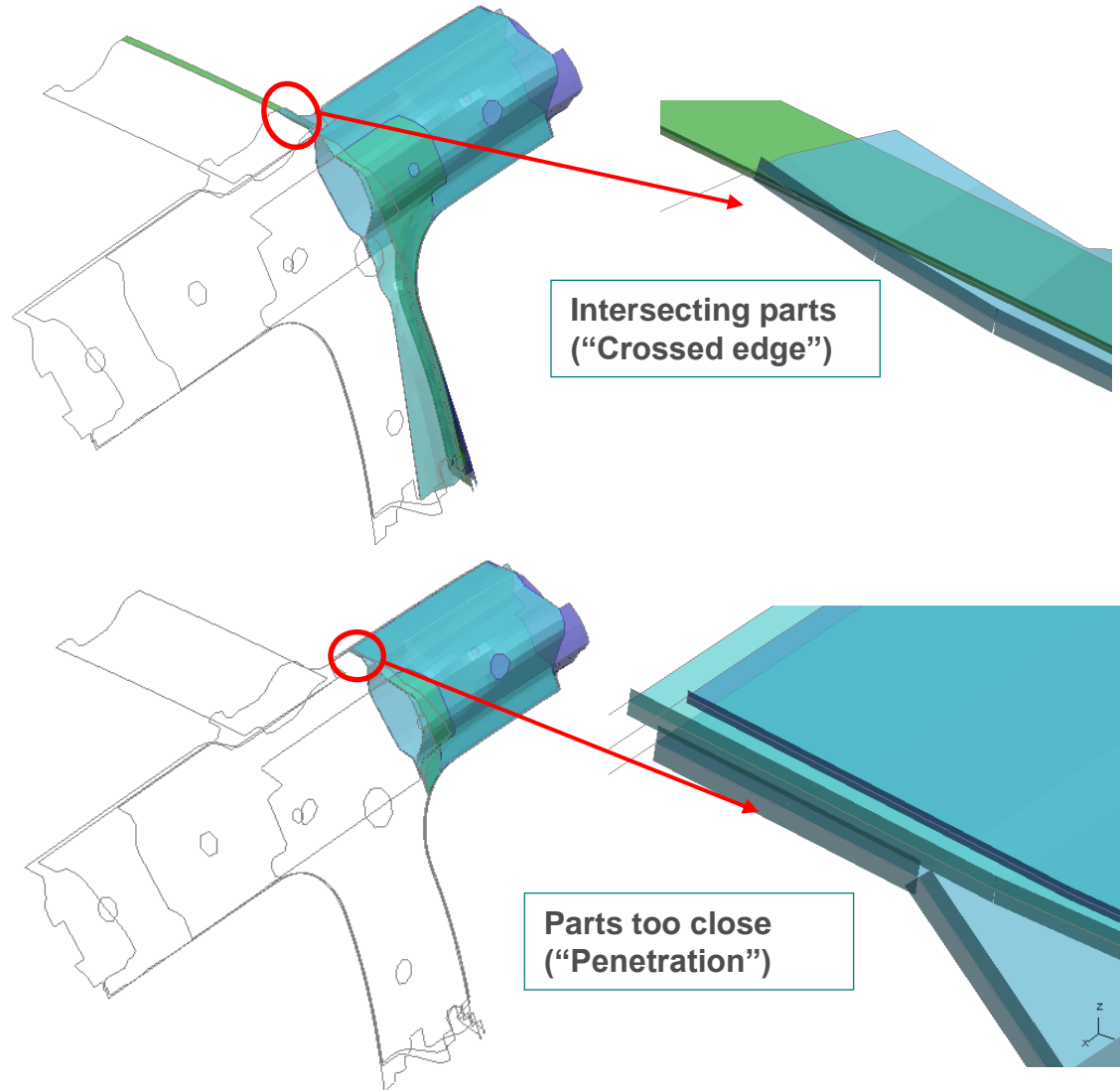
Contact Penetration Checking and Fixing

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

Abort modify Restore original Help

Update contact Copy existing Sketch

View xrefs Check defn Pen check

Include: M1 <Master file>

Modify contact M1/CONT2

Label: 2 Give label/title

Title: Optional data..

Type: ATIC_SINGLE_SURFACE <type> data..

_OFFSET THERMA THERMAL d..

_SMOOTH _MPP _MPP data..

SLAVE SIDE

Set type (sstyp) 2

Part set (ssid) 2

Box id (sboxid) 0

sboxid - Contact Vo NO

Penalty factor (sf) 0.0

Thickness value (0.0

Thickness fact (s 0.0

Force output flag (s YES

Friction attributes

Static friction (fs) 0.0

Dynamic friction (0.0

Fric Decay coeff 0.0

Viscous friction (0.0

Viscous fric fac (0.0

Coulomb fric fac (0.0

MASTER SIDE

Set type (mstyp) 0

Segm set (msid) none>

Box id (mboxid) 0

mboxid - Contact V NO

Penalty factor (sf 0.0

Thickness value (0.0

Thickness fact (s 0.0

Force output flag (YES

General attributes

Viscous damping 0.0

Birth time (bt): 0.0

Death time (dt): 0.0

Small pen chk (p 0

Create Delete List Pen check

Copy Keyword Check Help

Modify Sketch Renumber

Apply MODIFY contact M1/CONT2

ALL NON PR ?

FILTE VIS KEY I

CONTACT(s) (all models)

M1/CONT1 (TIED_SURFACE_TO_SURFACE)

M1/CONT2 (AUTOMATIC_SINGLE_SURFACE)

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

Dismiss Check all Options...
List Errors Check visible

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE
<No title defined>

select parts sel none sel all sel xedge ?

P8231:P8710 (5 x-edges 6 pens)
P8710:P82151 (0 x-edges 108 pens)
P8712:P82151 (0 x-edges 76 pens)

penetrations magnitude \geq 0.0

sketch unblank recursive

contour penetrating

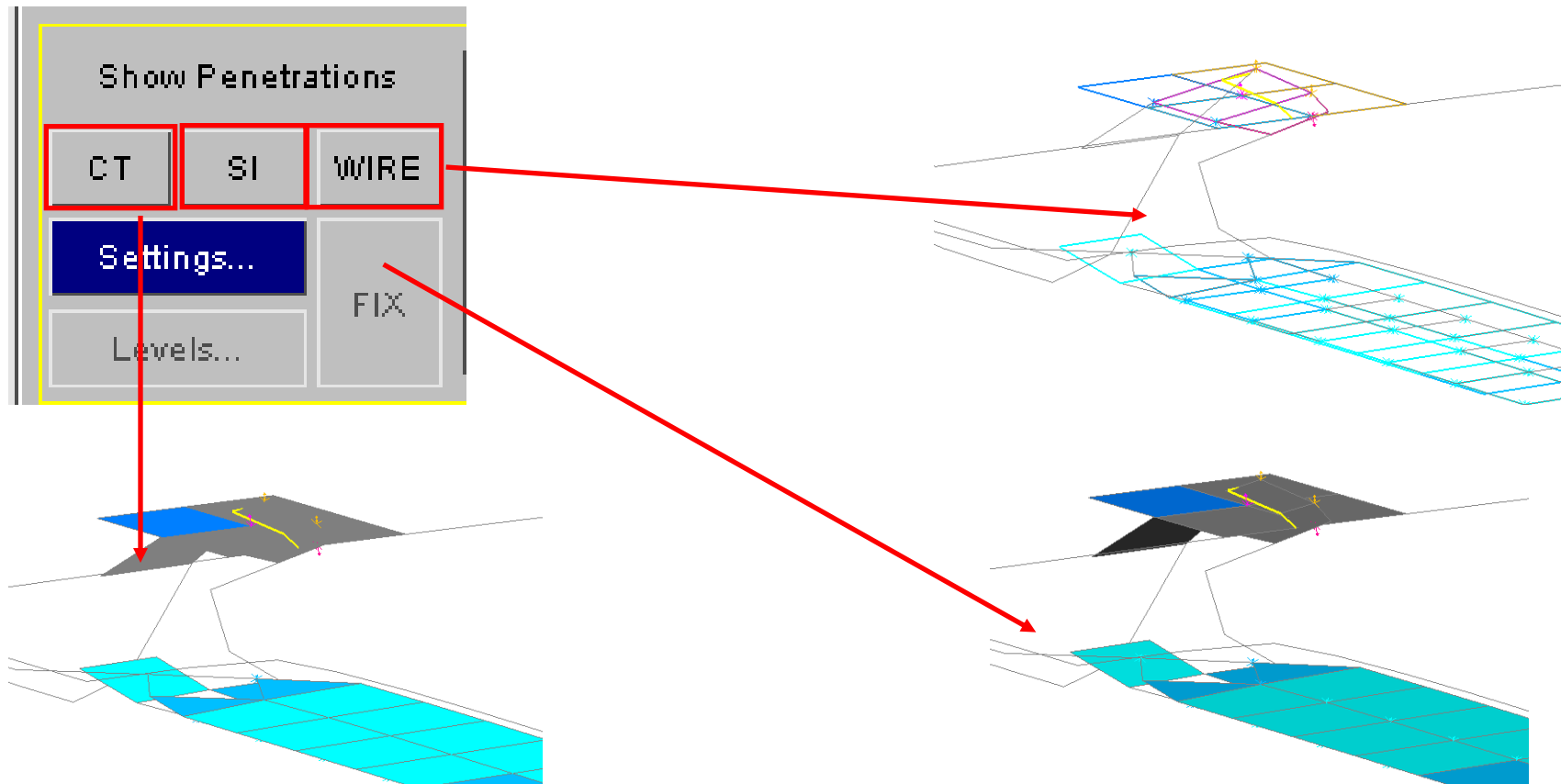
CT SI WIRE

Settings... FIX

Levels...

Pen: 190 vis 190 total
X'ed: 5 vis 5 total
Min: 3.2391e-002
Max: 1.1148e+000
Sum: 4.9069e+001

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



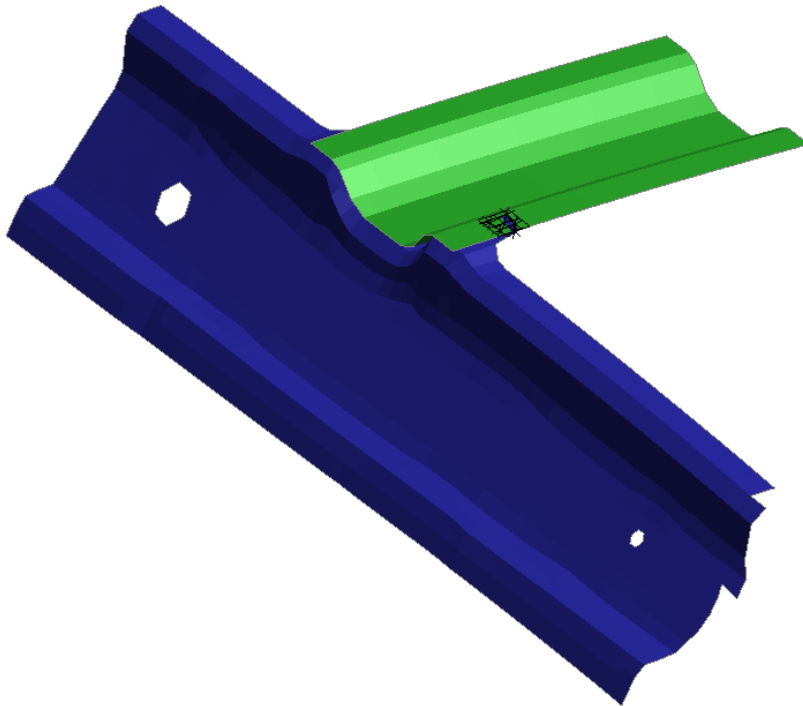
Try the different plotting modes.

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.

Penetration checking

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”.



PEN CHECK M1/CONT2

Dismiss Check all Options...

List Errors Check visible

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE

<No title defined>

select parts sel none sel all sel xedge ?

P8231:P8710 (5 x-edges 6 pens)

P8710:P82151 (0 x-edges 108 pens)

P8712:P82151 (0 x-edges 76 pens)

penetrations magnitude \geq 0.0

sketch unblank recursive

contour penetrating

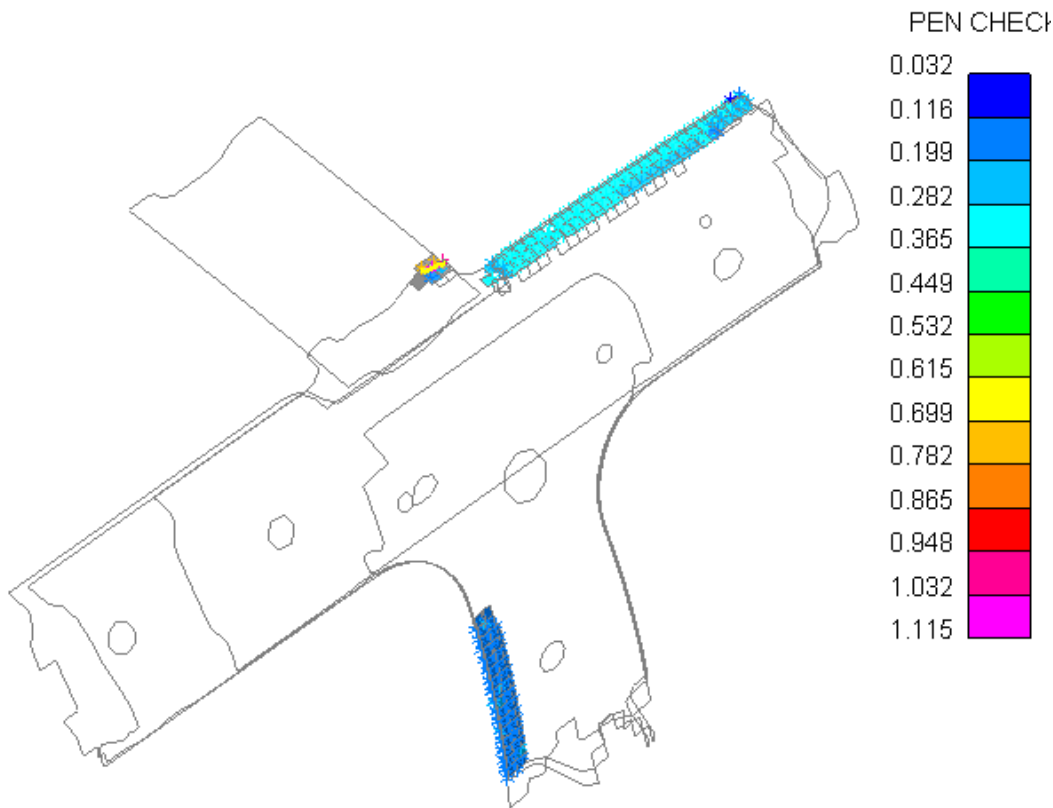
CT SI WIRE

Settings... Levels... FIX

Pen: 6 vis 190 total
X'ed: 5 vis 5 total
Min: 3.2391e-002
Max: 1.1148e+000
Sum: 4.9069e+001

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

Contact penetrations - Settings



Call up the Settings panel

PEN CHECK M1/CONT2

Dismiss Check all Options...
List Errors Check visible

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE
<No title defined>

select parts sel none sel all sel xedge ?

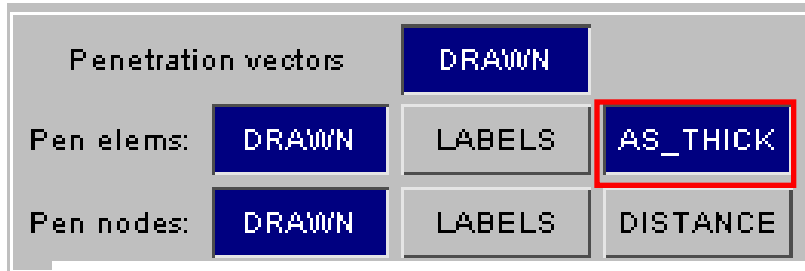
P8231:P8710 (5 x-edges 6 pens)
P8710:P82151 (0 x-edges 108 pens)
P8712:P82151 (0 x-edges 76 pens)

penetrations magnitude >= 0.0

sketch unblank recursive

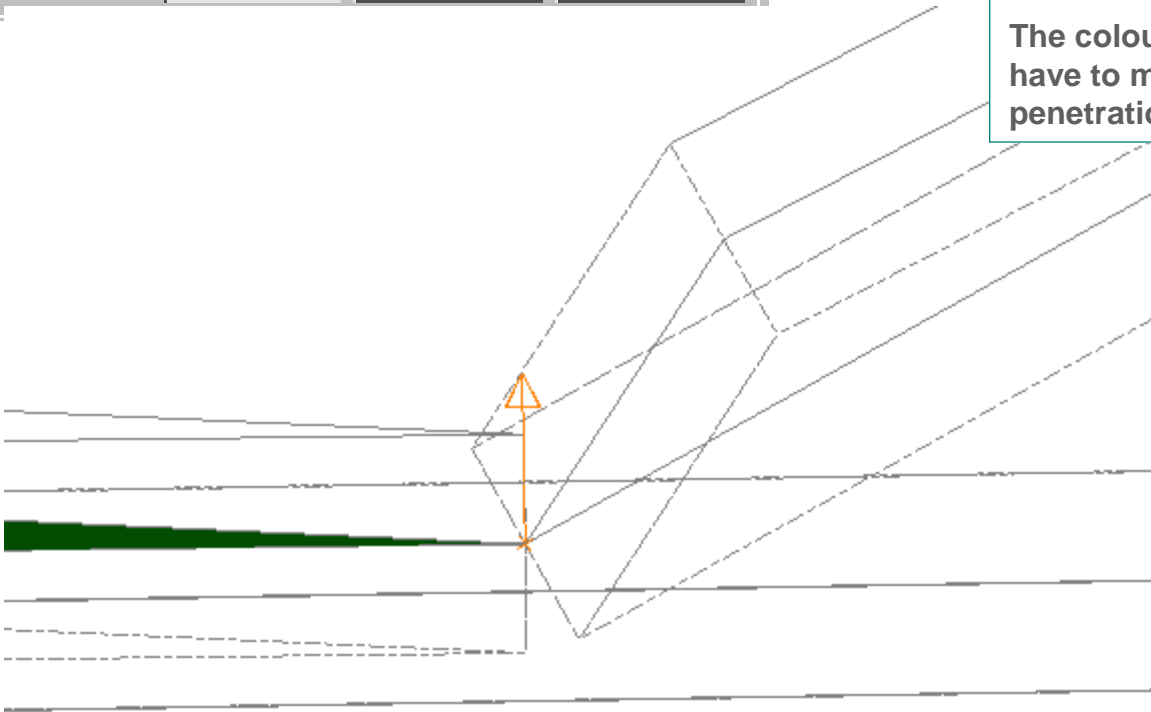
contour penetrating Pen: 190 vis 190 total
CT SI WIRE X'ed: 5 vis 5 total
Settings... Min: 3.2391e-002
Levels... Max: 1.1148e+000
FIX Sum: 4.9069e+001

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



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.

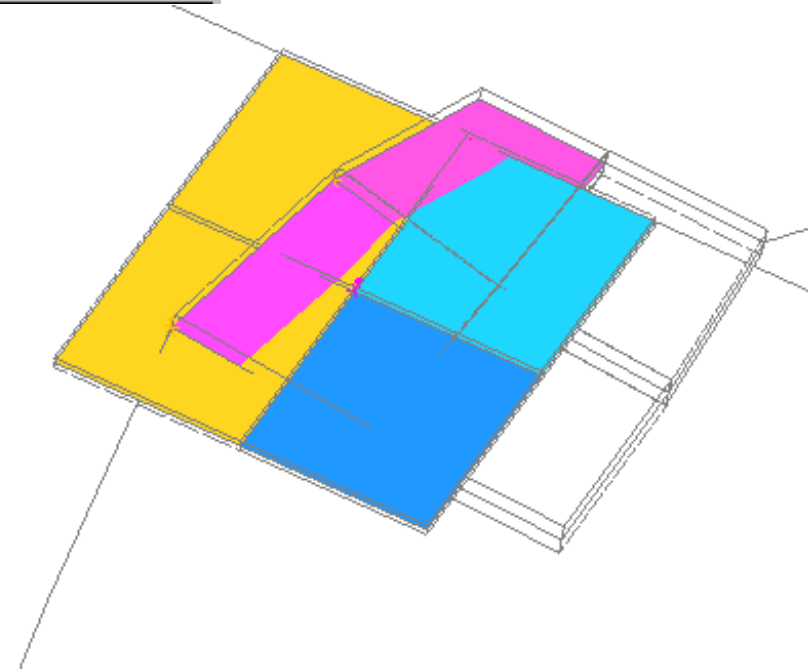
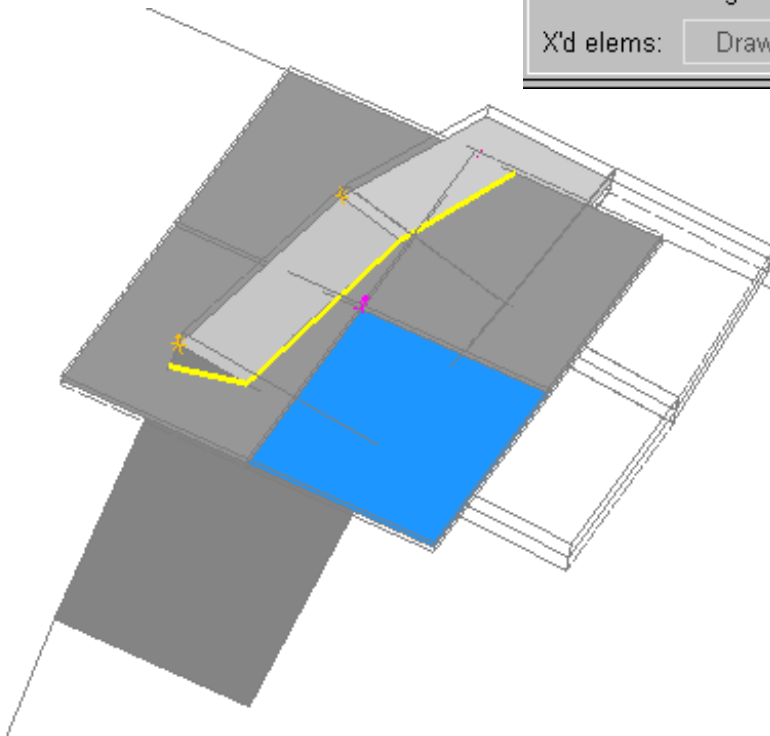


Contact penetrations - Settings

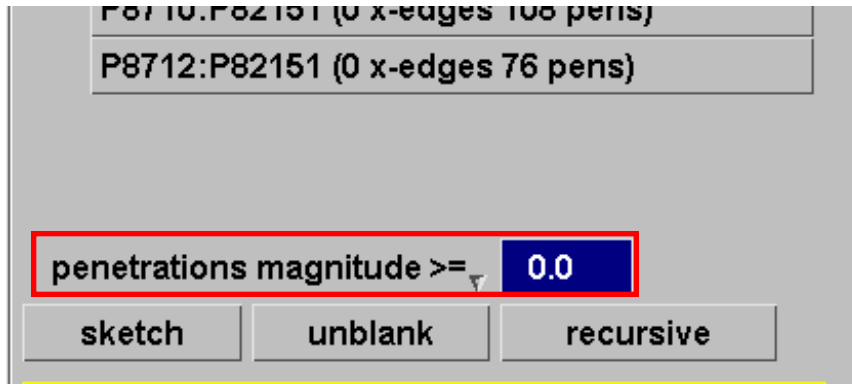
Penetration vectors	DRAWN		
Pen elems:	Drawn	Labels	As thick
Pen nodes:	Drawn	Labels	Distance
Crossed edges	NOT_DRA		
X'd elems:	Drawn	Labels	As thick

Try turning off Crossed edges, press UPDATE.

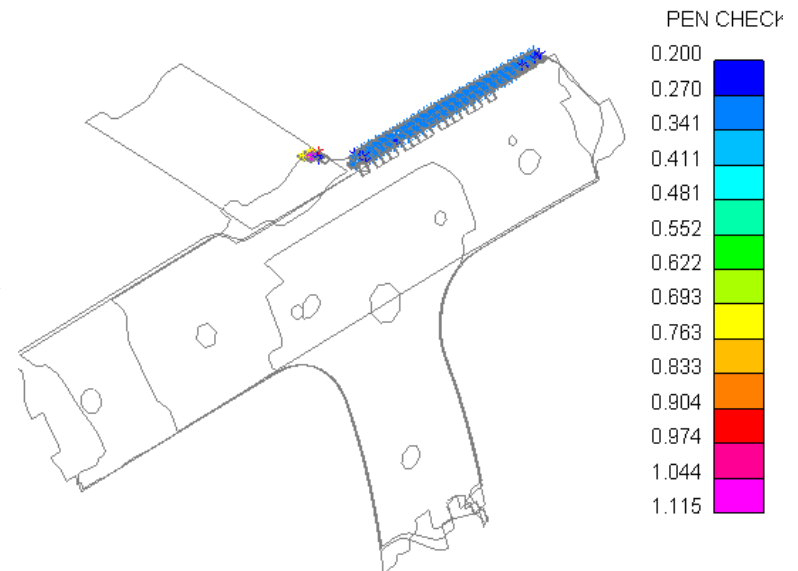
Before proceeding to the next step, turn the crossed edges back on, and turn off "As Thick".



Contact penetrations - Settings



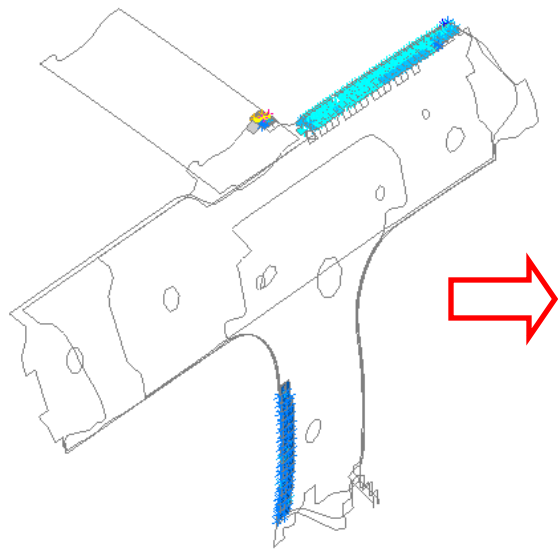
The LS-DYNA control flag IGNORE is recommended – this makes the contact algorithm tolerant to small penetrations. To find out whether any penetrations exceed a certain tolerance – say 0.2mm – input a minimum value in the Pen Check Panel. Update Plot. PRIMER now displays only penetrations greater than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



PEN CHECK M1/CONT2

Dismiss Check all Options...

List Errors Check visible

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE

<No title defined>

select parts sel none sel all sel xedge ?

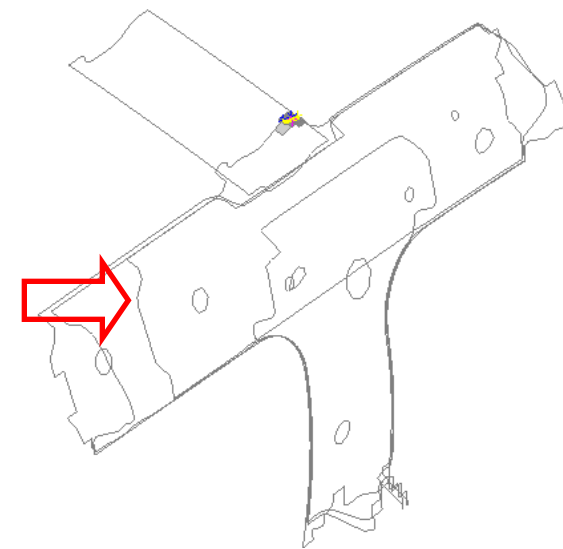
P8231:P8710 (0 x-edges 4 pens)

ratio thickness rem <= 0.8

sketch unblank recursive

contour penetrating Pen: 4 vis 4 total
No crossed edges

CT SI WIRE

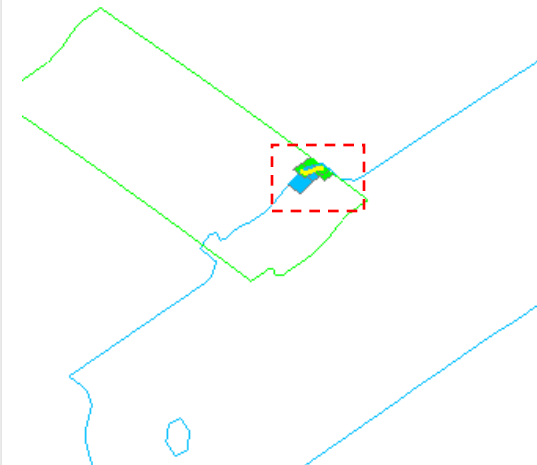


Crossed edges - Fixing

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.



PEN CHECK M1/CONT2

Dismiss Check all Options...
List Errors Check visible

All segments of contact checked

2 AUTOMATIC_SINGLE_SURFACE
<No title defined>

select parts sel none sel all sel xedge ?

P8231:P8710 (5 x-edges 5 pens)
P8710:P82151 (0 x-edges 105 pens)

penetrations magnitude >= 0.2

sketch unblank recursive

contour penetrating Pen: 110 vis 110 total
X'ed: 5 vis 5 total
Min: 2.2175e-001
Max: 1.1148e+000
Sum: 3.4155e+001

CT SI WIRE
Settings...
Levels...
FIX

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

PEN CHECK M1/CONT2

Dismiss Check all Options...
List Errors Check visible

Select nodes to move

Crossed Penetrating End fix
Apply fix Undo last Undo all

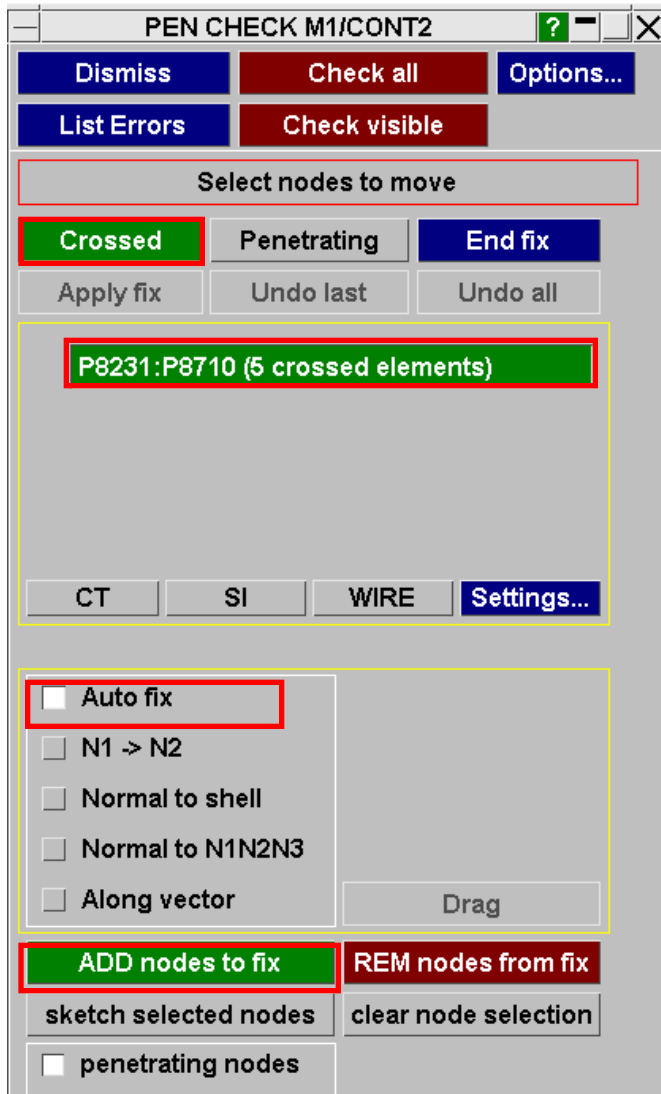
P8231:P8710 (5 crossed elements)

CT SI WIRE Settings...

☐ Auto fix
☐ N1 -> N2
☐ Normal to shell
☐ Normal to N1N2N3
☐ Along vector

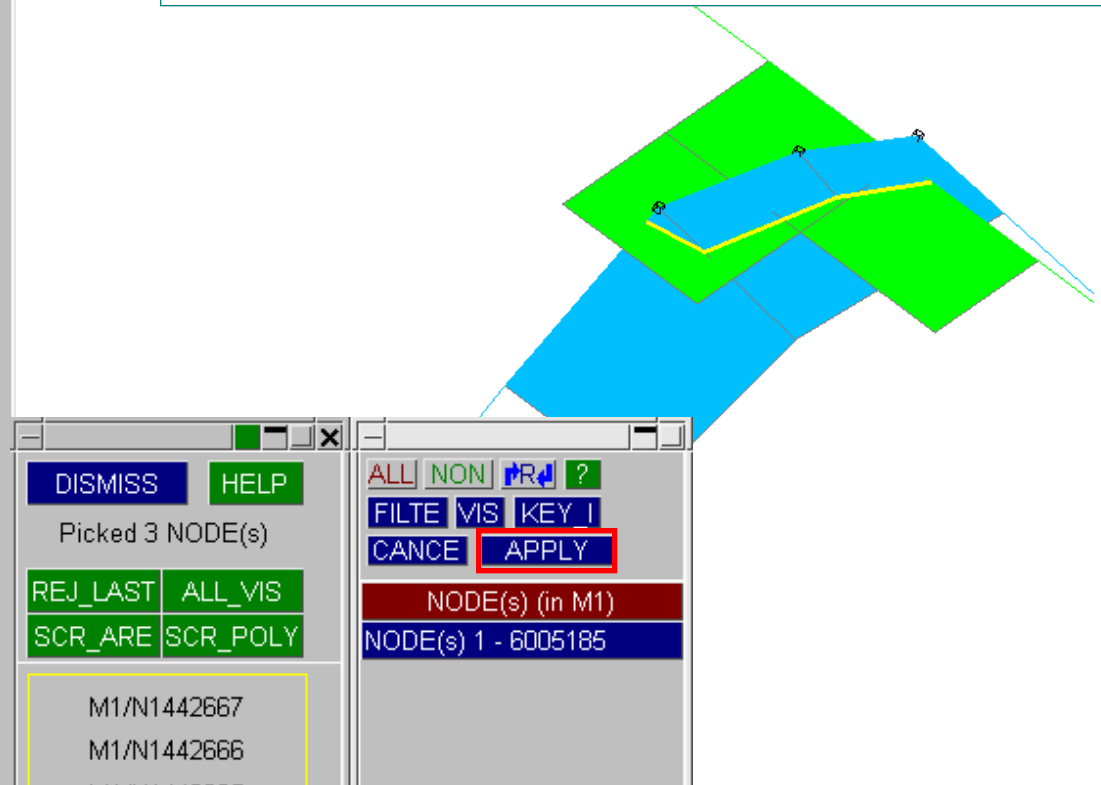
ADD nodes to fix REM nodes from fix
sketch selected nodes clear node selection
☐ penetrating nodes

Crossed edges - Fixing

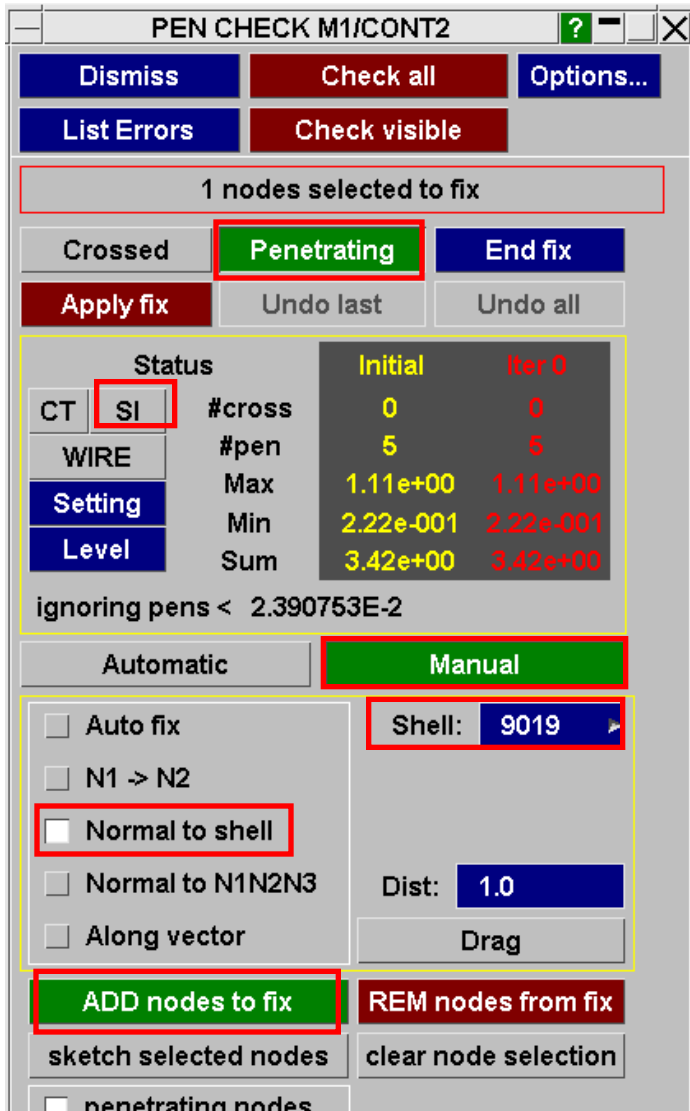


We must tell PRIMER which nodes are on the “wrong” side. Use “ADD nodes to fix” to select the three nodes shown, APPLY to confirm the node selection.

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”.



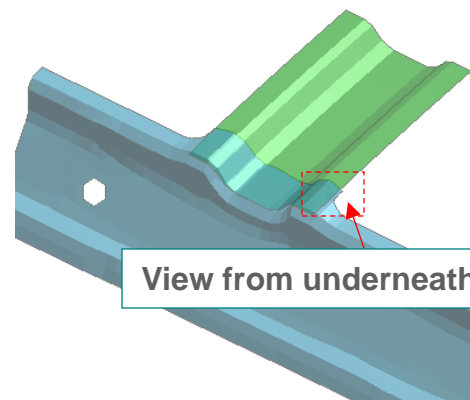
Penetrations – Manual Fixing



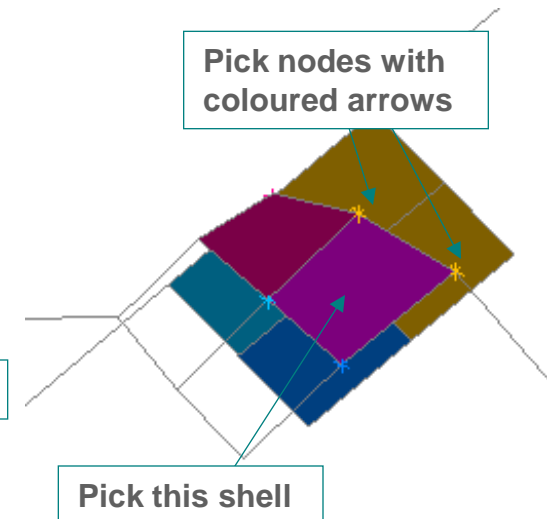
Switch to Penetrating, ensure the menu is switched to Manual, press SI.

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.



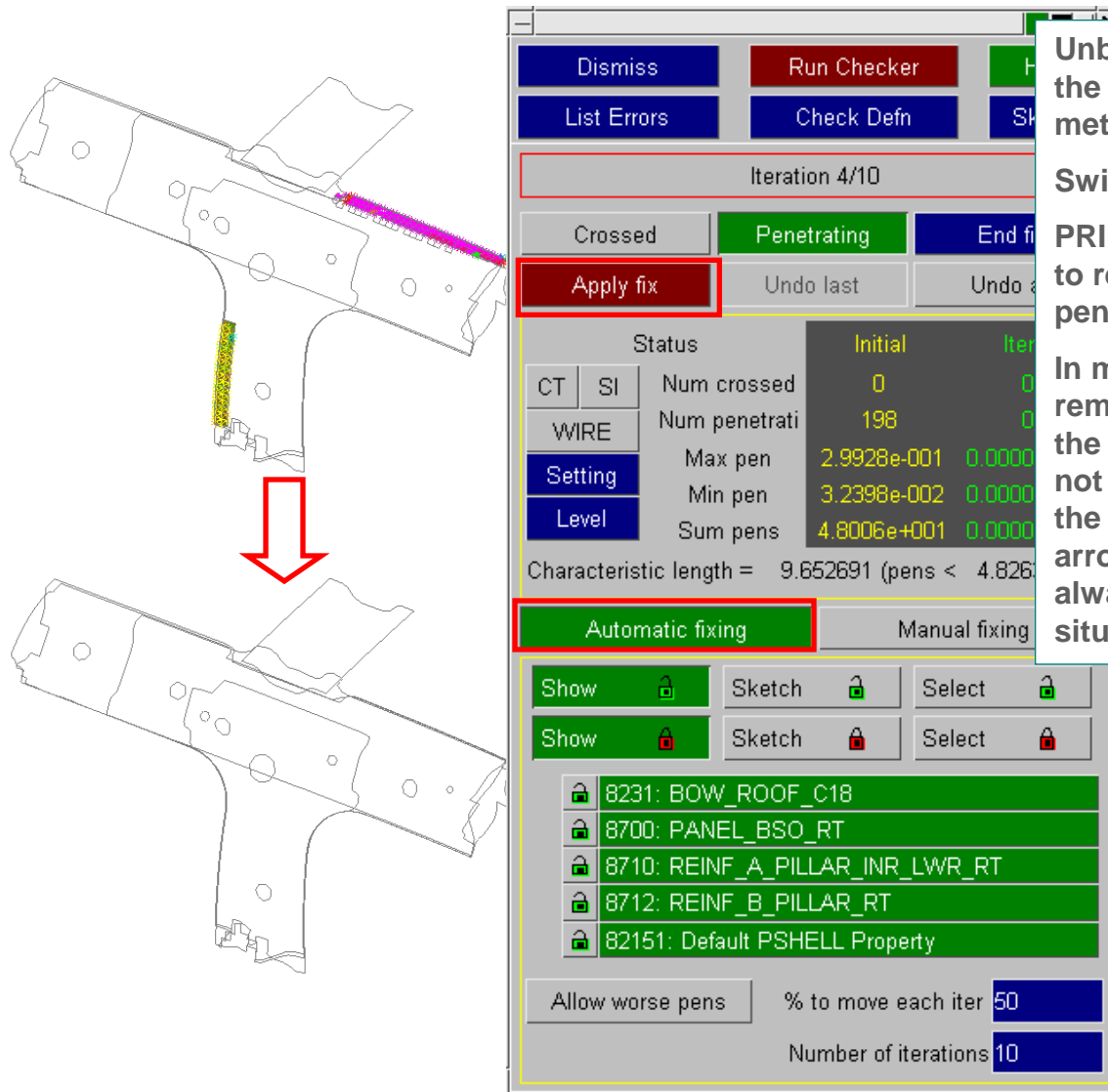
View from underneath



Pick nodes with coloured arrows

Pick this shell

Penetrations – Automatic Fixing



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

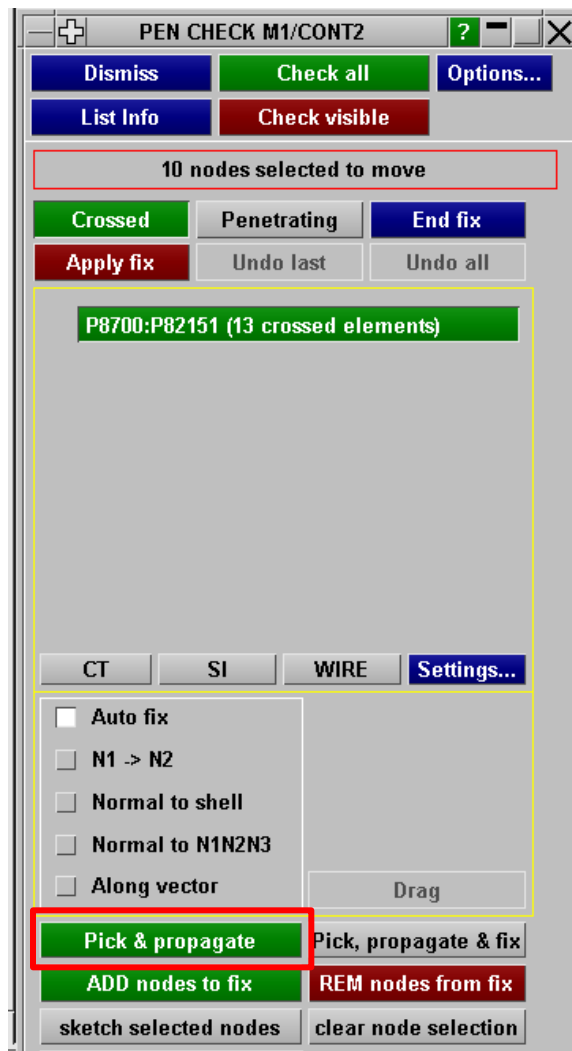
Switch to Automatic Fixing. Press Apply.

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.

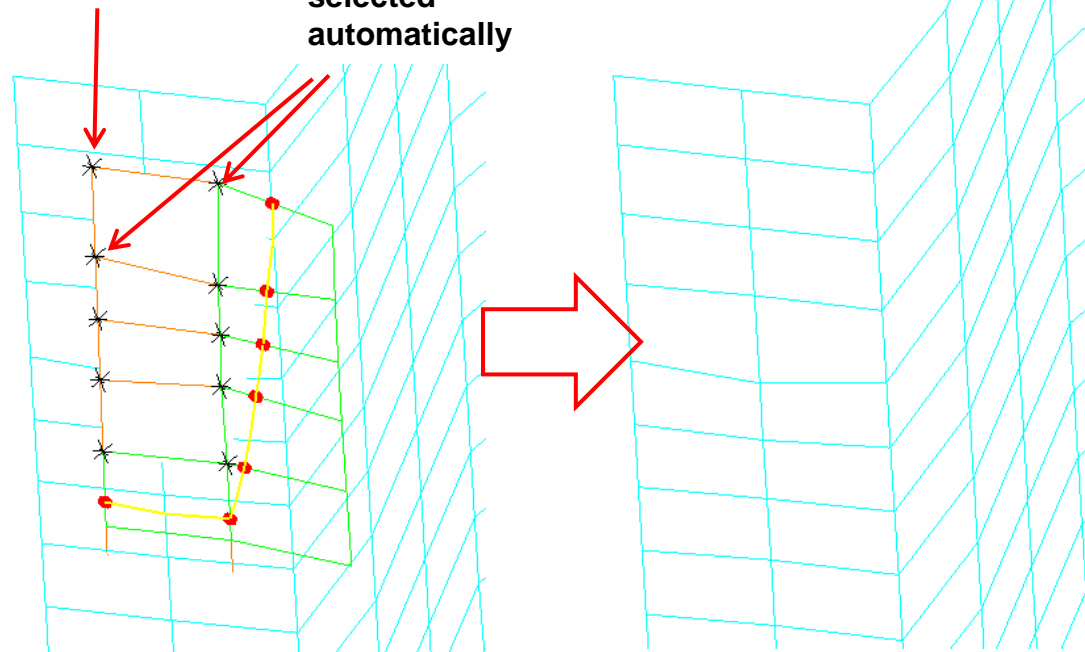
Penetrations: Crossed Edge Fixing

Try this on `crossed_edge.key`

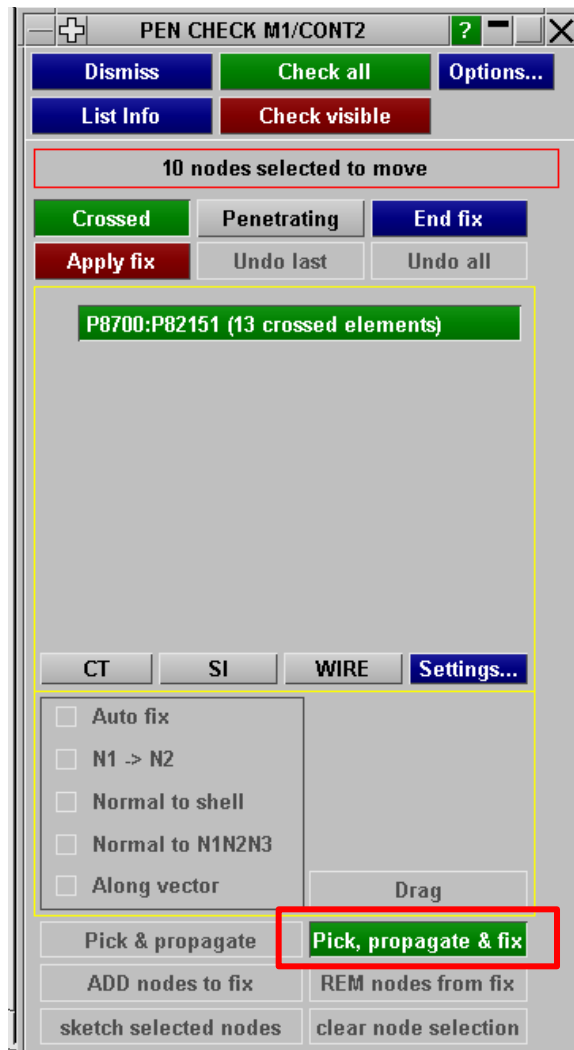


Pick one node

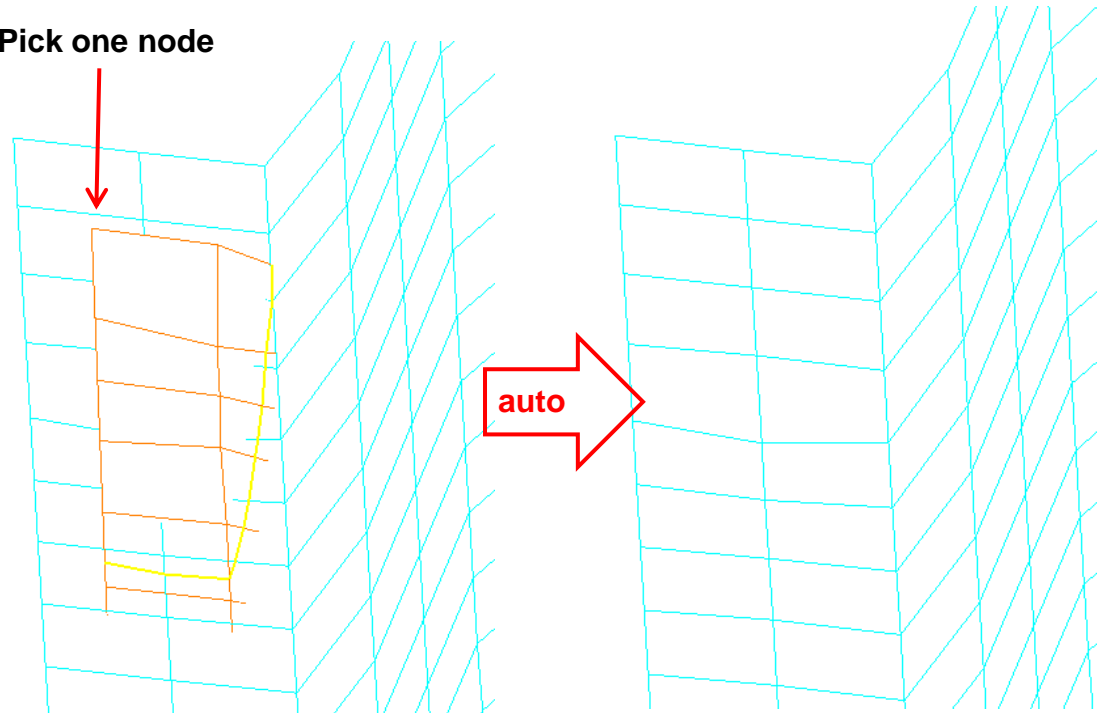
Other nodes are
selected
automatically



“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



Pick one node

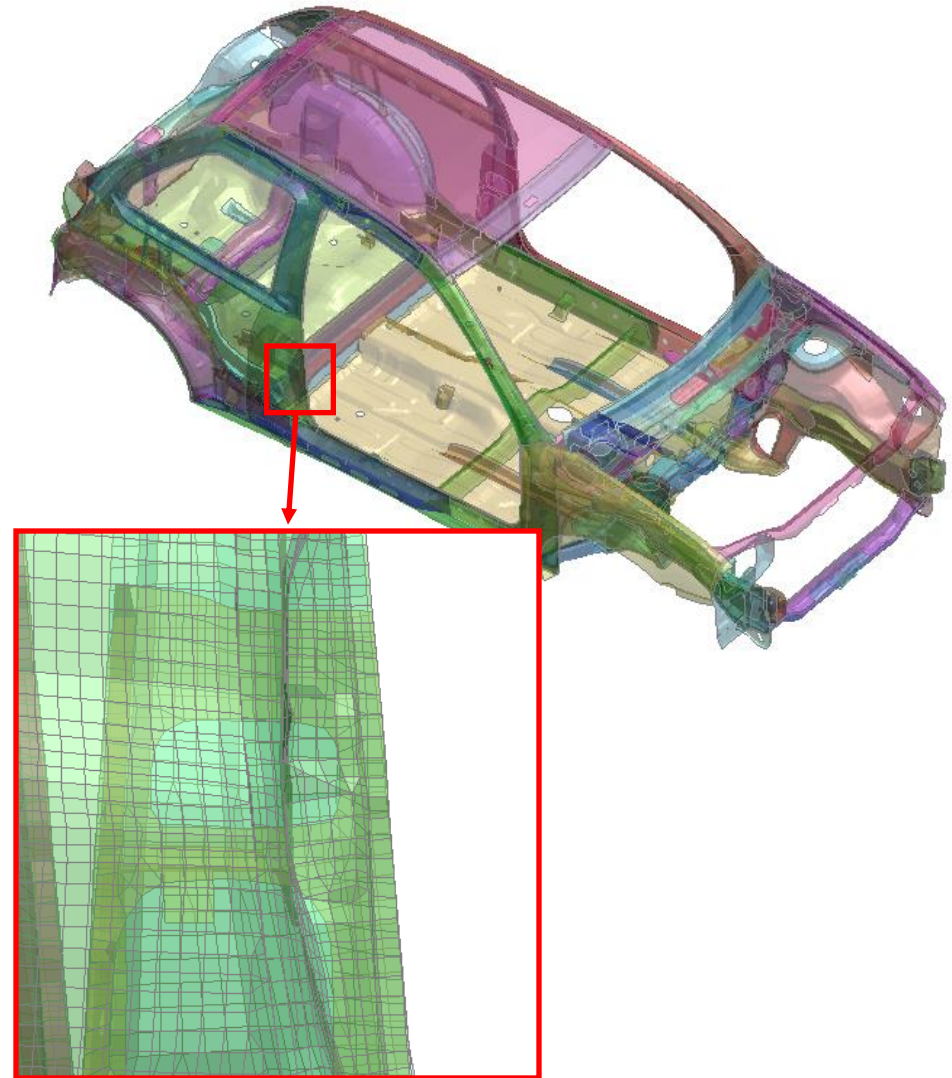


“Pick, propagate & fix” allows user to pick one node; the selection of other nodes that are on the same side of the Crossed Edge, and the fixing, are both performed automatically in a single operation

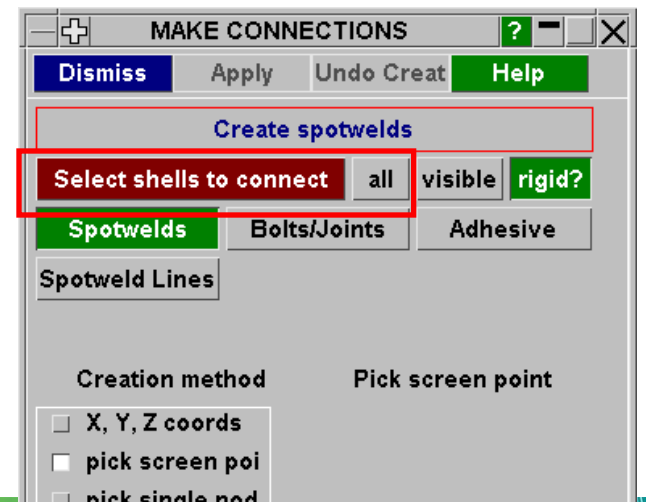
Spotwelding

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

- 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)

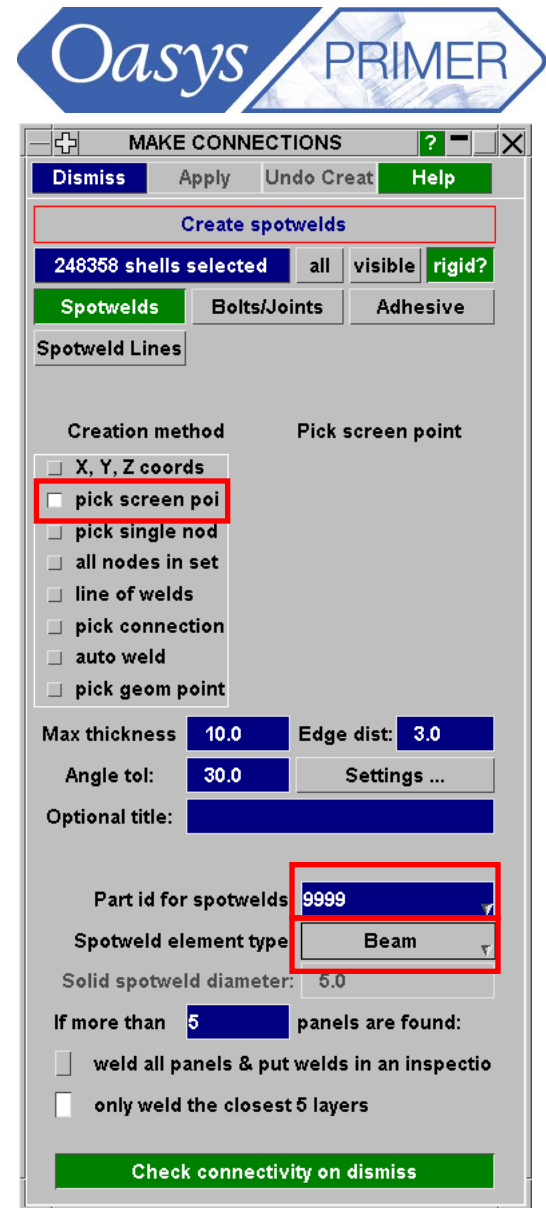
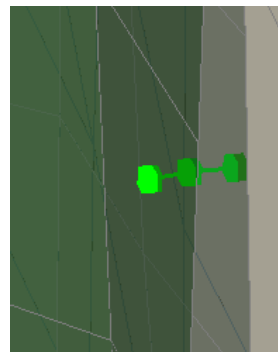
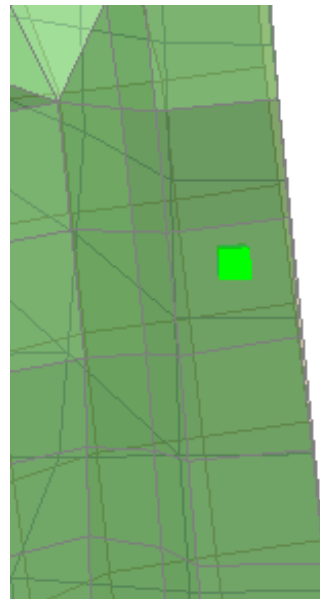


- Tools=>Connection=>Create
- The Connections menu is used for spotwelds and other connection types
- First we have to tell PRIMER which panels can be considered for welding. This model consists only of body panels so we can just select all the shell-element parts in the model.
- Press “Select Shells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



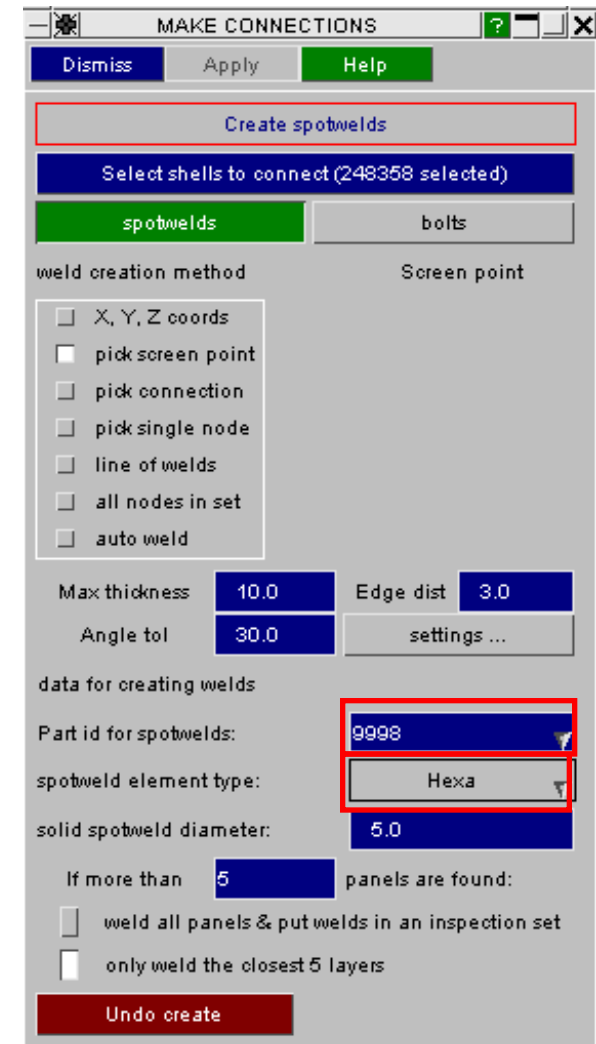
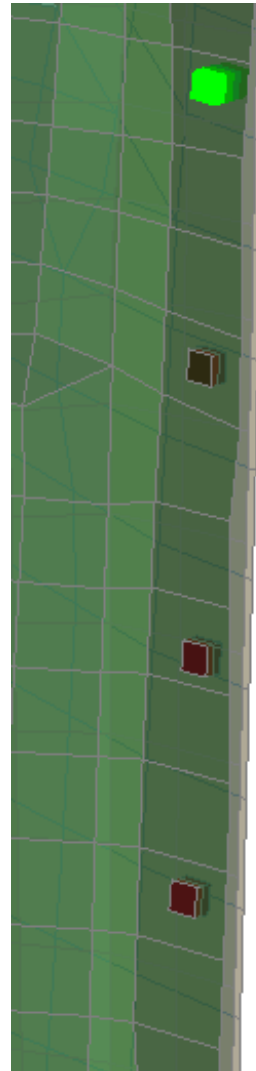
Creating spotwelds

- 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.



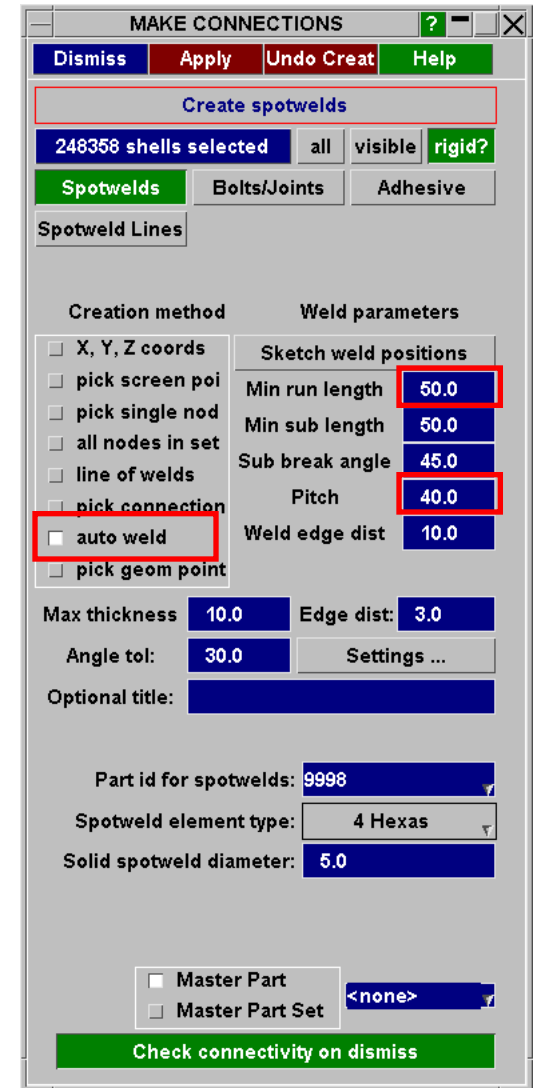
Creating spotwelds

- 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.

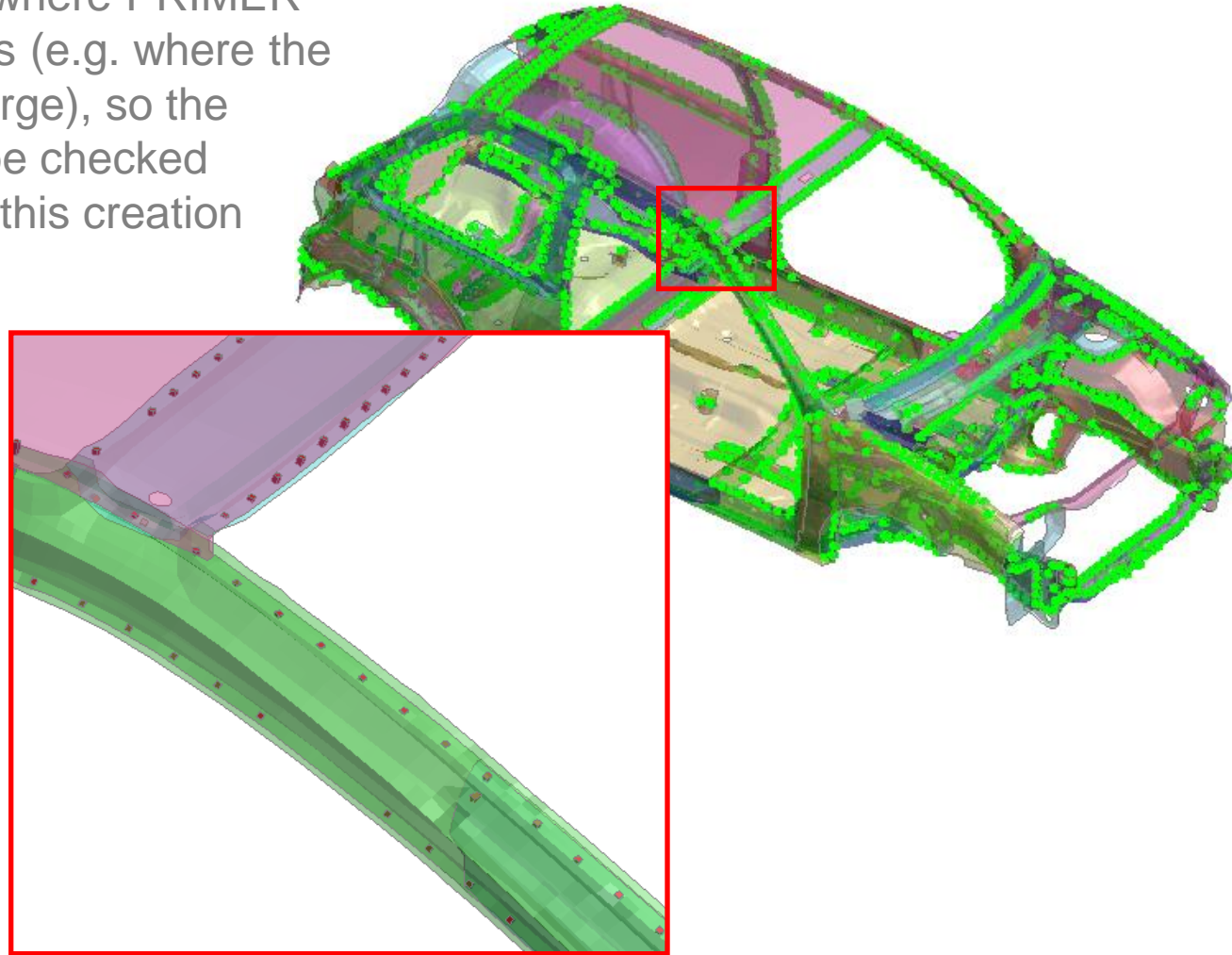


Autoweld

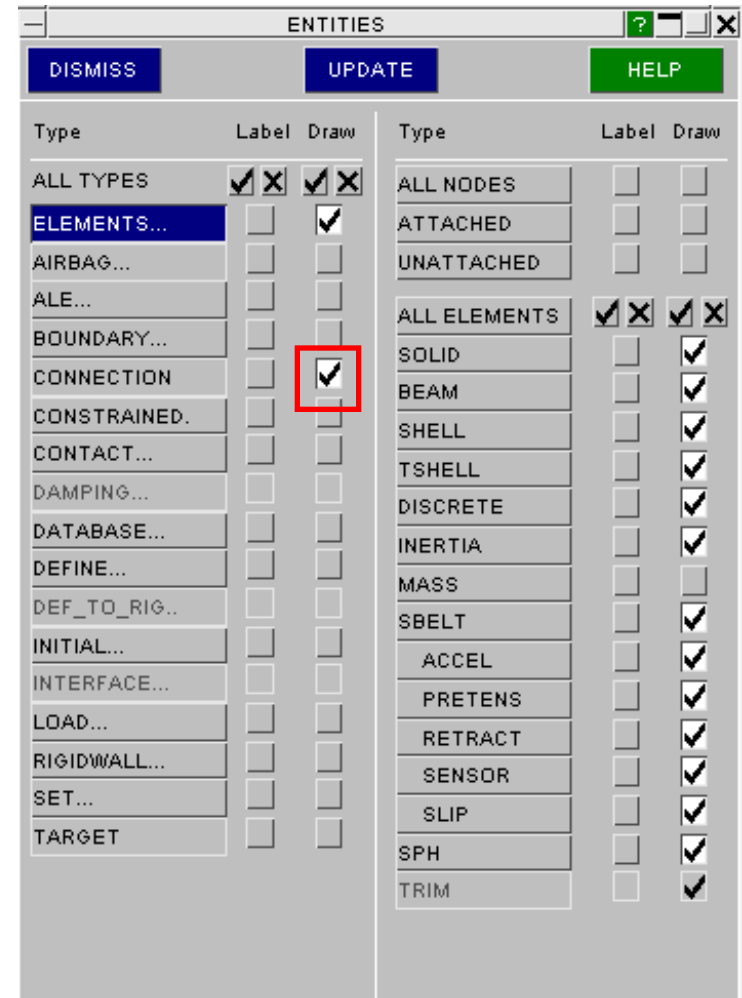
- Switch the creation method to Autoweld. With this option, PRIMER tries to create welds at a user-defined pitch, using the free edges of the panels as a guide.
- This option is used when no information is available about the actual locations of the spotwelds.
- Set the “min run length” to 50, i.e. PRIMER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



- 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!



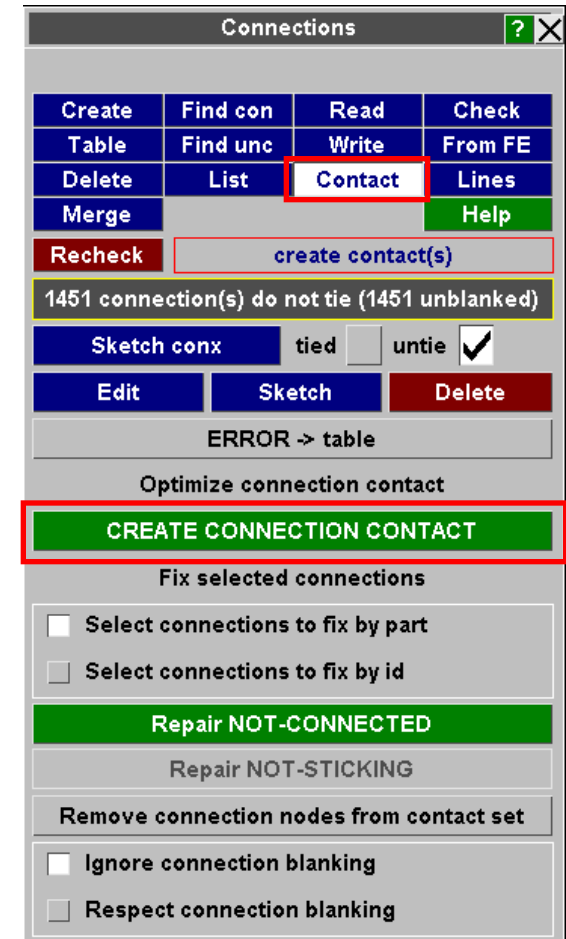
- When creating spotwelds, PRIMER also creates connection entities. These are the green or red circles, coloured according to status.
- If you wish, visibility of these can be switched off using the Entities panel.
- This entity visibility switch will automatically be switched on whenever new spotwelds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

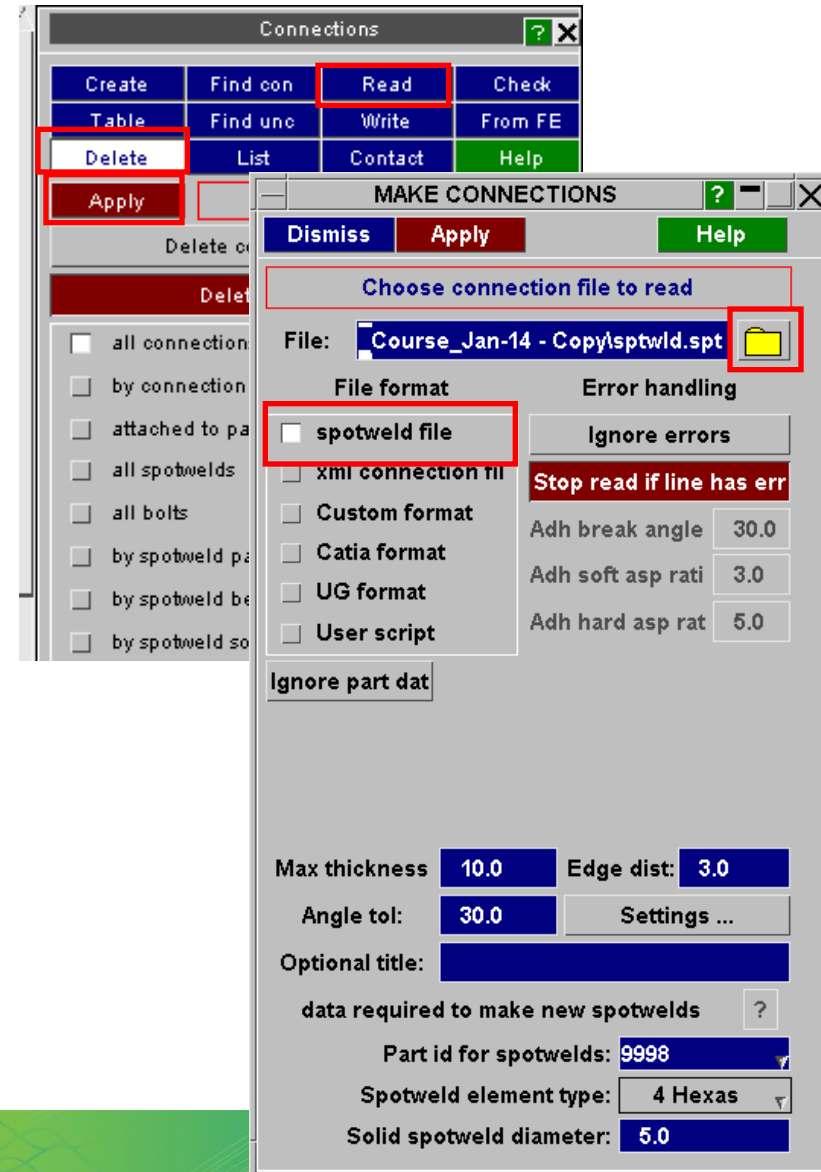
- Dismiss the Spotweld Create menu (e.g. press ESC while the mouse is in that menu)
- To save the newly created spotwelds, we should either write out a keyword file, or write a spotweld file. The spotweld file contains the coordinates of each spotweld, and which panels are connected. The same spotwelds 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.



Create	Find con	Read	Check
Table	Find unc	Write	From FE
Delete	List	Contact	Lines
Merge			Help
Apply		Write spotwelds	
File: <input type="text" value="made_by_autoweld.xml"/>			
Spotwelds	Bolts/Joint	Adhesive	Spot lines
<input type="checkbox"/> all connections			
<input type="checkbox"/> by connection id			
<input type="checkbox"/> by panels			
<input type="checkbox"/> by attached panels			
<input type="checkbox"/> by spotweld part			
<input type="checkbox"/> by spotweld beam			
<input type="checkbox"/> by spotweld solid			
<input type="checkbox"/> by adhesive part			
<input type="checkbox"/> by multiple seams			
<input type="checkbox"/> by single seam			
<input type="checkbox"/> by connection title			
<input type="checkbox"/> spotweld file			
<input checked="" type="checkbox"/> xml connection file			
<input type="checkbox"/> UG format			
<input type="checkbox"/> IGES format			
<input type="checkbox"/> User script			

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



- 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.

MAKE CONNECTIONS

CANCEL < PREV APPLY HELP

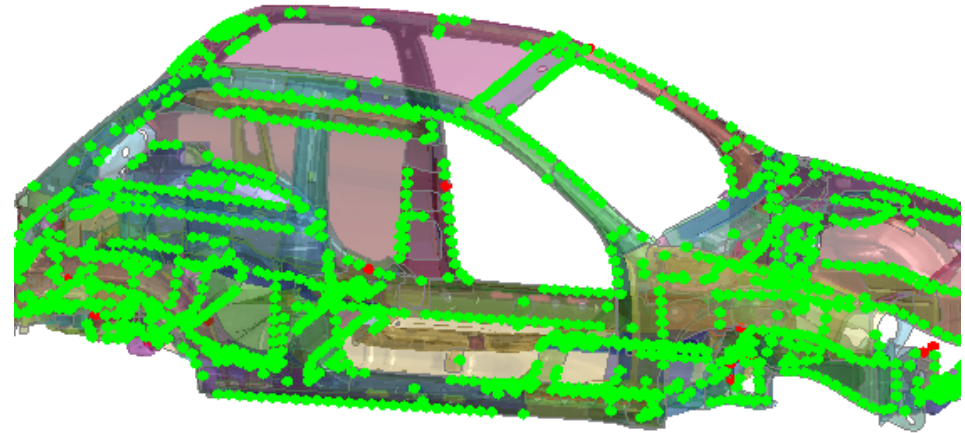
Drop-down menu to change meaning of column of data

Drag the yellow handles to change column widths. Columns for the X, Y and Z coordinates must be chosen. Part ID and weld ID c

Column 1	41	51	61	71	81	91				
Field	Skip field	Weld ID	Skip field	X coord	Y coord	Z coord	Skip field	Part ID	Part ID	Part ID
SPOTWELD	1	POINT	-1438.2809	649.83295	355.80676	PARTS	10000043	10000000	10000037	
SPOTWELD	2	POINT	-1399.597	650.64423	365.72354	PARTS	10000043	10000000	10000002	
SPOTWELD	3	POINT	-1367.7036	650.31738	389.37738	PARTS	10000001	10000002		

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.



Dismiss	View...	Options...	Refresh	Action: update & remake					
Apply:	Undo	All	Selected	Changed	Autoscale	Clear	Set all	Select	
ID	Diameter	Part ID	Status	Error	Details	Layer 1	Layer 2	Layer 3	
3	5	9998	Bad	The eleme	None	P10000001	P10000002		
4	5	9998	Bad	The eleme	None	P10000001	P10000002		
18	5	9998	Bad	The eleme	None	P10000001	P10000002		
19	5	9998	Bad	All pairs of	None	P10000001	P10000002		
20	5	9998	Bad	The eleme	None	P10000001	P10000002		
71	5	9998	Bad	Element w	Spotweld c	P10000043	P10000004	P10000070	
85	5	9998	Bad	Element would be too distorted (warped) due to mesh mismatch				P10000004	
105	5	9998	Bad					P10000043	
206	5	9998	Bad	Element w	Spotweld c	P10000017	P10000038	P10000016	

- 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

The screenshot shows the Oasys Connection Table window. The 'Action' dropdown menu is open, showing 'update & remake'. The 'Apply' button is highlighted with a red box. The 'View...' button is also highlighted with a red box. The 'Options...' button is highlighted with a red box. The 'Refresh' button is highlighted with a red box. The 'Autoscale' button is highlighted with a green box. The 'Clear' button is highlighted with a blue box. The 'Sel all' button is highlighted with a blue box. The 'Select' button is highlighted with a blue box. The table contains the following data:

ID	Diameter	Part ID	Status	Error	Details	Layer 1	Layer 2	Layer 3
3	5	9998	Bad	The eleme	None	P10000001	P10000002	
4	5	9998	Bad	The eleme	None	P10000001	P10000002	
18	5	9998	Bad	The eleme	None	P10000001	P10000002	
19	5	9998	Bad	All pairs of	None	P10000001	P10000002	
20	5	9998	Bad	The eleme	None	P10000001	P10000002	
71	5	9998	Bad	Element w	Spotweld c	P10000043	P10000004	P10000070
85	5	9998	Bad	Element would be too distorted (warped) due to mesh mismatch			P10000004	
105	5	9998	Bad				P10000043	
206	5	9998	Bad	Element w	Spotweld c	P10000017	P10000038	P10000016

Annotations in the image include:

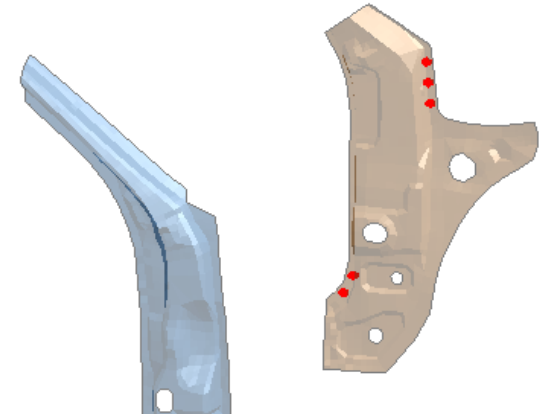
- A red box around the 'Apply' button.
- A red box around the 'View...' button.
- A red box around the 'Options...' button.
- A red box around the 'Refresh' button.
- A red box around the 'Action: update & remake' dropdown.
- A green box around the 'Autoscale' button.
- A blue box around the 'Clear' button.
- A blue box around the 'Sel all' button.
- A blue box around the 'Select' button.
- Blue text: 'Select welds here, or by clicking in the table'.

- 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 panels”

CONNECTION TABLE													
Dismiss		View...	Options...	Refresh	Action: update & remake		Show all	Spotweld	Adhesive	Write...			
Apply: Undo		All	Selected	Changed	Autoscale	Clear	Sel all	Select	Show sel	Bolts/Joi	Spot line	Set colu	
ID	Type	Subtype	Part ID	Status	Error	Details	Diam	P1	Layer 1	Layer 2	Layer 3	Layer 4	contact id
CNX1	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1438.281	P10000043	P10000000	P10000037		
CNX3	Cannot change ID		98	Bad	LAYER - Co	Only one la	5	-1367.704	P10000001	P10000002			
CNX4	Update & remake		98	Bad	LAYER - Co	Only one la	5	-1348.509	P10000001	P10000002			
CNX8	Sketch conx		98	Bad	Element wo	Spotweld c	5	-1365.643	P10000043	P10000000	P10000002		
CNX15	Sketch FF		98	Bad	Element wo	Spotweld c	5	-1404.0 63	P10000043	P10000000	P10000004		
CNX16	Show conx & panels		98	Bad	Inconsisten	Inconsisten	5	-1340.14 6	P10000043	P10000000	P10000004		
CNX17	Empty conx		98	Bad	Element wo	Spotweld c	5	-1275.359	P10000043	P10000000	P10000002		
CNX18	Delete conx		98	Bad	LAYER - Co	Only one la	5	-1195.257	P10000001	P10000002			
CNX19	Upd & remake (repos)		98	Bad	All pairs of	None	5	-1193.127	P10000001	P10000002			
CNX20	Upd & remake (swap)		98	Bad	LAYER - Co	Only one la	5	-1190.346	P10000001	P10000002			
CNX30	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1437.56 -6	P10000037	P10000001	P10000044		
CNX31	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1399.146 -	P10000003	P10000001	P10000044		

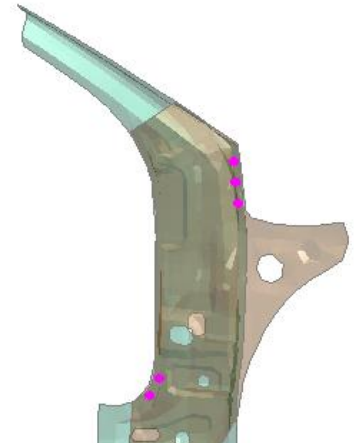
Fixing errors

- 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.

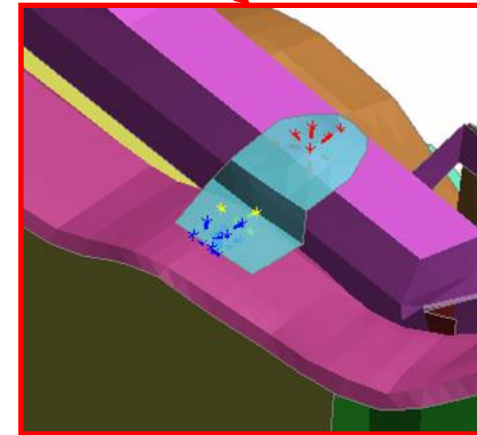
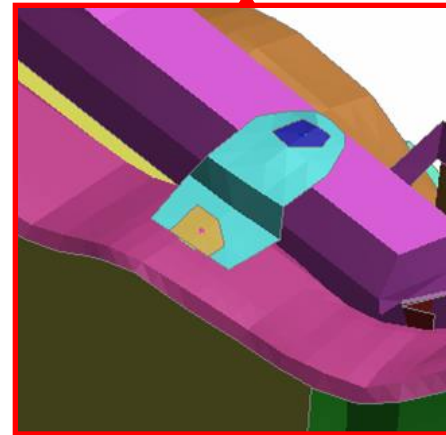
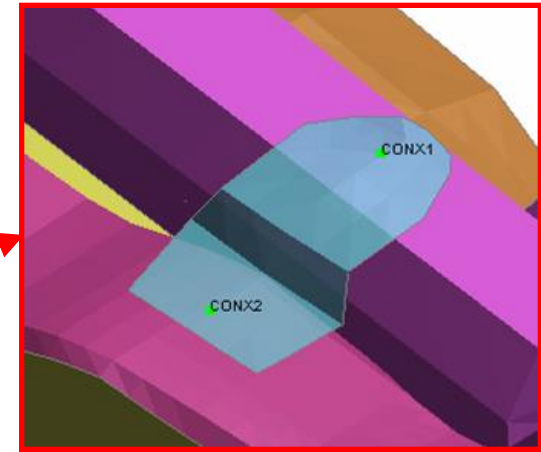
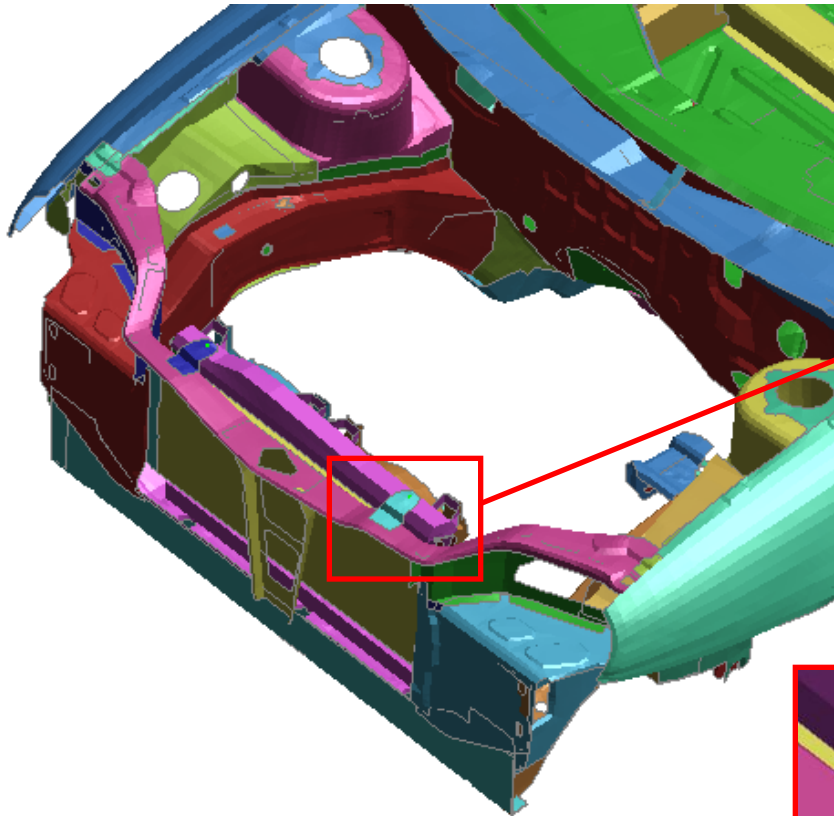


CONNECTION TABLE													
Dismiss		View...	Options...	Refresh	Action: update & remake		Show all	Spotweld	Adhesive	Write...			
Apply: Undo		All	Selected	Changed	Autoscale	Clear	Sel all	Select	Show sel	Bolts/Joi	Spot line	Set colu	
ID	Type	Subtype	Part ID	Status	Error	Details	Diam	P1	Layer 1	Layer 2	Layer 3	Layer 4	contact id
CNX1	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1438.281	P10000043	P10000000	P10000037		
CNX3	SPOTWEL	4 solids	9998	Bad	LAYER - Co	Only one la	5	-1367.704	P10000001				
CNX4	SPOTWEL	4 solids	9998	Bad	LAYER - Co	Only one la	5	-1348.509	P10000001				
CNX8	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1365.643	P10000001				
CNX15	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1404.0 63	P10000001				
CNX16	SPOTWEL	4 solids	9998	Bad	Inconsisten	Inconsisten	5	-1340.14 6	P10000001				
CNX17	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1275.359	P10000001				
CNX18	SPOTWEL	4 solids	9998	Bad	LAYER - Co	Only one la	5	-1195.257	P10000001				
CNX19	SPOTWEL	4 solids	9998	Bad	All pairs of	None	5	-1193.127	P10000001				
CNX20	SPOTWEL	4 solids	9998	Bad	LAYER - Co	Only one la	5	-1190.346	P10000001				
CNX30	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1437.56 -6	P10000001				
CNX31	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1399.146 -	P10000001				

- 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.

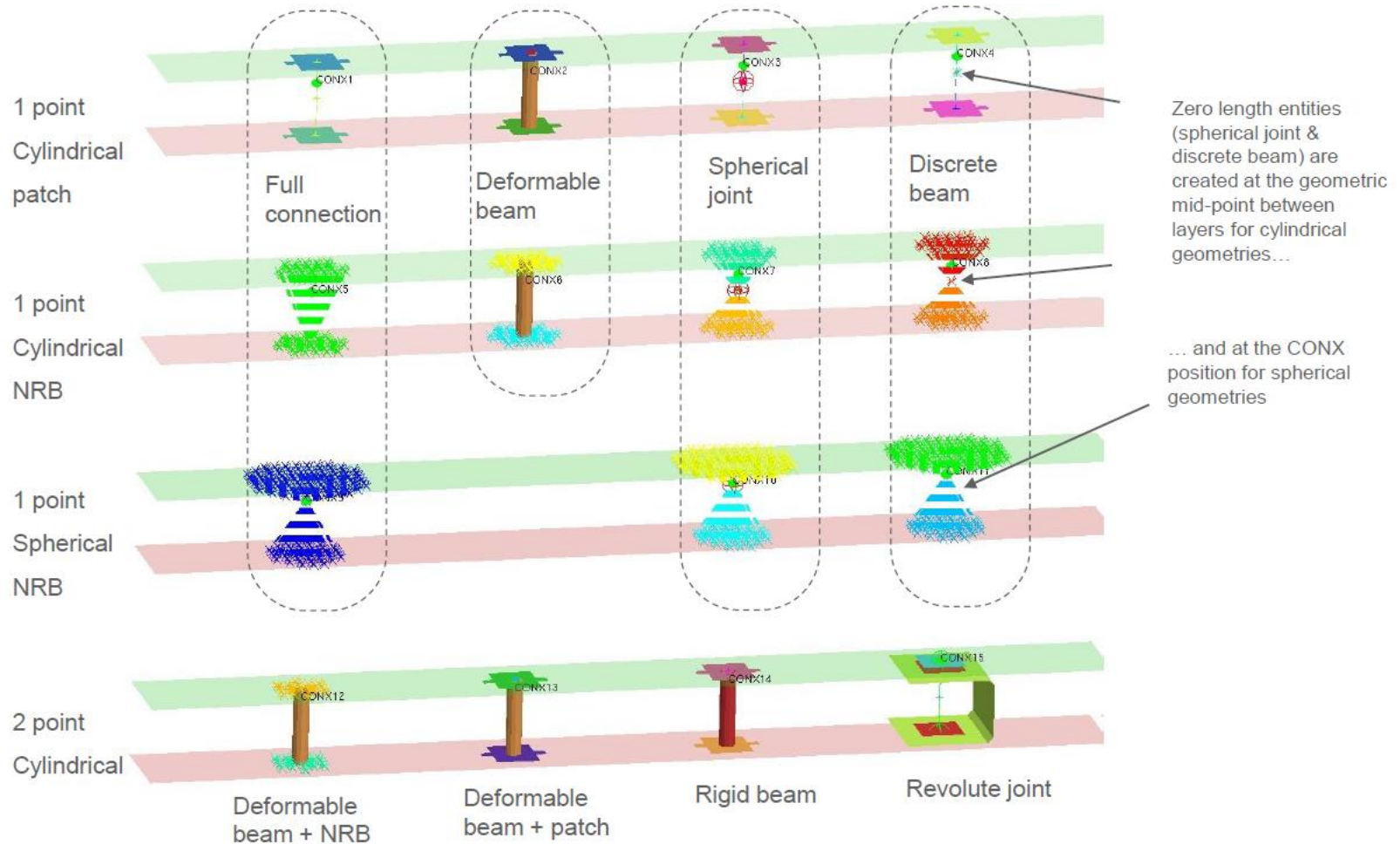


CONNECTION TABLE													
Dismiss		View...	Options...	Refresh	Action: update & remake		Show all	Spotweld	Adhesive	Write...			
Apply: Undo		All	Selected	Changed	Autoscale	Clear	Sel all	Select	Show sel	Bolts/Joi	Spot line	Set colu	
ID	Type	Subtype	Part ID	Status	Error	Details	Diam	P1	Layer 1	Layer 2	Layer 3	Layer 4	contact id
CNX1	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1438.281	P10000043	P10000000	P10000037		
CNX3			9998	Unchecked	LAYER - Co	Only one la	5	-1367.704	P10000000	P10000002			
CNX4			9998	Unchecked	LAYER - Co	Only one la	5	-1348.509	P10000000	P10000002			
CNX8			9998	Bad	Element wo	Spotweld c	5	-1365.643	P10000043	P10000000	P10000002		
CNX15	Sketch conx		9998	Bad	Element wo	Spotweld c	5	-1404.0 63	P10000043	P10000000	P10000004		
CNX16	Sketch FE		9998	Bad	Inconsisten	Inconsisten	5	-1340.14 6	P10000043	P10000000	P10000004		
CNX17	Show conx & panels		9998	Bad	Element wo	Spotweld c	5	-1275.359	P10000043	P10000000	P10000002		
CNX18	Empty conx		9998	Unchecked	LAYER - Co	Only one la	5	-1195.257	P10000000	P10000002			
CNX19	Delete conx		9998	Unchecked	All pairs of	None	5	-1193.127	P10000000	P10000002			
CNX20	Upd & remake (repos)		9998	Unchecked	LAYER - Co	Only one la	5	-1190.346	P10000000	P10000002			
CNX30	Upd & remake (swap)		9998	Bad	Element wo	Spotweld c	5	-1437.56 -6	P10000037	P10000001	P10000044		
CNX31	SPOTWEL	4 solids	9998	Bad	Element wo	Spotweld c	5	-1399.146 -	P10000003	P10000001	P10000044		

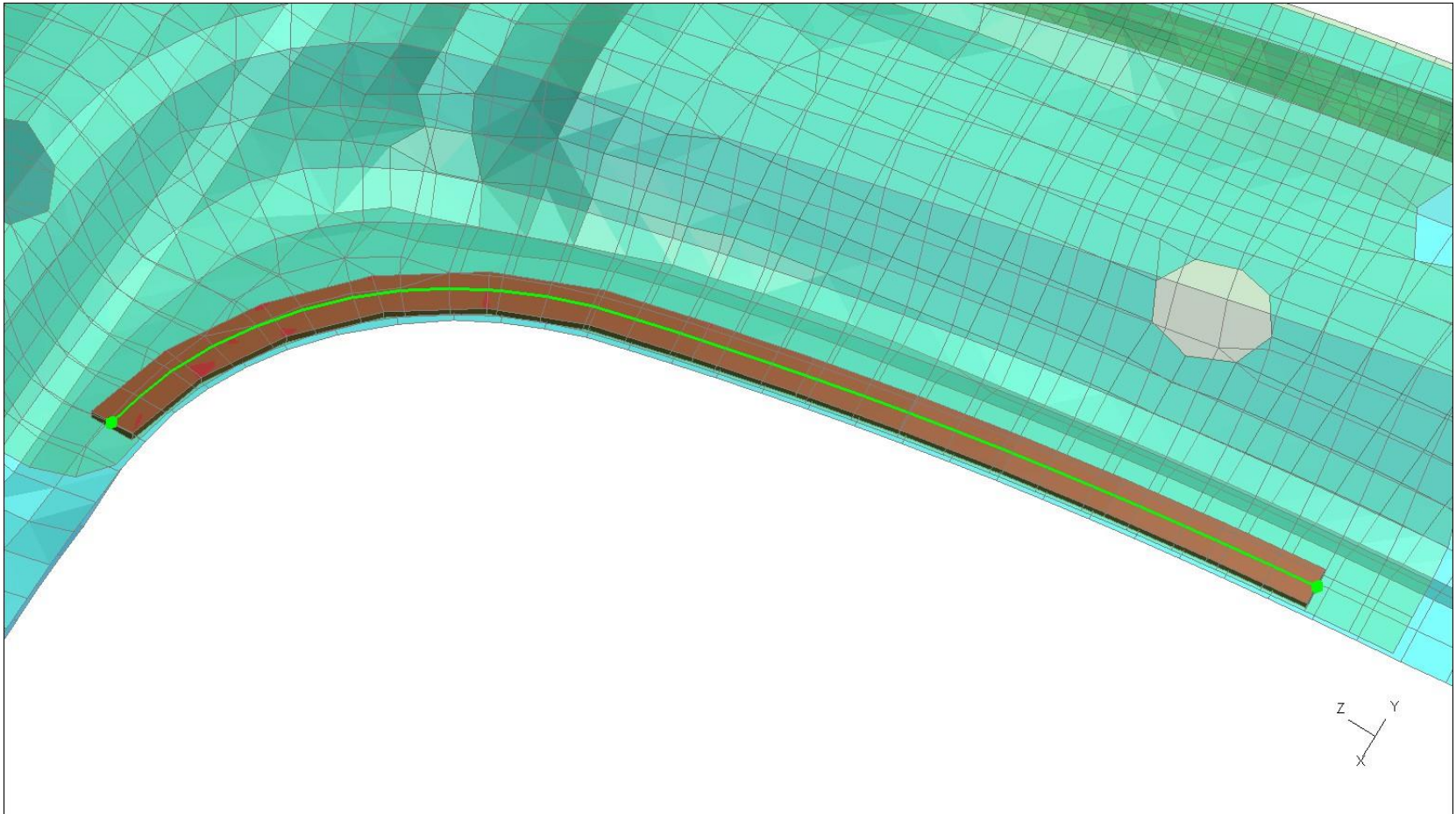


Connections may be spotwelds or bolts. Each type can be created interactively or from a file. Bolt connections can be “realized” as rigid patches or nodal rigid bodies

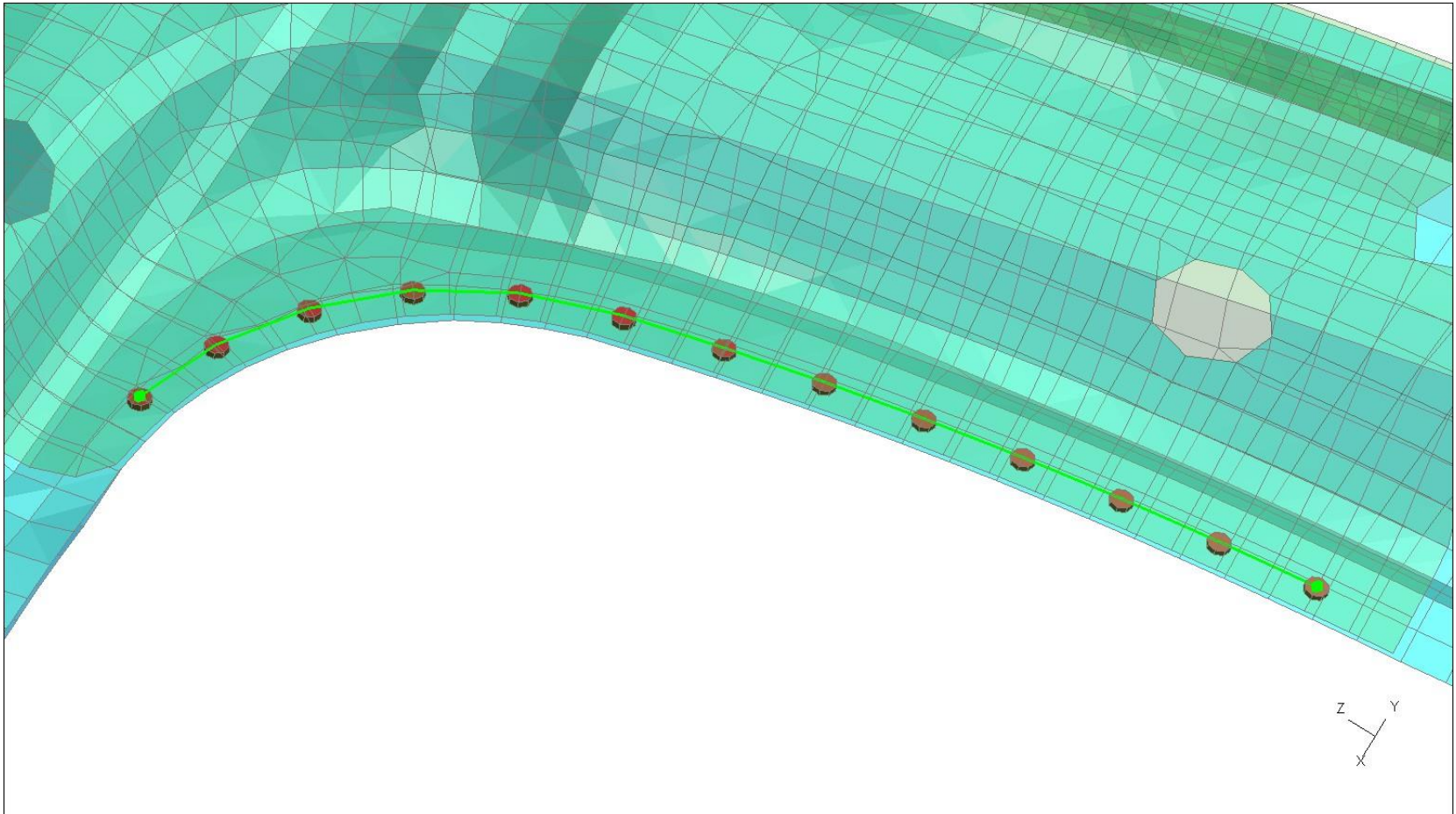
- Various bolt/joint connections can be created in PRIMER:



- Runs of solid adhesive can be created in PRIMER:



- Lines of spotwelds can be created in PRIMER:



- 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”

- 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

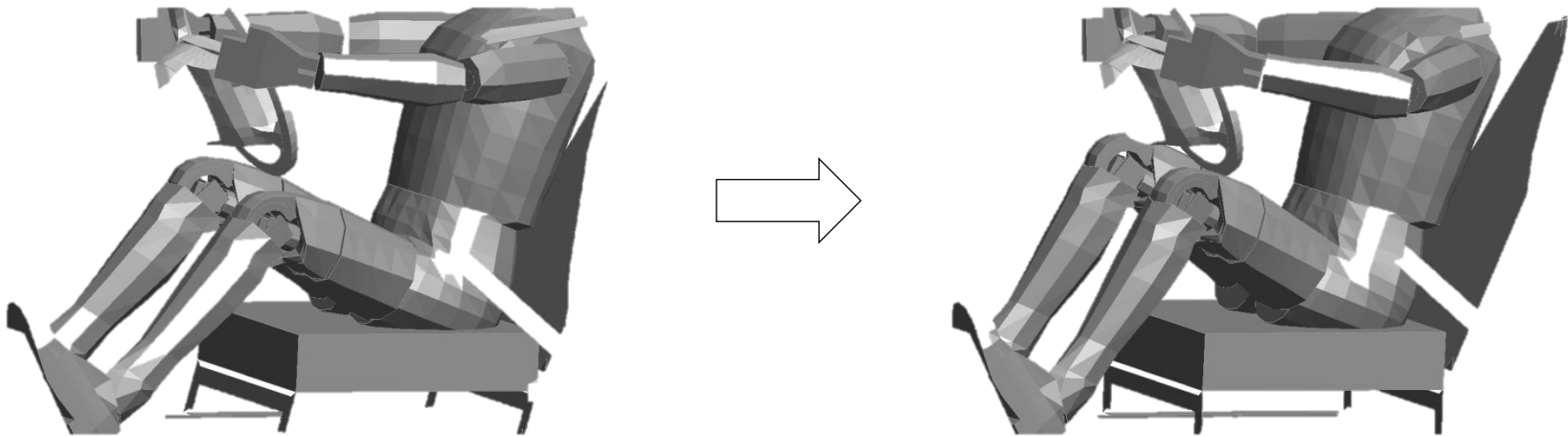


- Seat mechanisms and dummies can be linked so that any seat movement will lead to the dummy moving as well.

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.

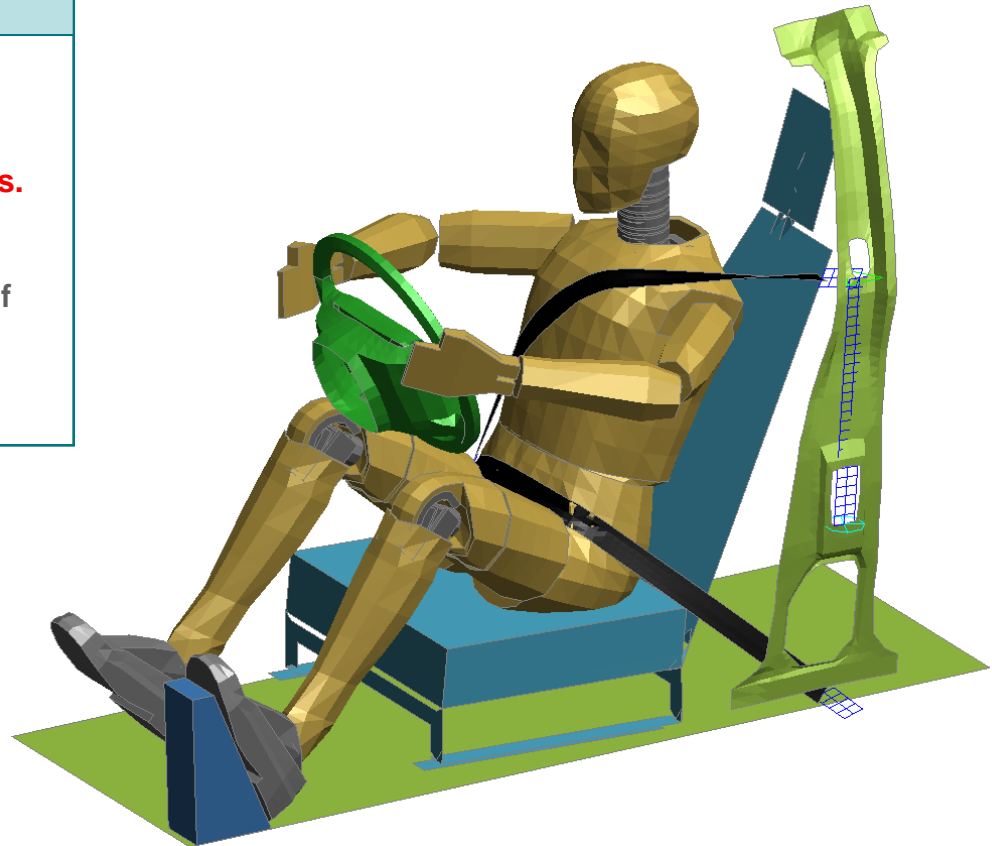


- After the occupant has been moved, the belt can easily be refitted over the dummy in the new position.

Occupants

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

1. In the Tools menu, click **Occupant->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**.



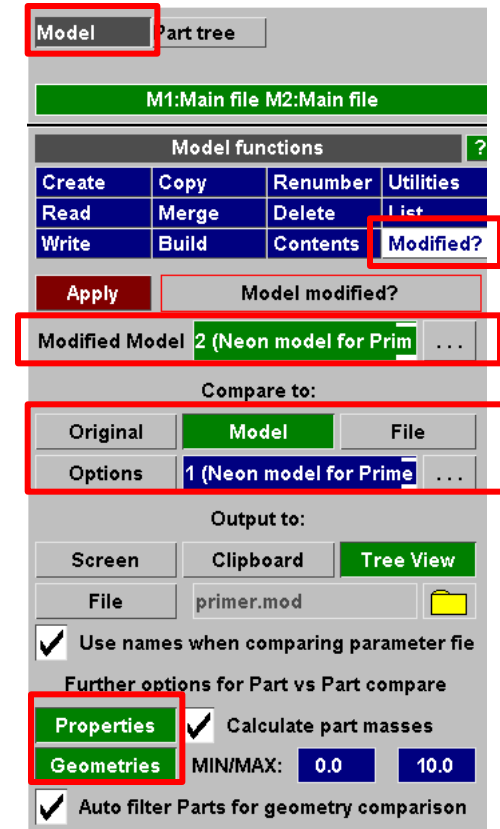
Model modified

- Often, it is useful to see differences between models

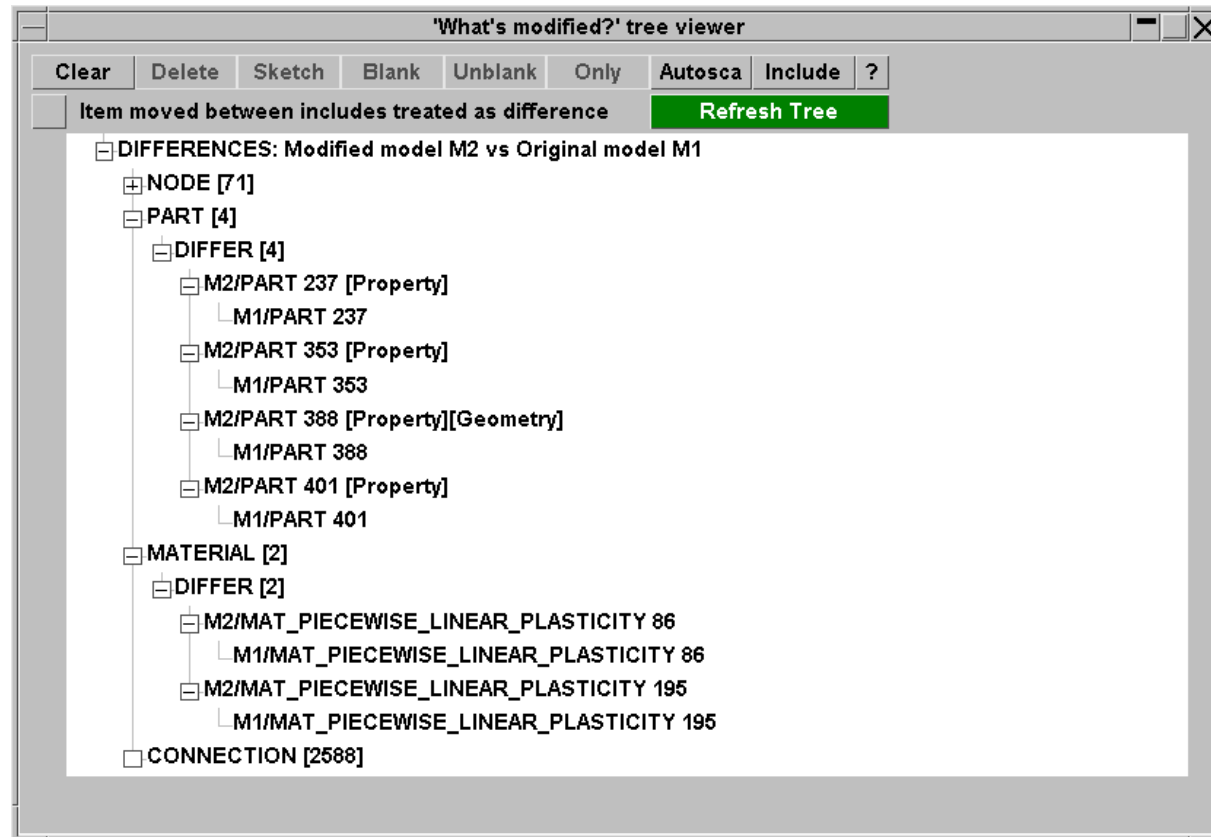
Model Modified

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

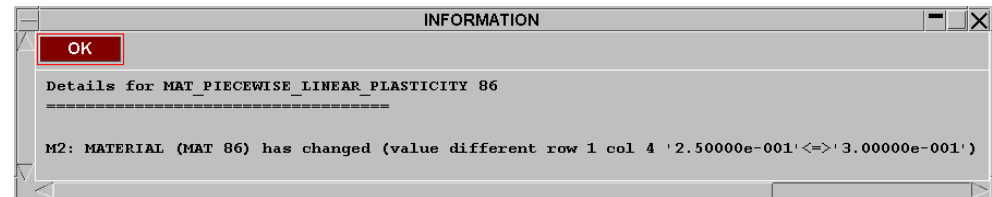
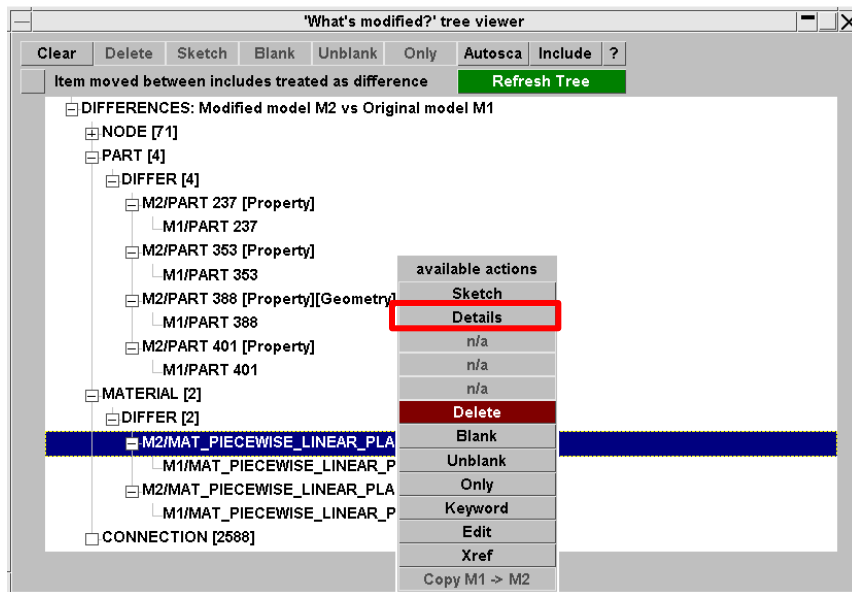
1. Go to **Model->Modified**
2. Select model 2 for the “Modified model”.
3. Select to compare to model 1.
4. Ensure that the **Properties** and **Geometries** switches are on.
5. Click **Apply**.



- 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.

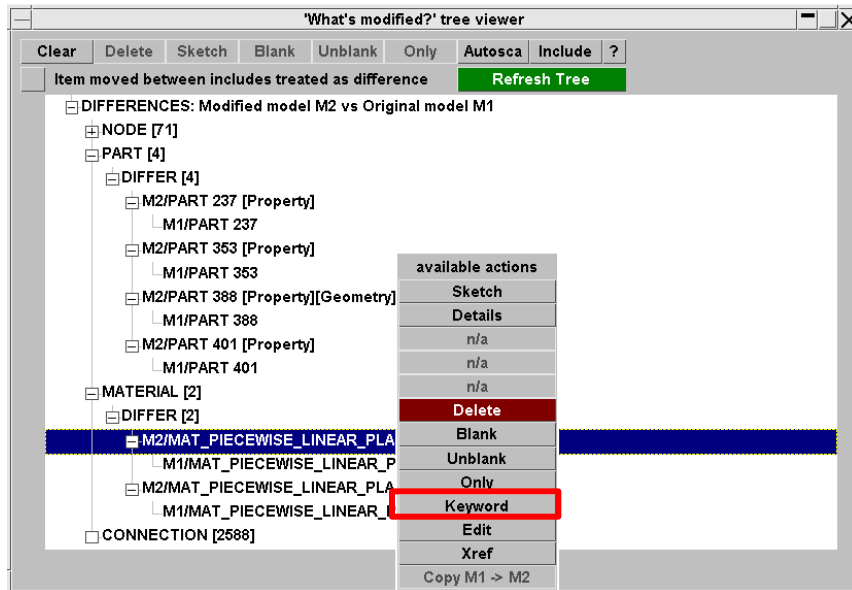


- Various options are available when right mouse clicking on one of the differences:
 - **Details** will print details of the differences found.



Model modified

- 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.



Keyword: M2/MATERIAL

CANCEL RESET_ALL HELP

UPDATE CHECK_ALL SKETCH_ALL

Keyword M2 MATERIAL (1/0 mod)

Filter by: MAT <auto> <auto> <auto>

#	Options..	Incl	Surfaces	MID	La	RO	F	E	F	PR	F	SIGY	F	ETAN	F	FAIL	F	TDEL	F
				C	F	P	F	LCSS	F	LCSR	F	VP	F						
				EPS1	F	EPS2	F	EPS3	F	EPS4	F	EPS5	F	EPS6	F	EPS7	F	EPS8	F
				ES1	F	ES2	F	ES3	F	ES4	F	ES5	F	ES6	F	ES7	F	ES8	F
48			misc	MAT_02	86	7.89E-9		210000.0		0.25		250.0		0.0		0.0		0.0	
					0.0	0.0		5		0		0.0				0.0		0.0	
					0.0	0.0		0.0		0.0		0.0		0.0		0.0		0.0	
					0.0	0.0		0.0		0.0		0.0		0.0		0.0		0.0	

Keyword: M1/MATERIAL

CANCEL RESET_ALL HELP

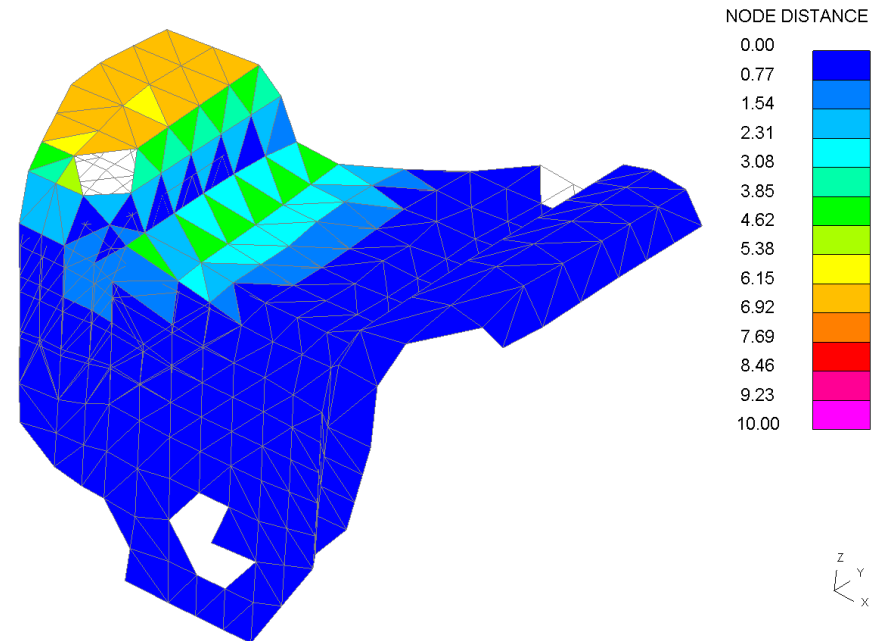
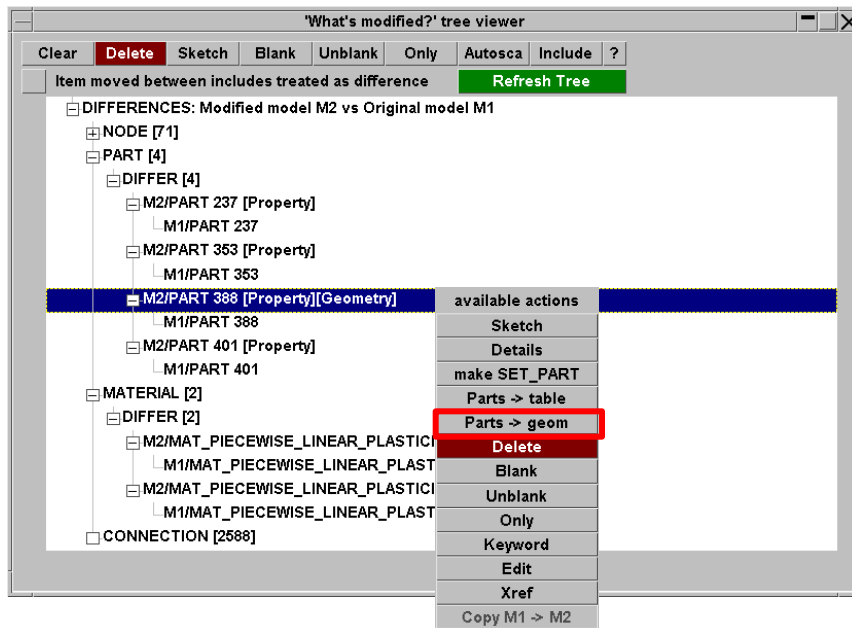
UPDATE CHECK_ALL SKETCH_ALL

Keyword M1 MATERIAL (1/0 mod)

Filter by: MAT <auto> <auto> <auto>

#	Options..	Incl	Surfaces	MID	La	RO	F	E	F	PR	F	SIGY	F	ETAN	F	FAIL	F	TDEL	F
				C	F	P	F	LCSS	F	LCSR	F	VP	F						
				EPS1	F	EPS2	F	EPS3	F	EPS4	F	EPS5	F	EPS6	F	EPS7	F	EPS8	F
				ES1	F	ES2	F	ES3	F	ES4	F	ES5	F	ES6	F	ES7	F	ES8	F
48			misc	MAT_02	86	7.89E-9		210000.0		0.3		250.0		0.0		0.0		0.0	
					0.0	0.0		5		0		0.0				0.0		0.0	
					0.0	0.0		0.0		0.0		0.0		0.0		0.0		0.0	
					0.0	0.0		0.0		0.0		0.0		0.0		0.0		0.0	

- Various options are available when right mouse clicking on one of the differences:
 - Geometry differences between parts can also be checked and contoured.



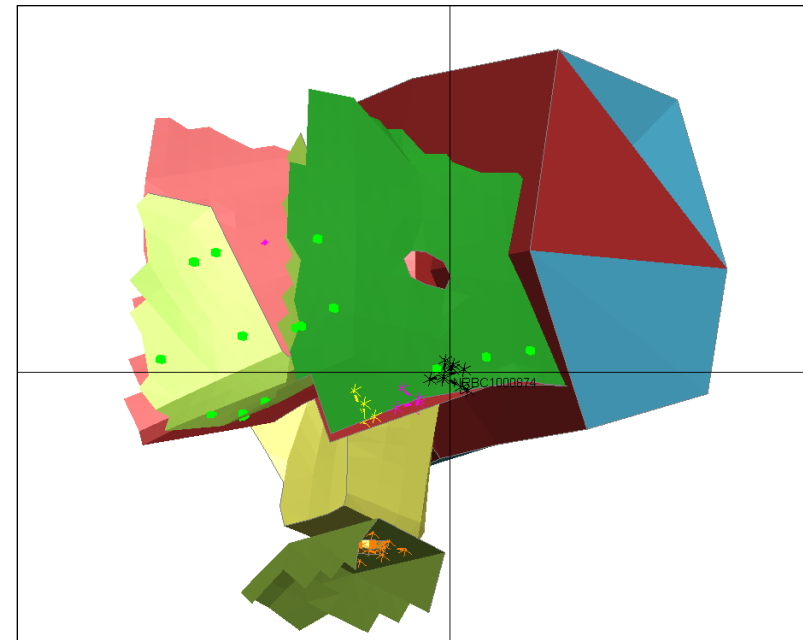
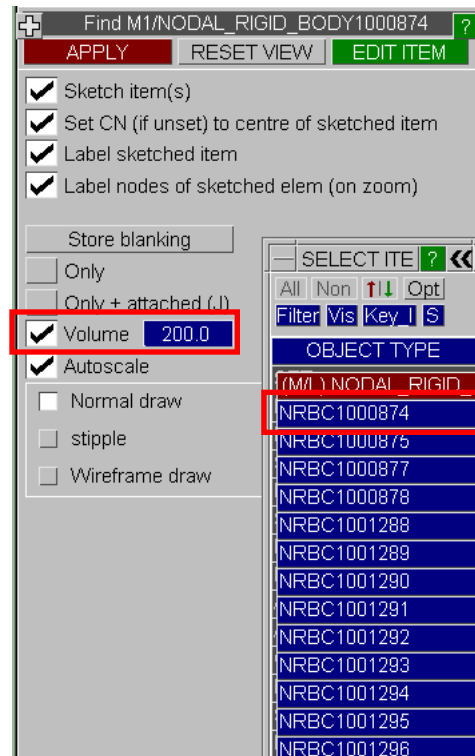
Find

- 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.

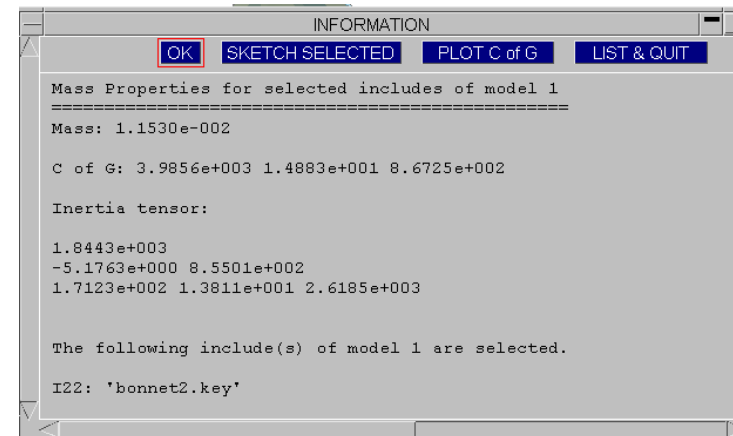
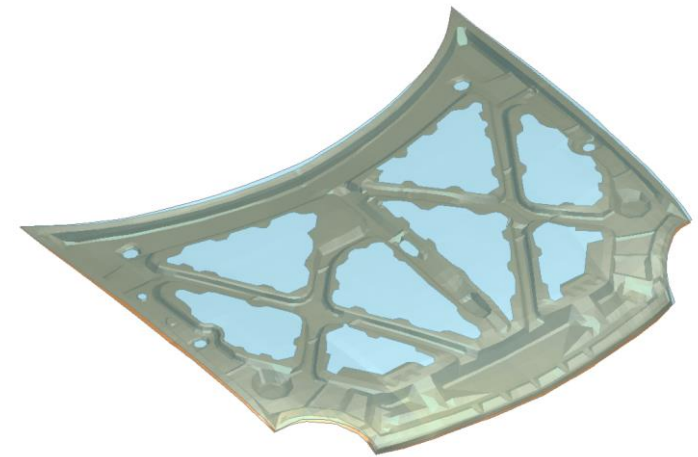
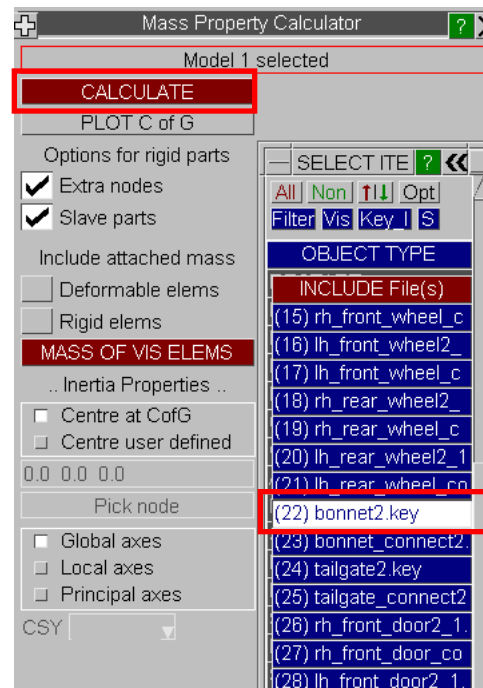


Mass Properties

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

Find

- 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.



Multi-Model Build

- PRIMER can build multiple models, with an impactor in different positions using information from a CSV file (e.g. pedestrian impact).

Multi-Model Build

- Delete all models.
- In the Model menu go to **Build**, and change the method to **Build from CSV targeting file**. Click **Apply**.
- Click **Read CSV** to read in "postion.csv".
- Click **Make** to read the models into PRIMER.
- The model contains a simple bonnet and a headform in a reference position.
- The panel contains options for impactor orientation and depenetration.

Model build from csv file

Read CSV Write CSV Apply

Diffcheck Delete write includes

Interactive model build

Build: PEDHEAD

Model: Jan-14\simple_bonnet.key Pick

Impactor: Jan-14\simple_head.key Pick

Make

☒ Orient N1N3 is normal to impactor XZ plane

☐ node Name1/id1 Name2/id2 Name3/id3

☐ csys 1536 1802 999

☒ Depenetrate

Type: Contact contact name/id

Method: XZ head to bonnet

☐ Vertical

Chin set name/id <none>

Method: X

Z (for PEDLEG_LOWER) 0.0

☐ Offset (for deployable bonnets)

Distance 0.0

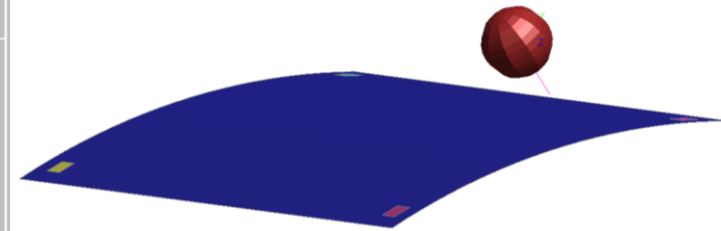
Output dir primer_Intro_Course_Jan-4

Output name

Reporter individu

Reporter summa

Edit loadcases



Model build from csv file

Return to main panel

Loadcases:

Dir	Zone	X	Y	Sketch all		
		0.0	0.0	Sketch	Pick	+
C1	C1	850.0	1350.0	Sketch	Pick	✗
C2	C2	850.0	750.0	Sketch	Pick	✗
C3	C3	850.0	150.0	Sketch	Pick	✗
C4	C4	450.0	1350.0	Sketch	Pick	✗
C5	C5	450.0	750.0	Sketch	Pick	✗
C6	C6	450.0	150.0	Sketch	Pick	✗
				Sketch	Pick	✗

You can view/edit/input target points in the loadcase panel

- PRIMER can build multiple models, with an impactor in different positions using information from a CSV file (e.g. pedestrian impact).

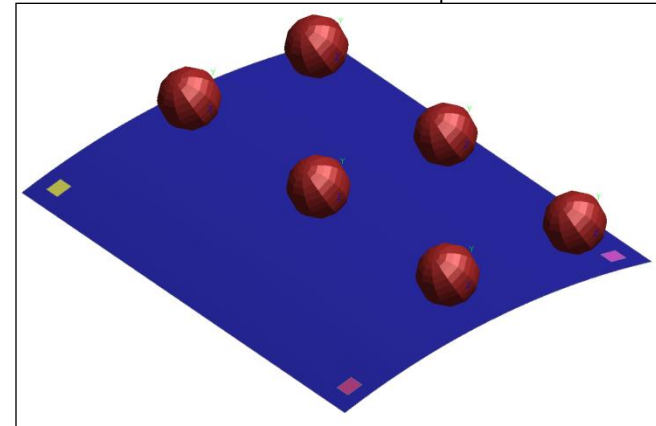
Multi-Model Build

1. Un-toggle (turn off) the **Delete** switch at the top.
2. Click **Apply**.
3. Primer will automatically position and depenetrate the impactor at the specified target locations. 6 new models will exist in Primer, and master files containing ***INCLUDE_TRANSFORM**'s and references to the includes will be created (directories C1-C6 have been created).

```

*KEYWORD

$ =====
$ DEFINE_TRANSFORMATION cards
$ =====
$
*DEFINE_TRANSFORMATION
  1
TRANSL      487.78326      1350.0-111.62807
$
$ =====
$ INCLUDE cards
$ =====
$
*INCLUDE
  ..\simple_bonnet.key
$
*INCLUDE_TRANSFORM
  ..\simple_head.key
      0          0          0          0          0          0          0
      0
      0.0        0.0        0.0          0
      1
$
$
*TITLE
C1
$
$
*END
    
```



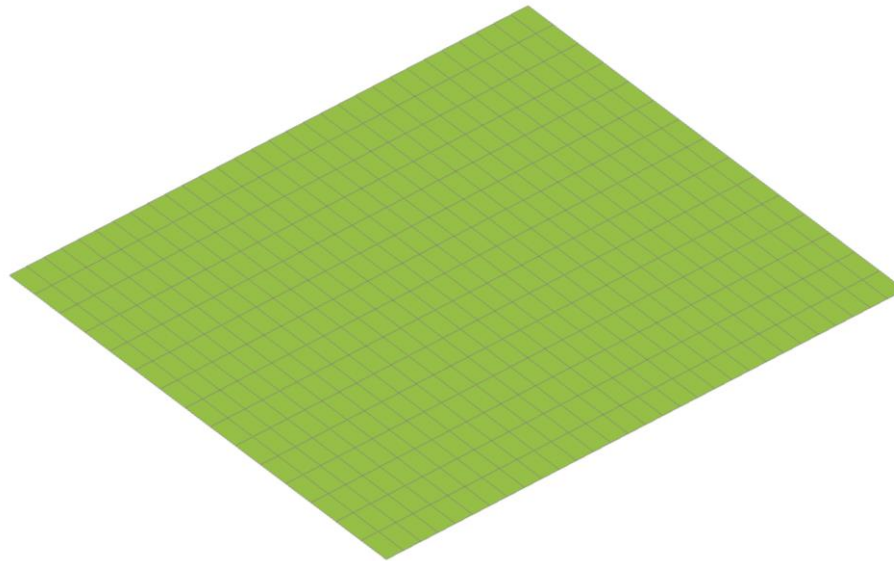
Scripting

- 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

- 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 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
```

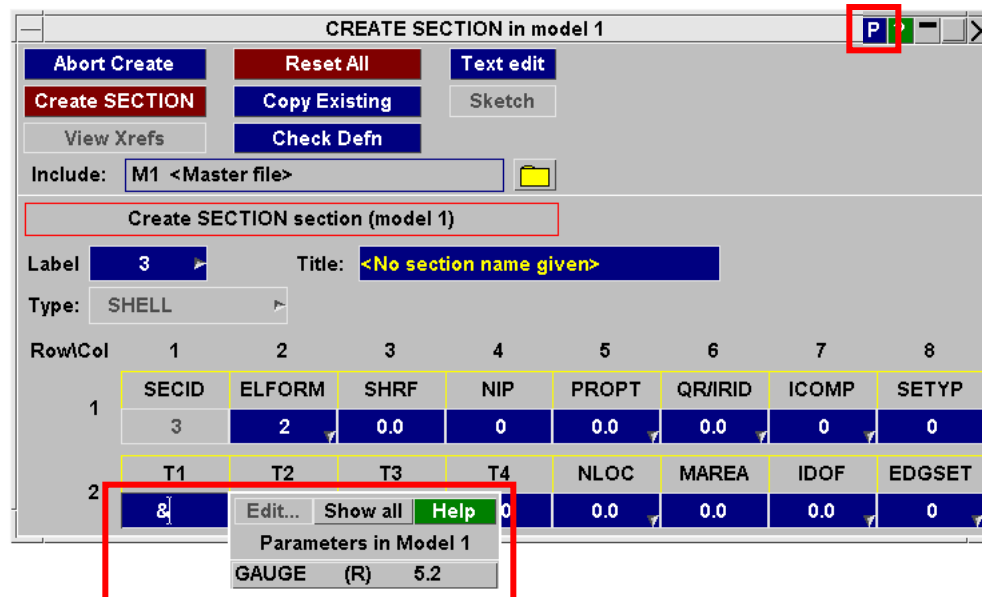
```
*SECTION_SHELL
```

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



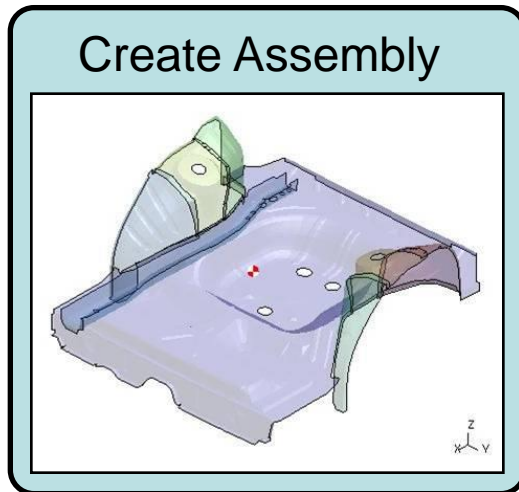
- *PARAMETER is supported in PRIMER.
- Type “&” in the input boxes and a popup will allow you to select/edit/create a *PARAMETER.



“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

- Massing up an assembly:



Input Required Mass

CREATE ASSIGN MASS in model 1

ABORT_CREATE RESET_ALL HELP

CREATE_ASSM COPY_EXISTING SKETCH

PLOT_MASS CHECK_DEFN CALCULATE

Create ASSIGN MASS (model 1)

Label: 1 Title:

☐ Group menu Group ID: 1 SKETCH

☐ Part menu Title:

ADD MASS TO GROUP of 0.0300 (total mass 0.0545) SHOW CG

Target CoG: X: 1200.0 Y: 0.0 Z: 558.0

Reset mass & CoG Set mass only Incl attached mass start id: 1

☐ Change mass and CoG by changing entire group end id: 99999999

☐ Change mass and CoG by changing a subset of the group OVERMA ?

Subgroup ID: N/A SKETCH

Title:

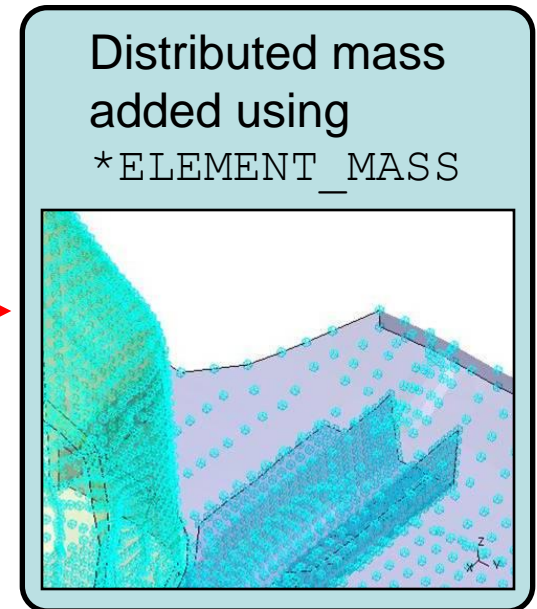
Original mass and properties of group

Actual mass:	0.0245	SHOW CG
Actual CoG: X:	1189.1	Y: -83.30 Z: 557.8
Inertia tensor: IXX:	5335.6	IXY: 102.0 IXZ: 717.3
	IVY: 4418.5	IVZ: -166.1
		IZZ: 8653.4

Included mass from Part Inertias <none> SKETCH

Included mass from NRB Inertias <none> SKETCH

Excluded Part Inertia & NRB elements: <none> SKETCH



- 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

BILL OF MATERIALS

CANCEL < PREV APPLY HELP

Define the fields in the file

Field type

PID

Skip field

CAD part no.

Part description

Material title

Material ID

Section ID

Hourglass ID

Gauge

Hourglass type

Hourglass coeff

Element form

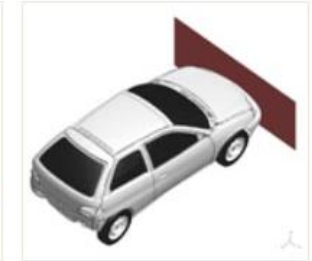
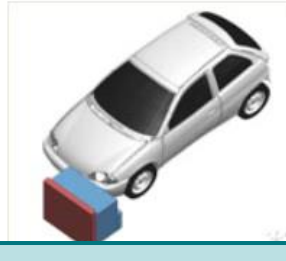
No. int pts

Lower id

Upper id

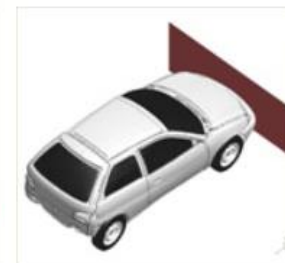
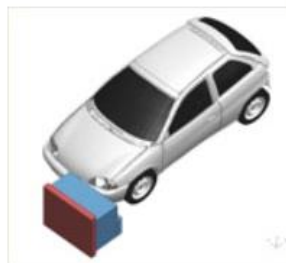
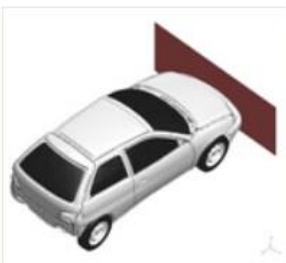
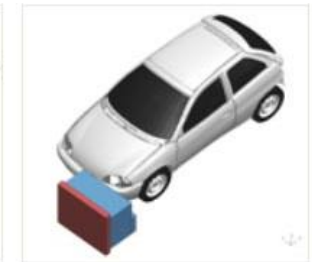
Select fields in Bill of Materials

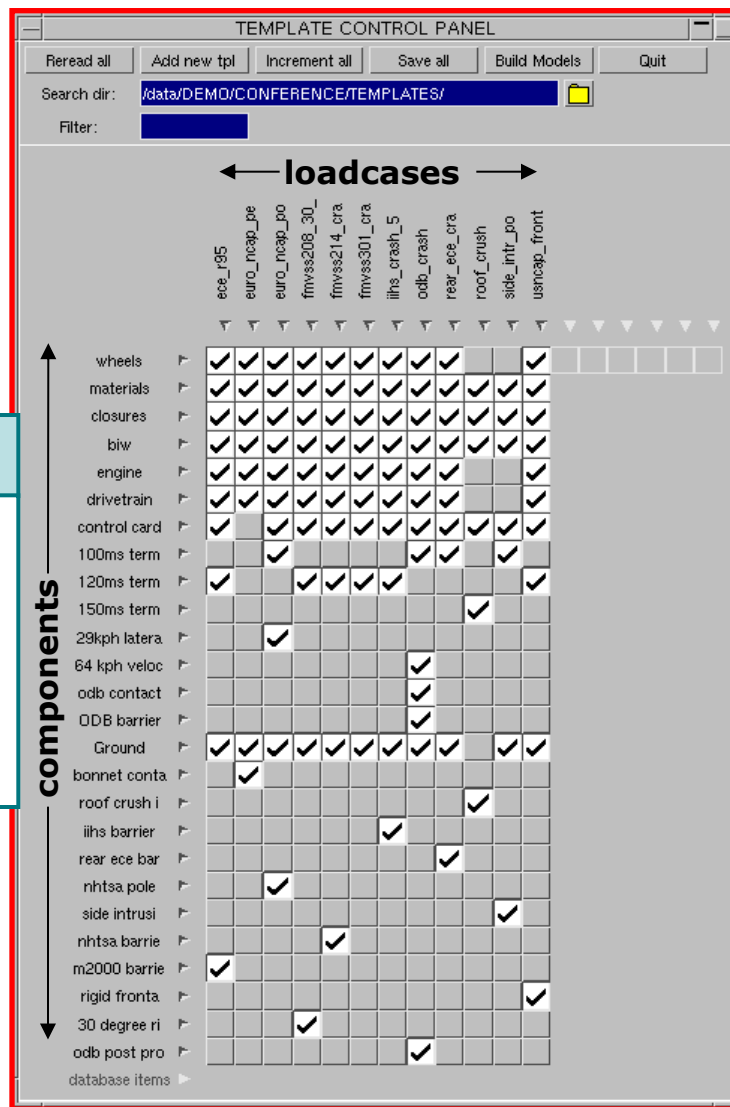
	A	B		D	E	F	G	H	I
Field	PID	CAD part no.		Material ID	Section ID	Hourglass ID	Material title	Gauge	Hourglass type
1	\$ Primer Bill								
2	\$ =====								
3	\$								
4	\$ Created on:								
5	\$								
6	\$ from model:								
7	\$								
8	Part ID	CAD Part No		Material ID	Section ID	Hourglass ID	Material	Gauge	HG type
9	\$								
10	281			129	129			1.076	



Automatic Model Assembly

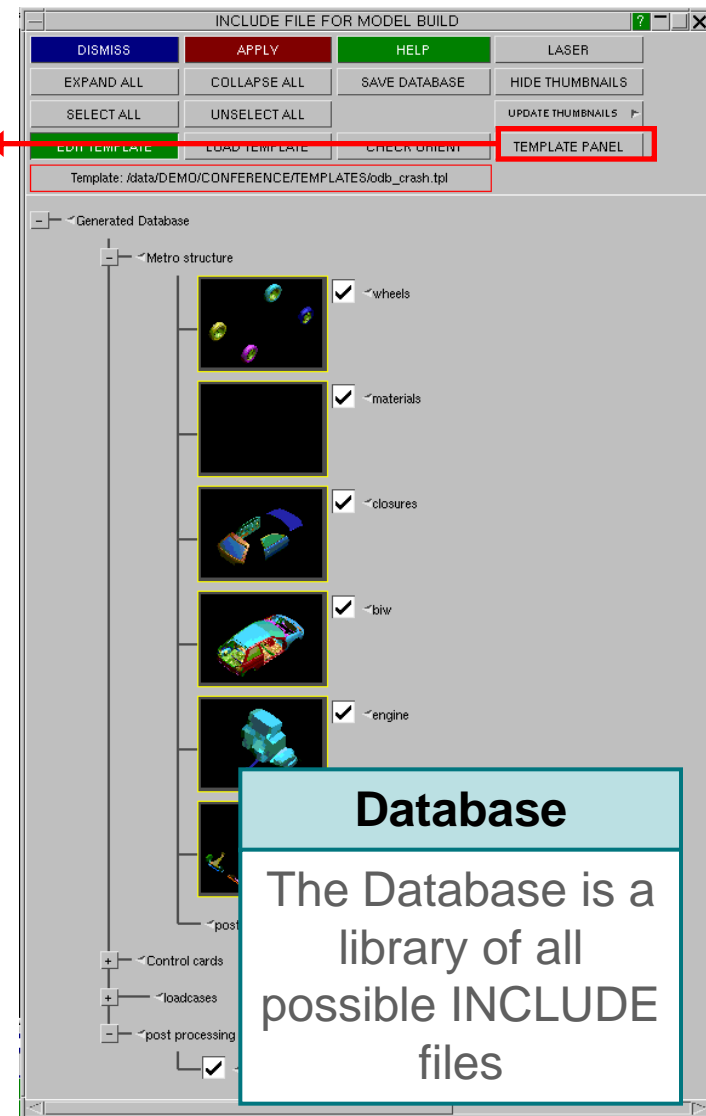
PRIMER can assemble models from a Database of INCLUDE files. For each loadcase or model configuration, a template defines which INCLUDE files are selected.





Template Panel

The Templates show which INCLUDE files are needed for each loadcase

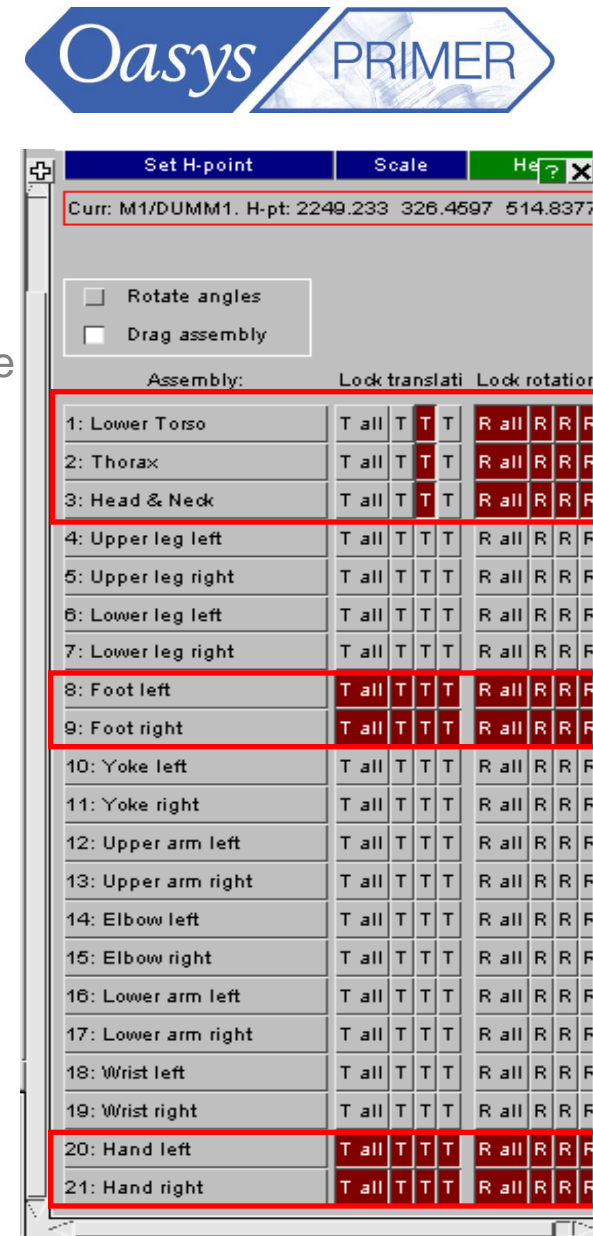
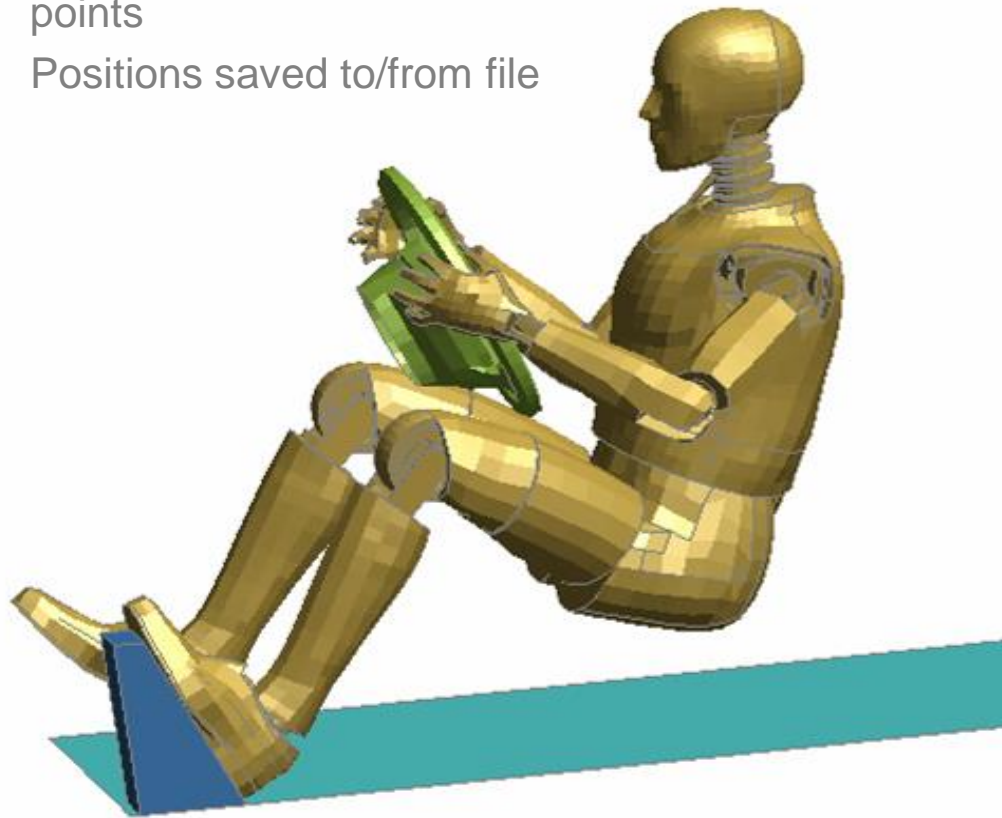


Database

The Database is a library of all possible INCLUDE files

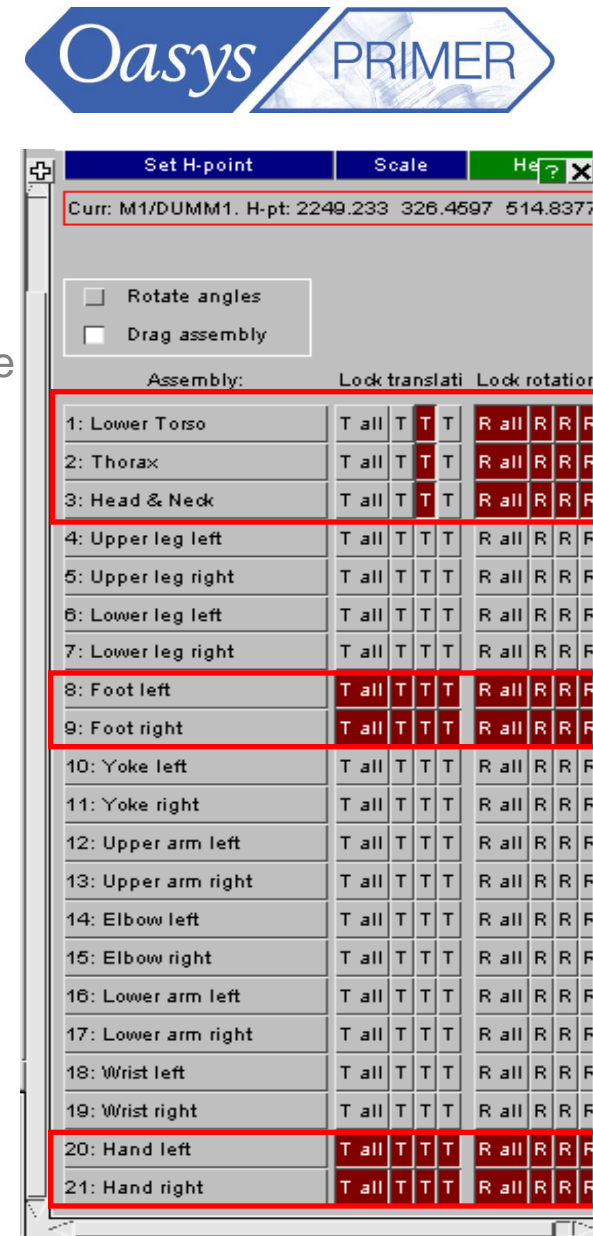
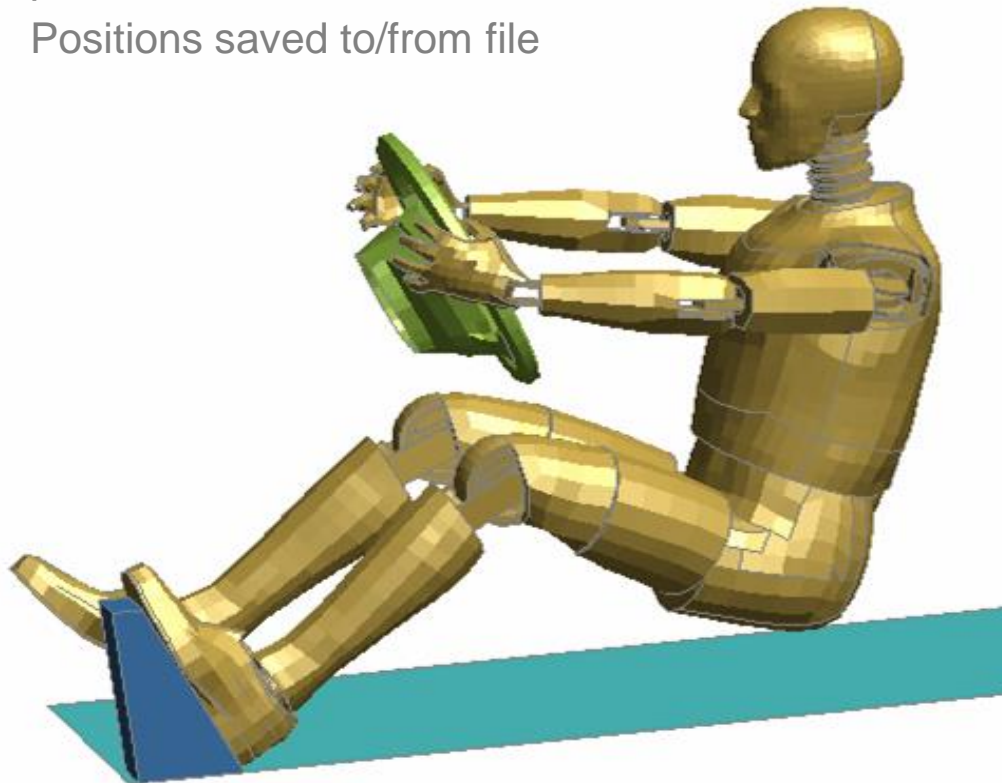
Dummy positioning

- 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



Dummy positioning

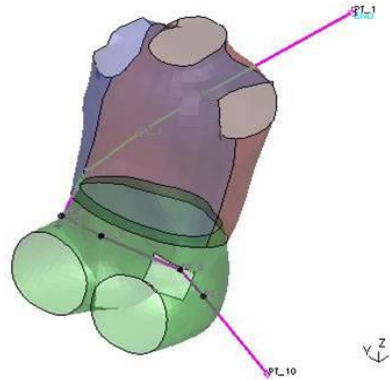
- Dummy can be positioned by
 - Rotating joints
 - Dragging parts, restraining other parts
 - Typing desired coordinates for H-point or other reference points
 - Positions saved to/from file



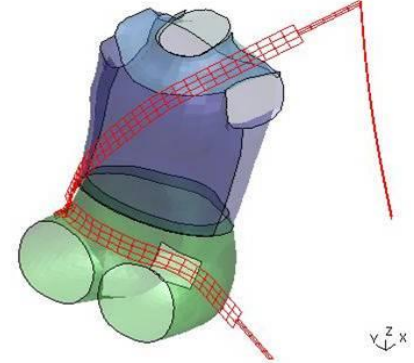
1. Define belt



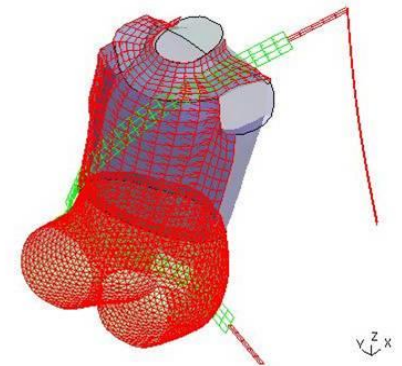
2. Define path



3. Mesh belt

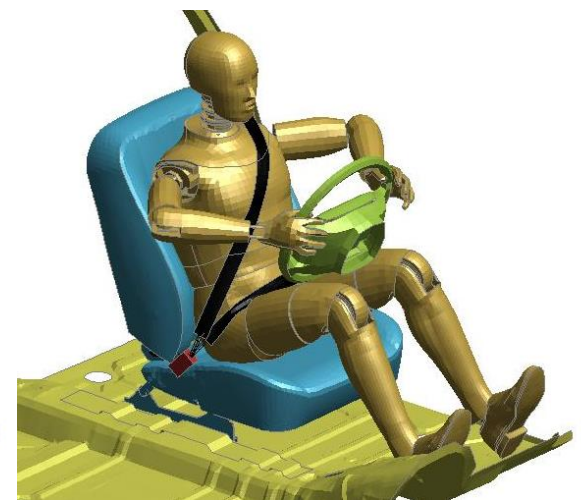
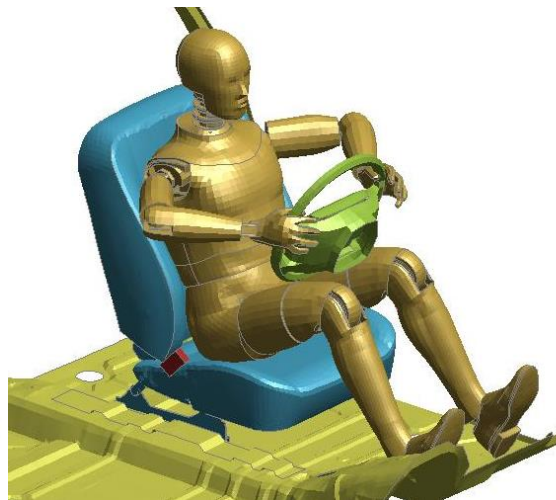


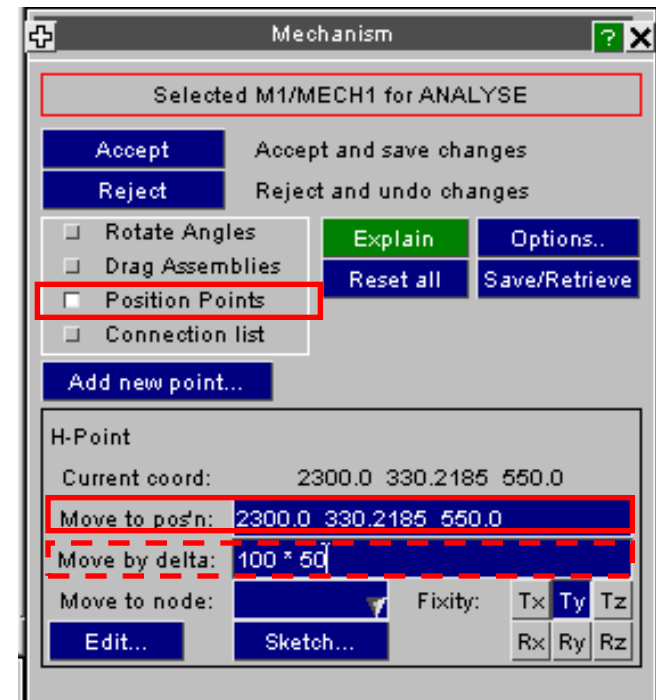
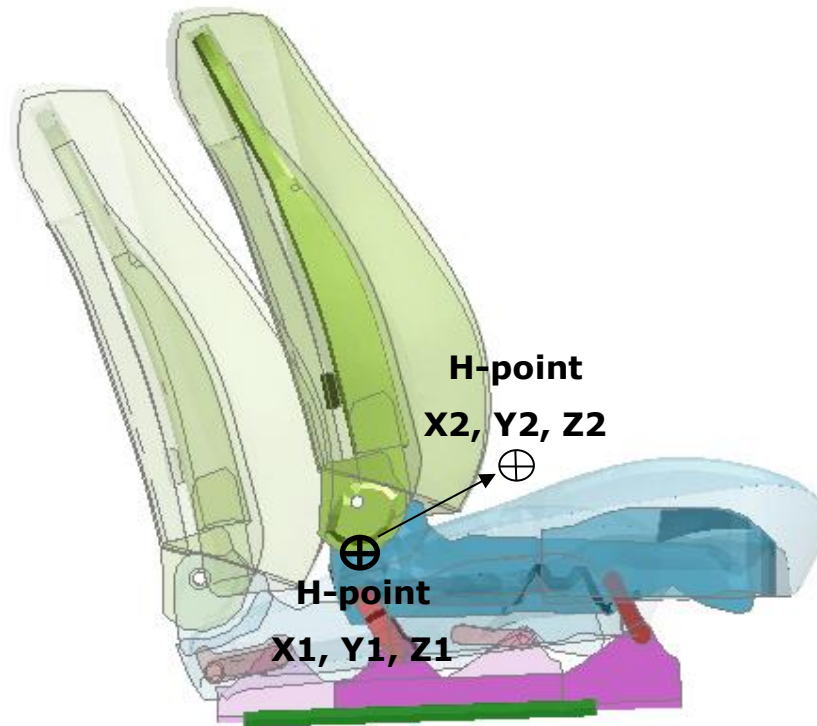
4. Setup contact



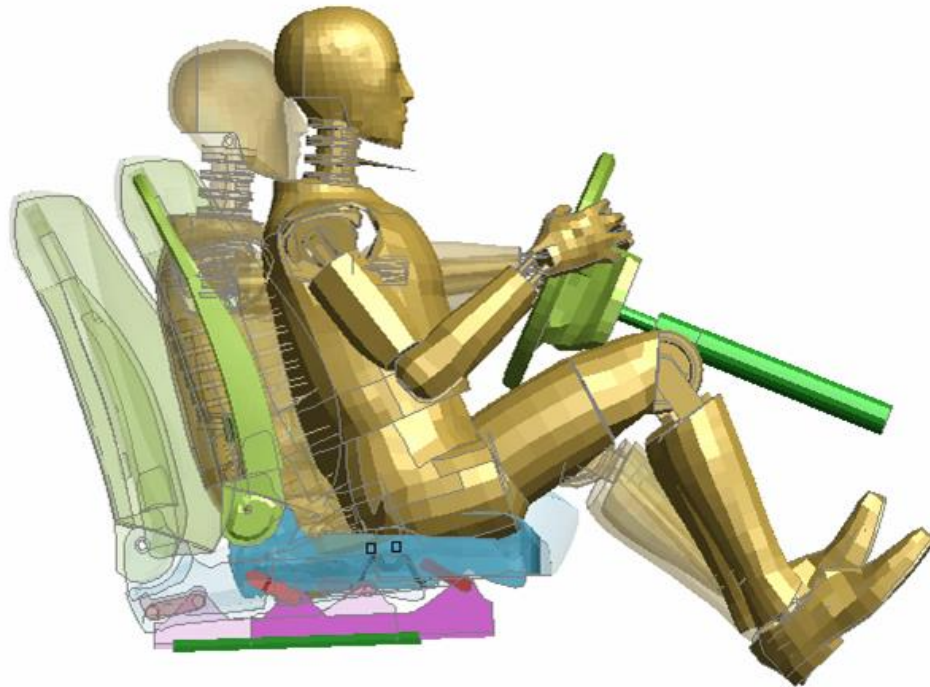
- Belt creation in 4 steps
- Once created slipping, pretensioner, retractor can be added from belt menu

- 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

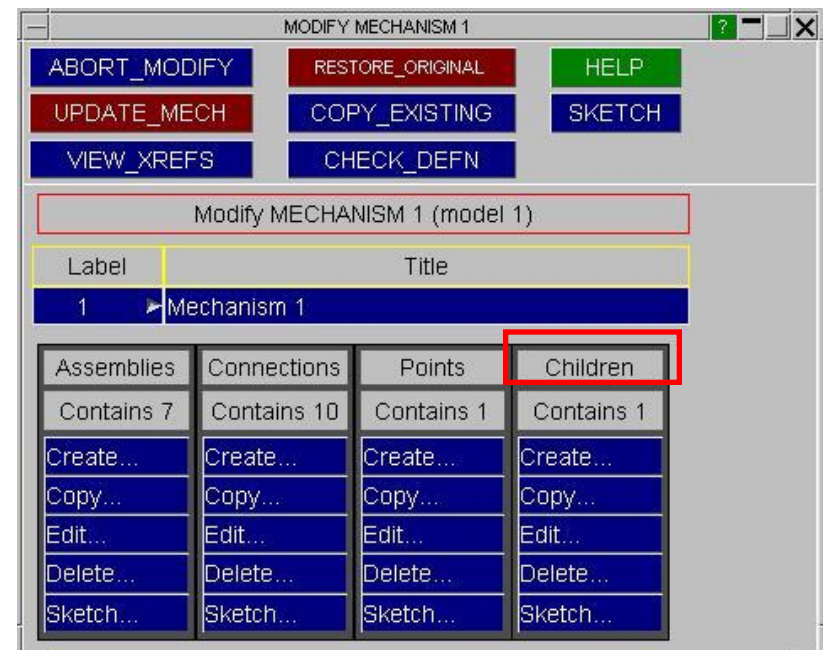


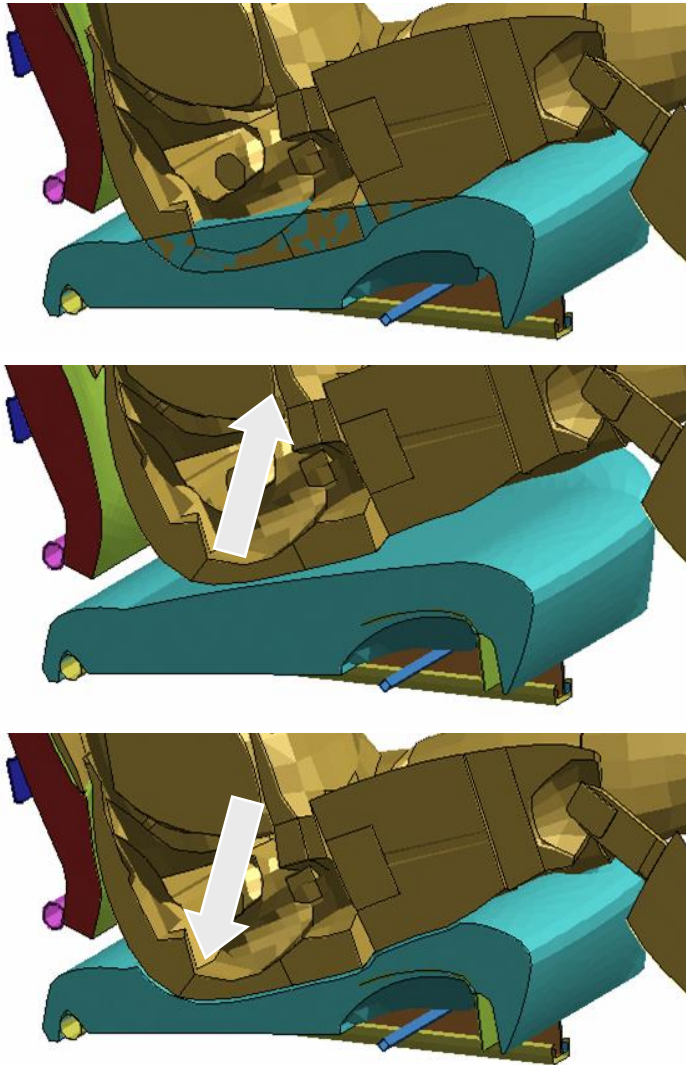


- A mechanism (for example, a seat) can be defined within PRIMER. The mechanism consists of Assemblies (groups of parts) and joints (pin, hinge or sliding).
- The mechanism can be positioned by dragging, or by typing in the desired coordinates of “reference points”



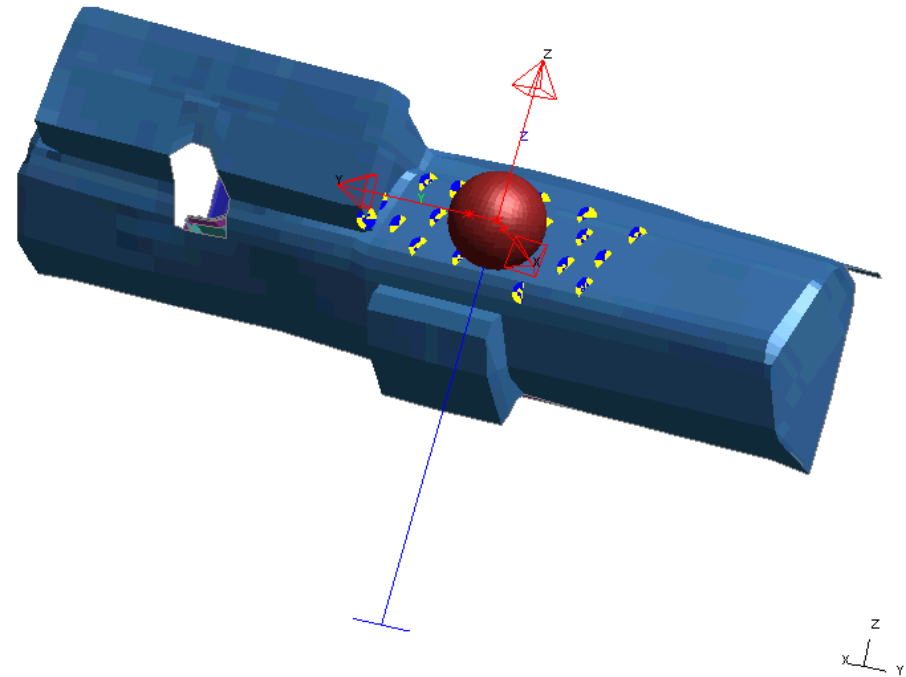
- One mechanism may be linked to another, e.g. seat squab to dummy pelvis. Then the dummy and seat can be dragged together in a single action.



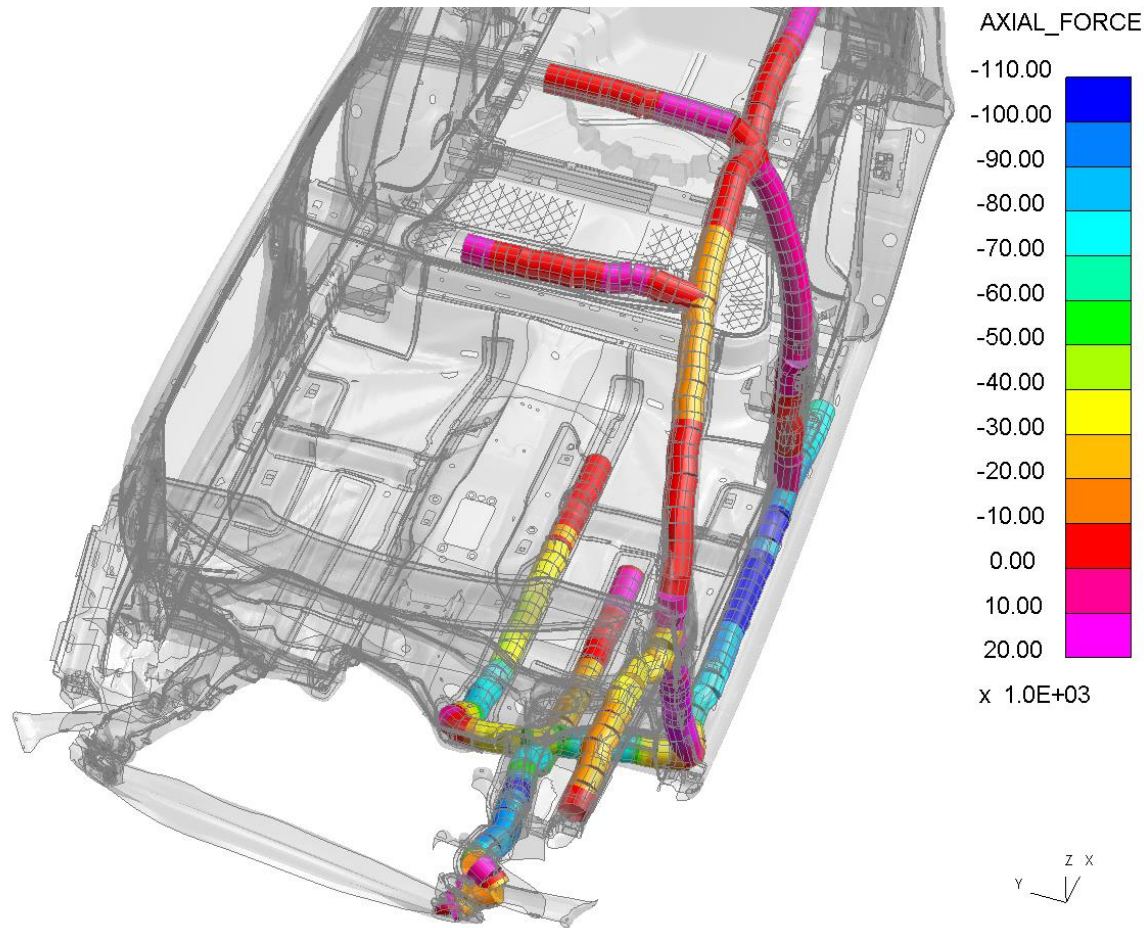


- Dummy has been positioned, penetrating the foam.
- “PRIMER” method depenetrates the top surface of the foam, and morphs the interior foam nodes to spread the strain evenly.
- “Dyna” method helps the user to set up an LS-DYNA analysis in which a rigid dummy compresses the seat foam until the correct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

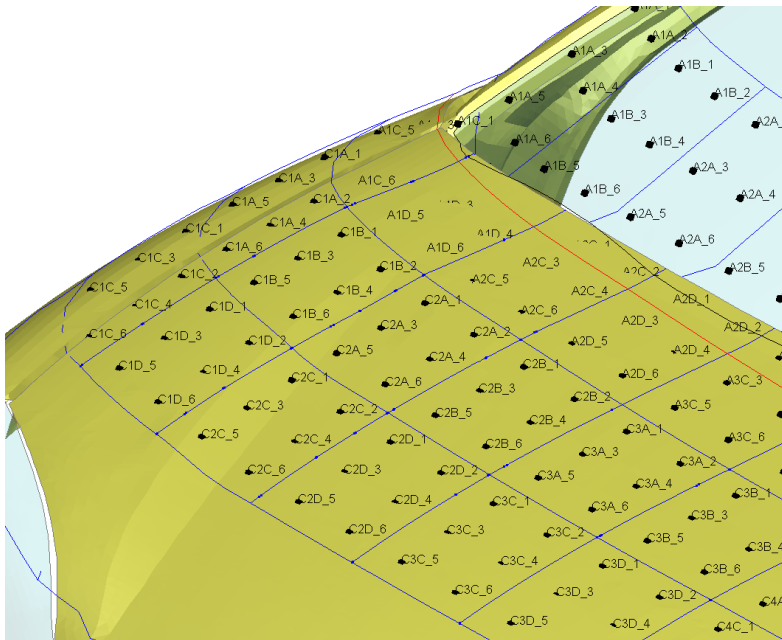
- Setup of models for ECER21.
- Both interactive and batch mode supported.
- Easy creation/modification of target points.
- Automated positioning and de-penetration.
- True and realigned approach angle supported.
- Data saved in model or written to csv target file.
- Multiple models (for multiple impact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



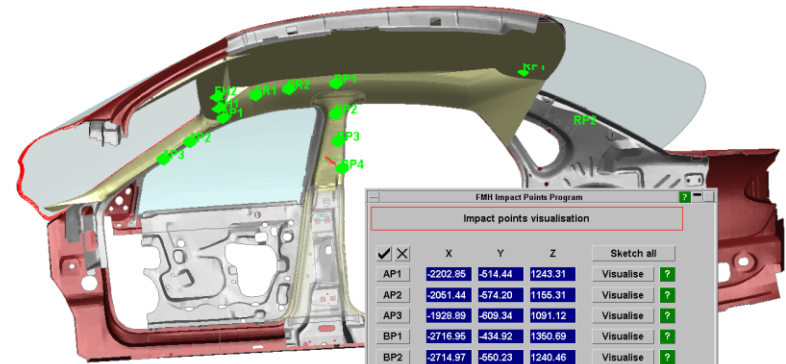
*DATABASE_CROSS_SECTION's can be grouped together in PRIMER to create load paths. These can be viewed and contoured within D3PLOT.



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

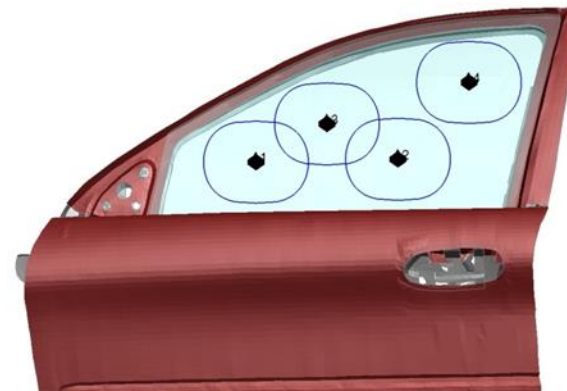


Pedestrian head/leg markup



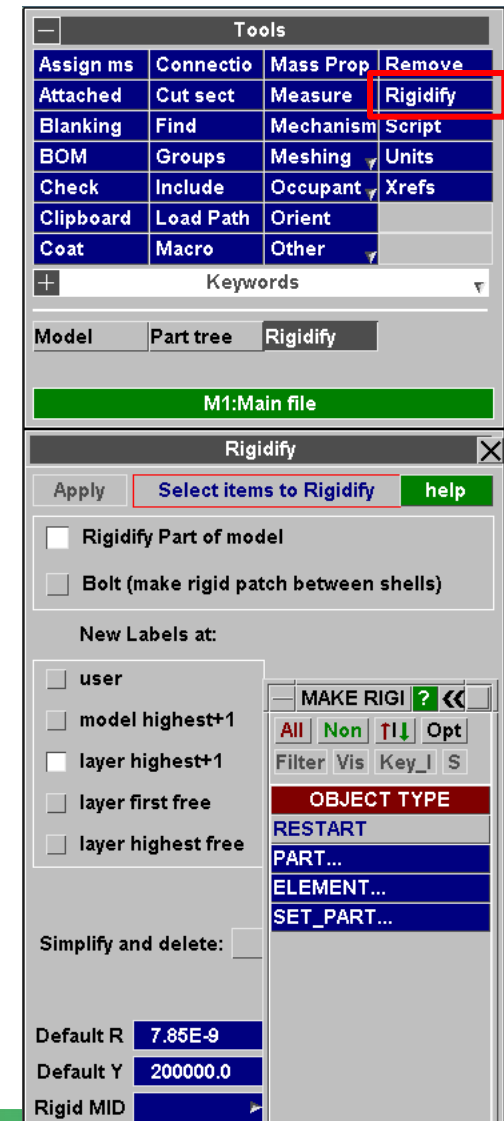
FMVSS201u
(FMH) markup

Impact points visualisation					
	X	Y	Z	Sketch all	
AP1	-2202.86	-514.44	1243.31	Visualise	?
AP2	-2051.44	-574.20	1155.31	Visualise	?
AP3	-1928.89	-609.34	1081.12	Visualise	?
BP1	-2716.96	-434.92	1350.69	Visualise	?
BP2	-2714.97	-560.23	1240.46	Visualise	?
BP3	-2722.68	-616.84	1131.63	Visualise	?
BP4	-2737.86	-683.26	1022.09	Visualise	?
RP1	-3663.38	-378.56	1346.17	Visualise	?
RP2	-3801.97	-687.80	1167.38	Visualise	?
FH1	-2175.14	-408.48	1261.81	Visualise	?
FH2	-2167.33	-270.39	1277.80	Visualise	?
SR1	-2352.86	-460.14	1321.60	Visualise	?
SR2	-2503.96	-443.43	1338.77	Visualise	?

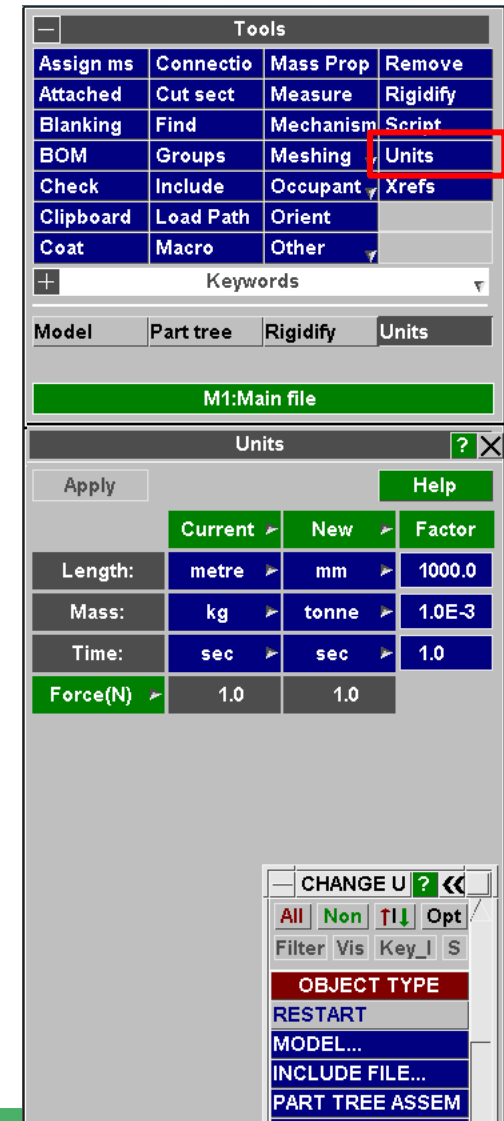


FMVSS226
(ejection
mitigation)
markup

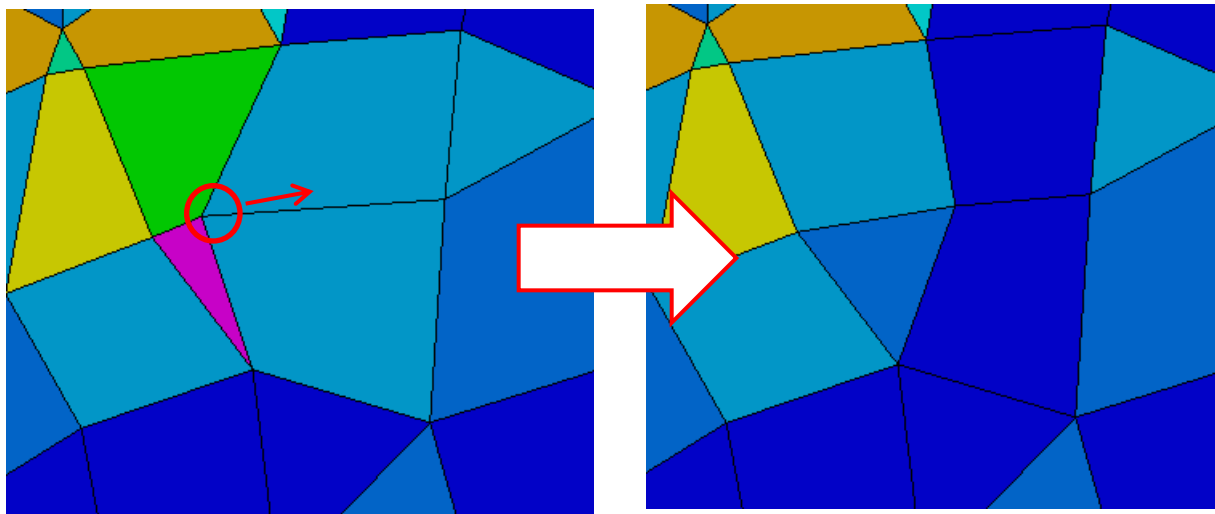
- Rigidify is used to rigidify selected parts of the model.
- Example uses include replacing complex deformable meshes with rigidified representations to reduce runtime.
- Rigid constraint clashes, tied contacts etc. are all taken care of during the rigidification process.



- 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.

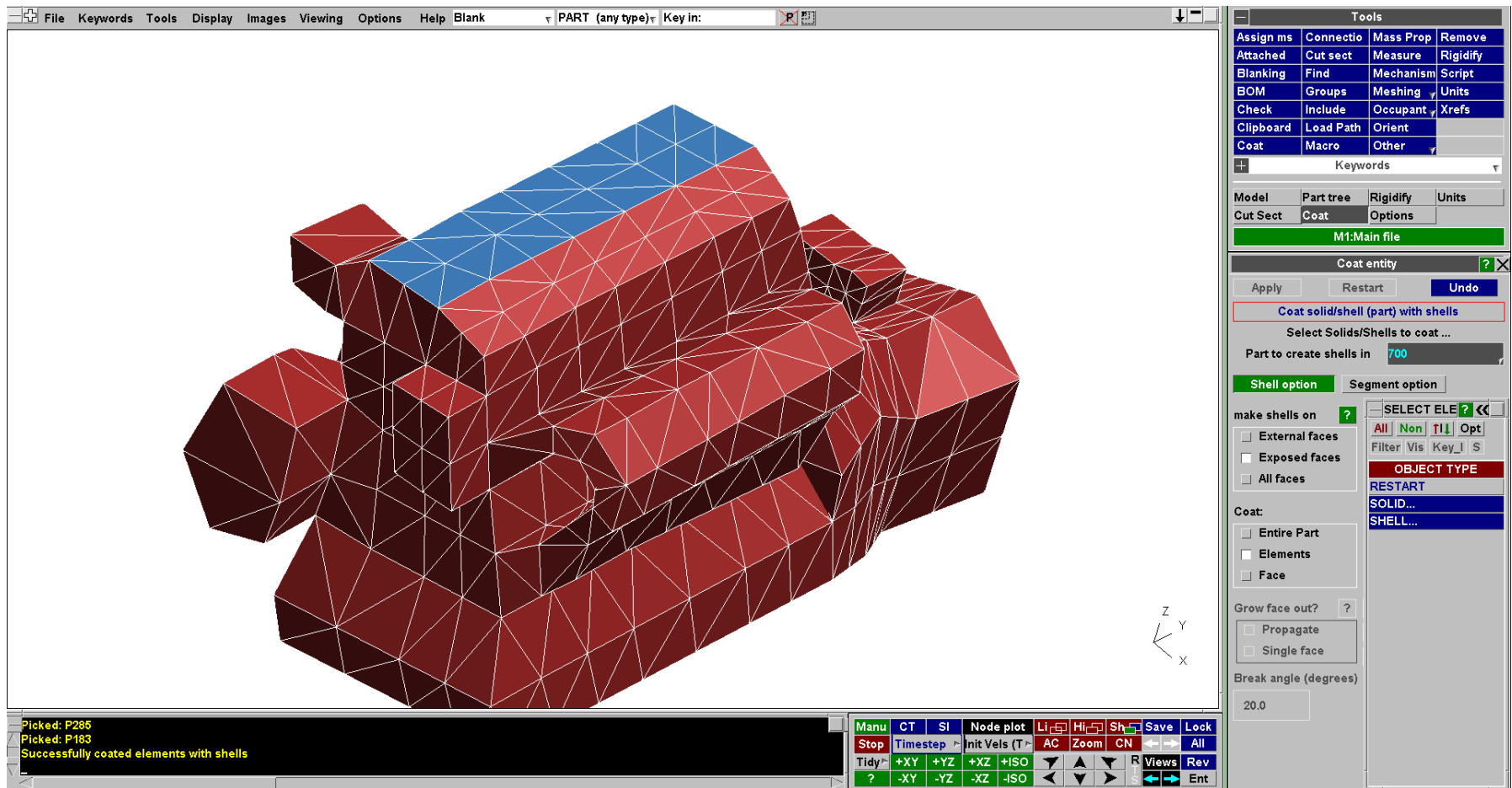


- Various basic meshing tools are available in PRIMER:
 - Mesh surface drag tool to improve mesh quality

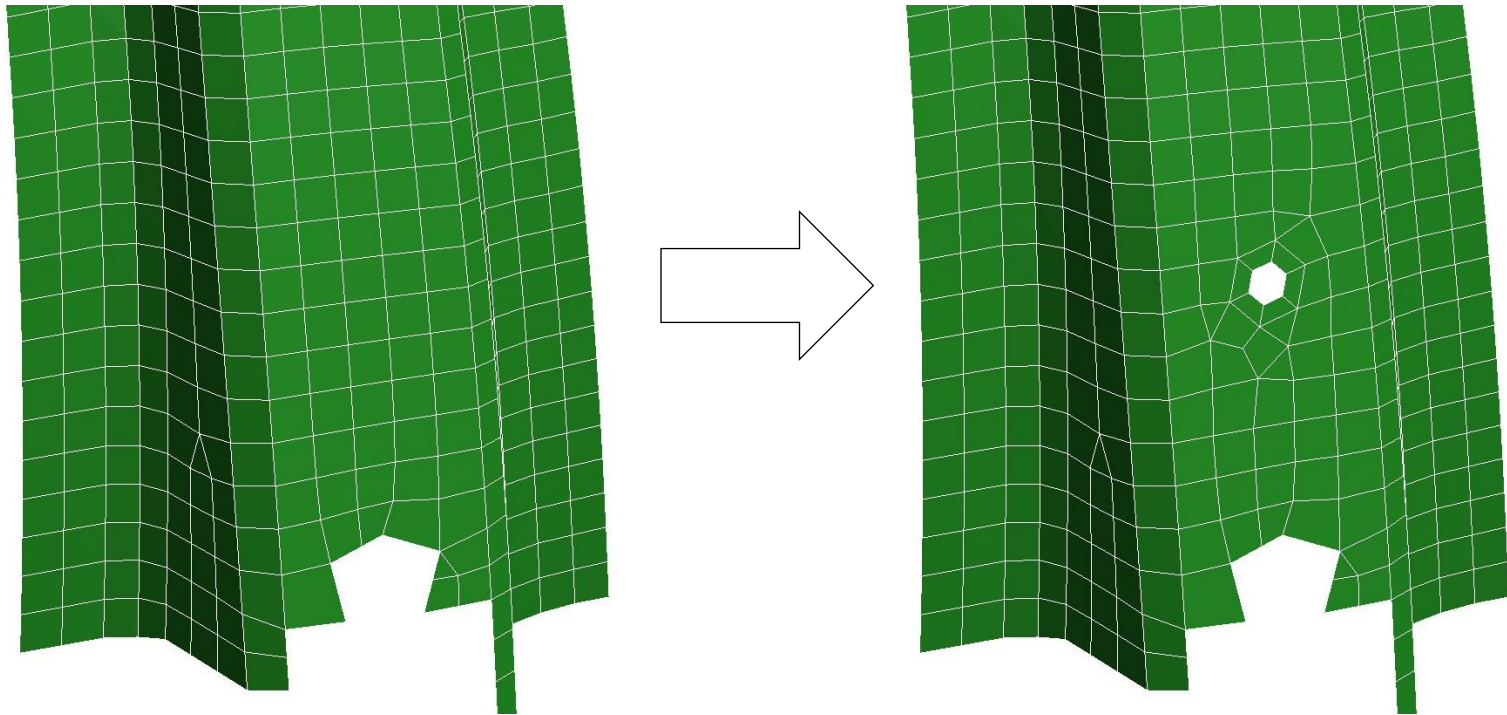


- Element mesh creation:
 - Standard shapes:
 - Plate
 - Cylinder
 - Block
 - Sphere/Hemisphere
 - Standard creation methods
 - Area
 - Extrude
 - Offset
 - Ruled
 - Standard modification tools:
 - Split
 - Combine
 - Align normals

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

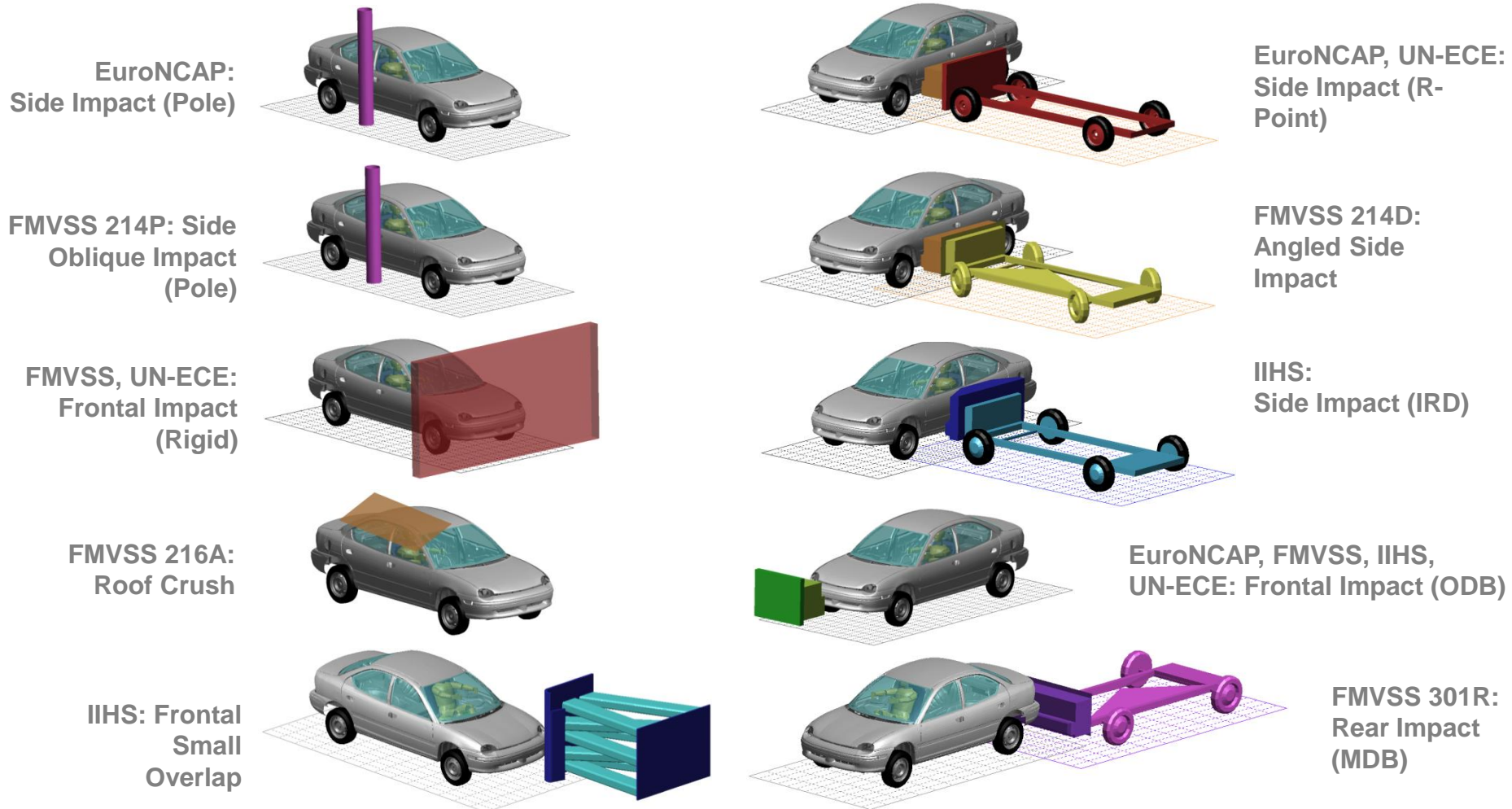


- Create/remove holes in shell mesh:

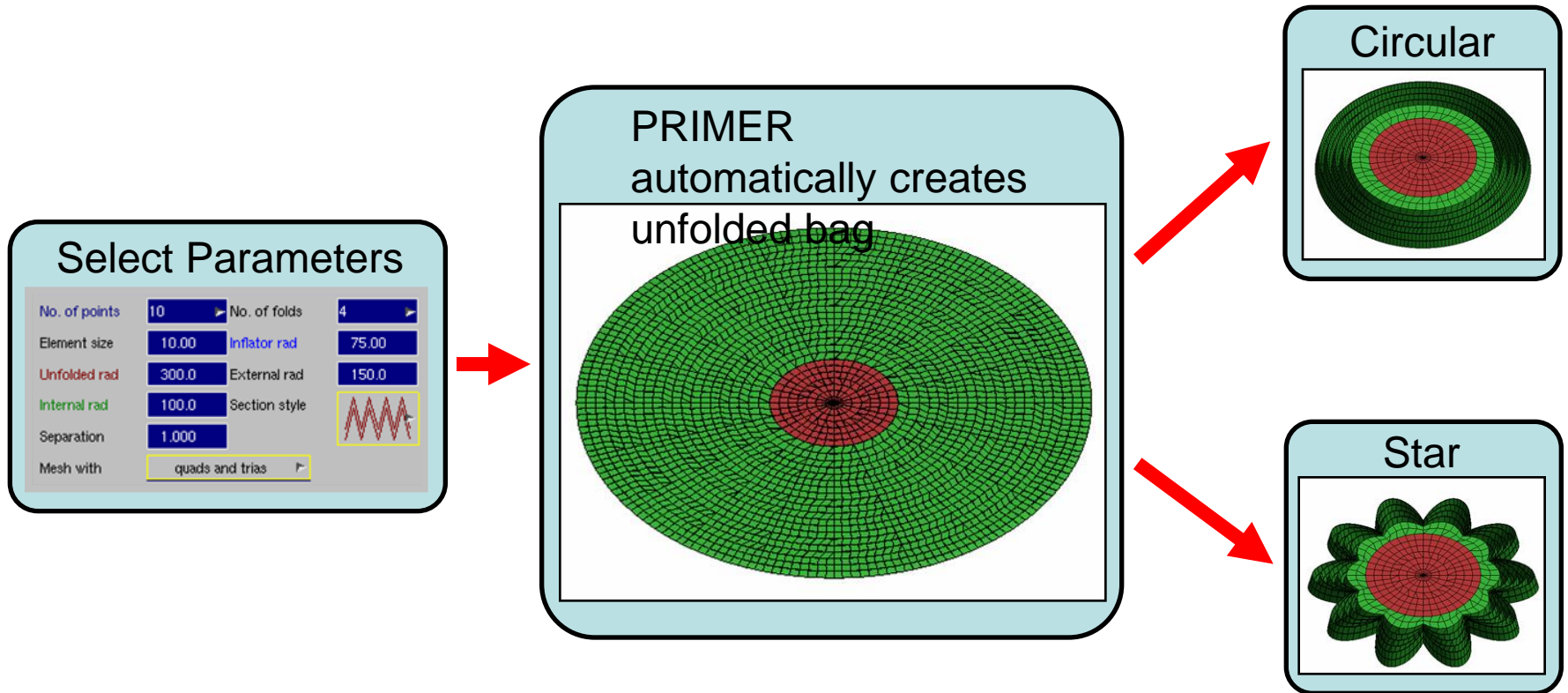


Barrier positioning tool

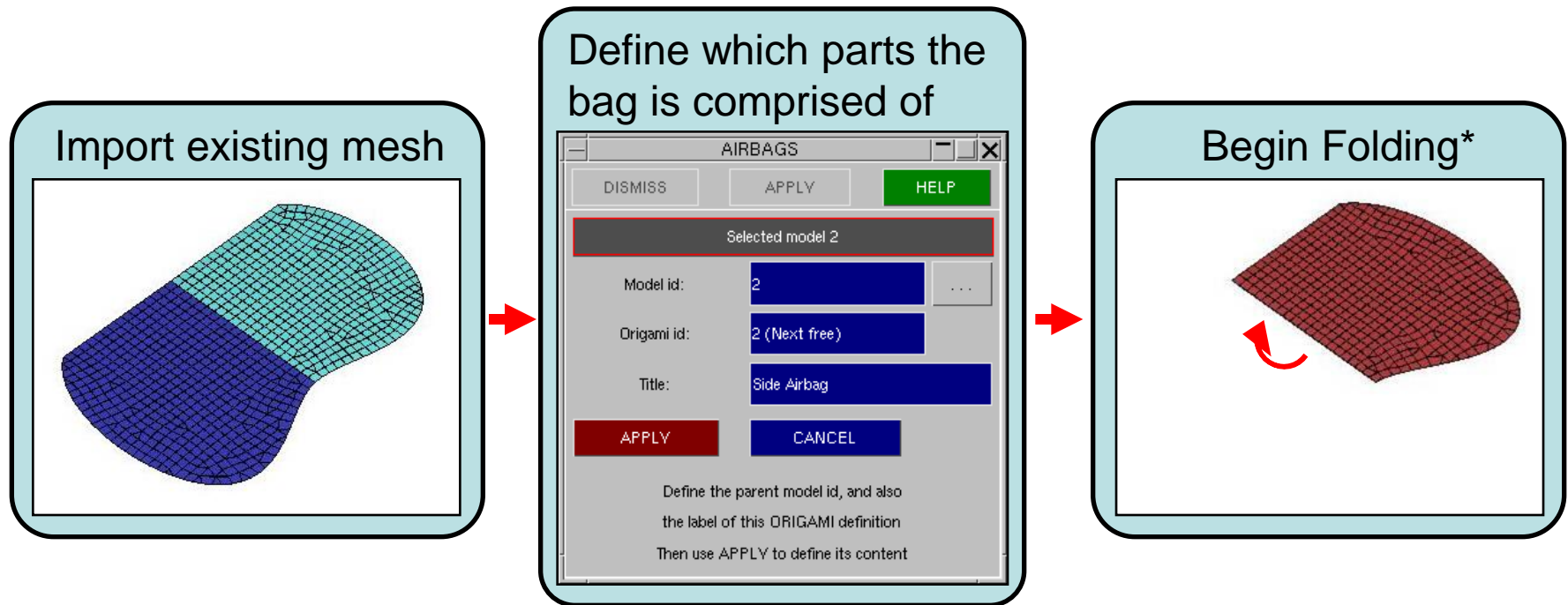
The supported crash test protocols are listed here. Small changes to these crash types (e.g. impact velocity) can be made by the user and saved easily:



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

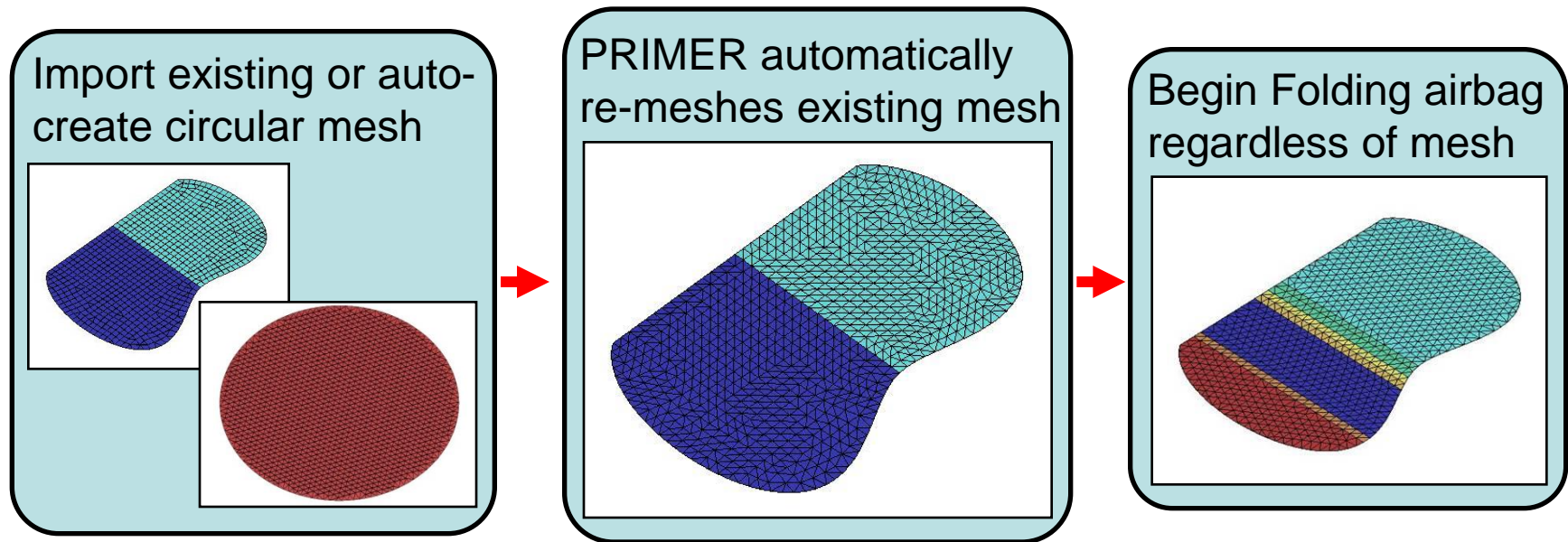


2. Fold an existing mesh using PRIMER's step-by-step functions



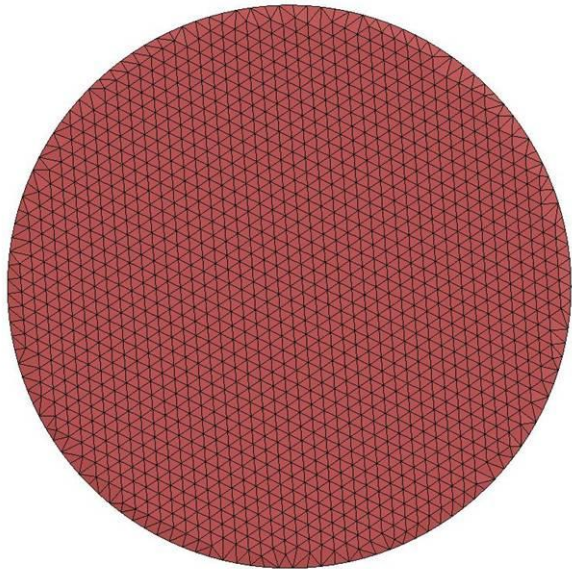
*If an existing mesh is used, the fold lines must be already defined

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

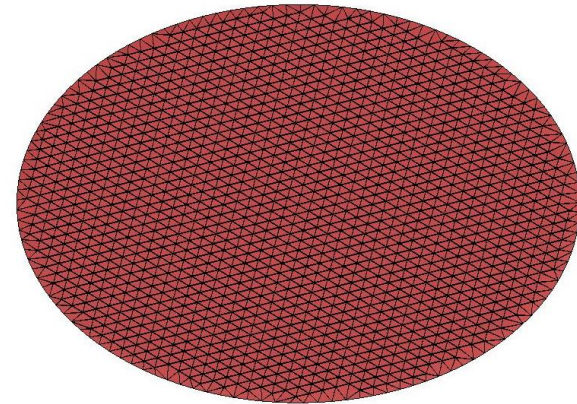


Mesh-free folding allows any airbag shape to be folded in any pattern. PRIMER takes care of creating fold lines and aligning the mesh

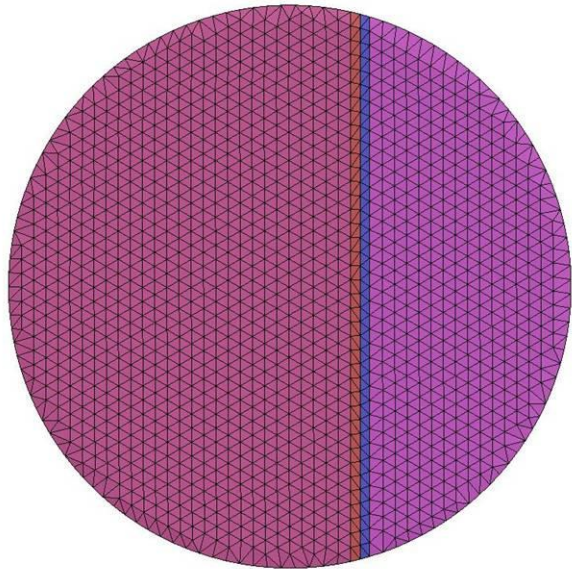
Unfolded View



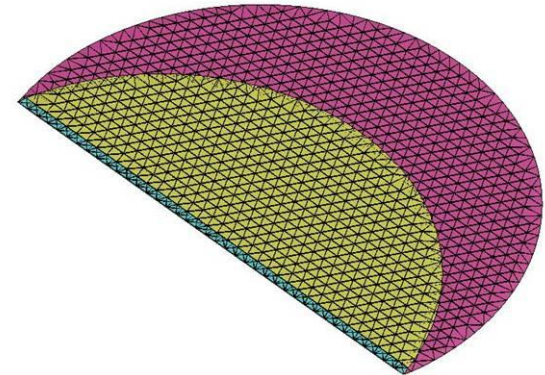
Unfolded Bag



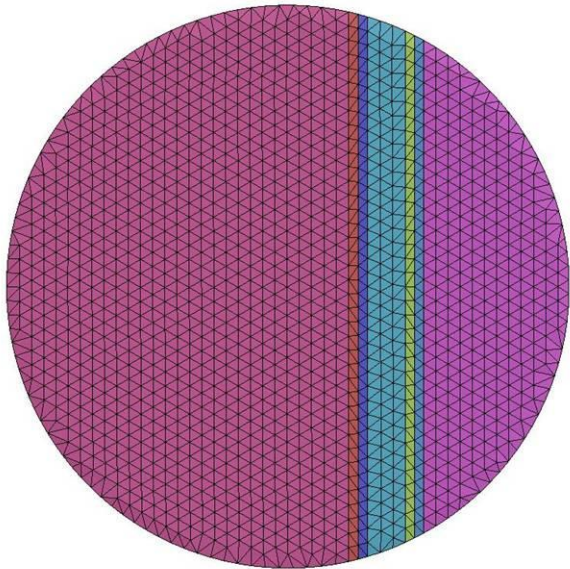
Unfolded View



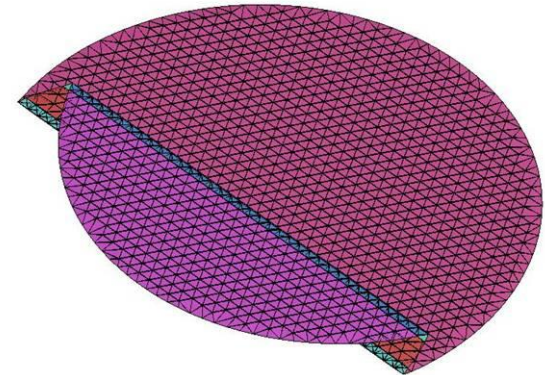
1st Fold: Forward



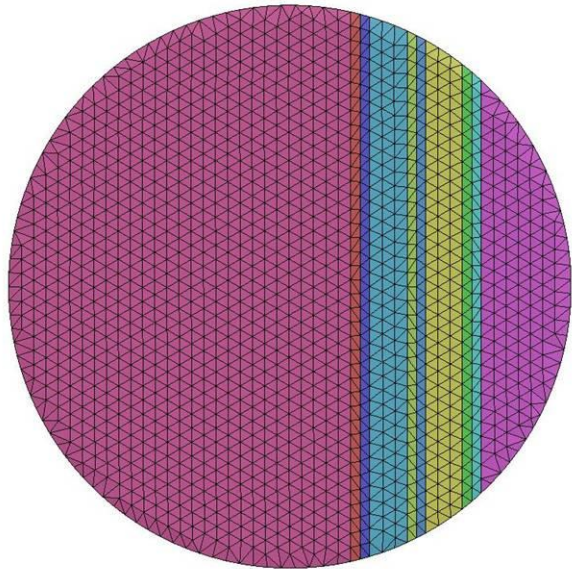
Unfolded View



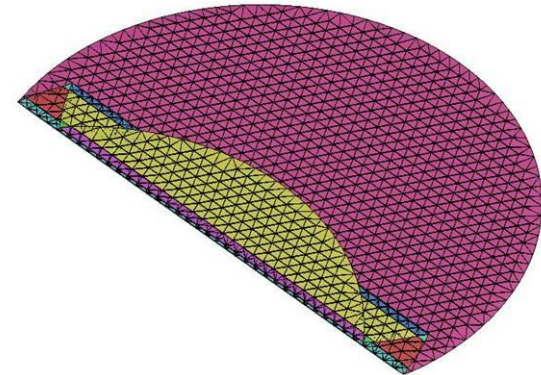
2nd Fold: Reverse



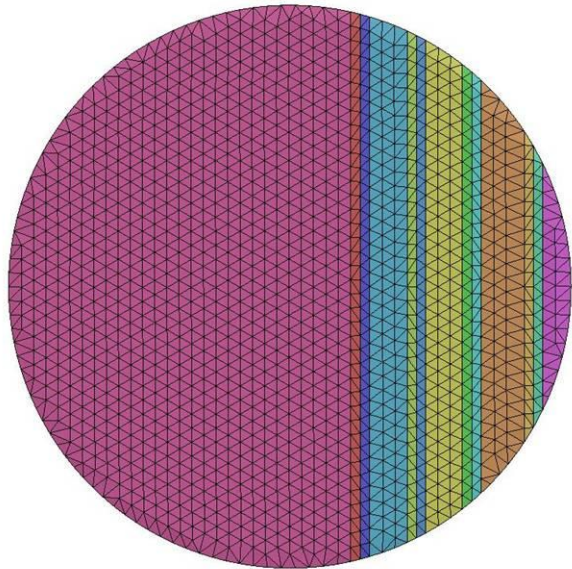
Unfolded View



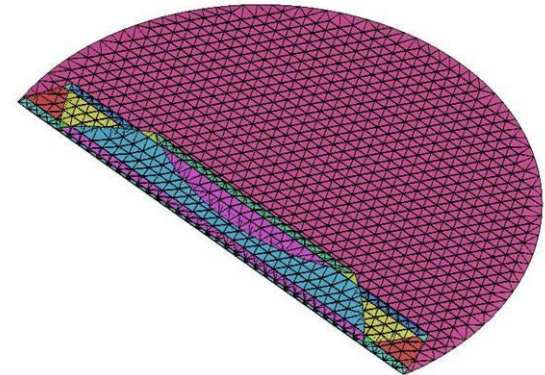
3rd Fold: Forward Subset



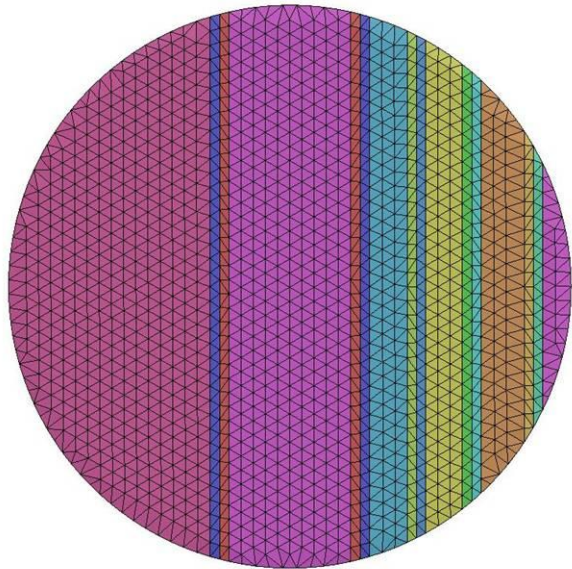
Unfolded View



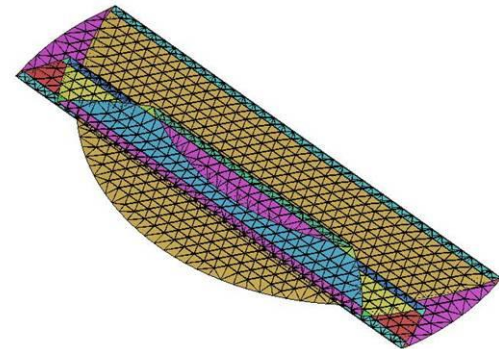
4th Fold: Reverse Subset



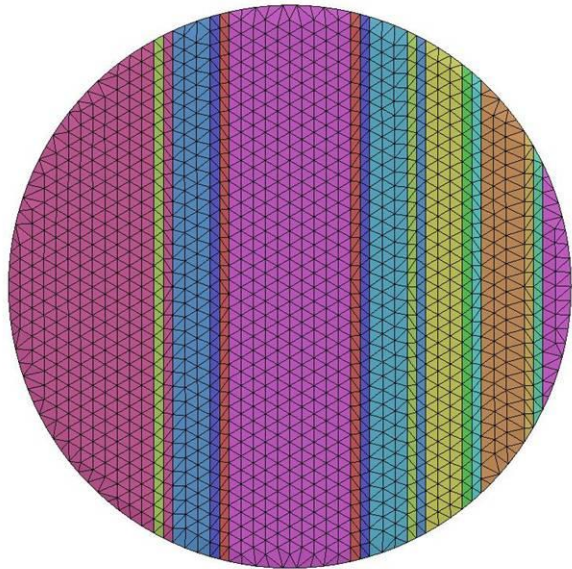
Unfolded View



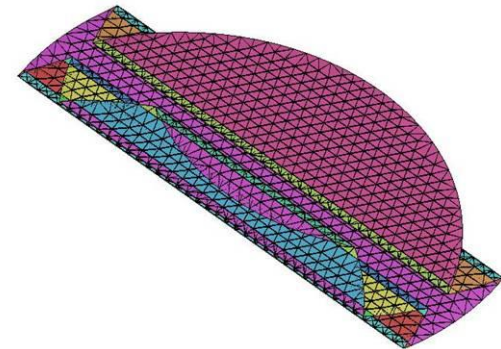
5th Fold: Reverse



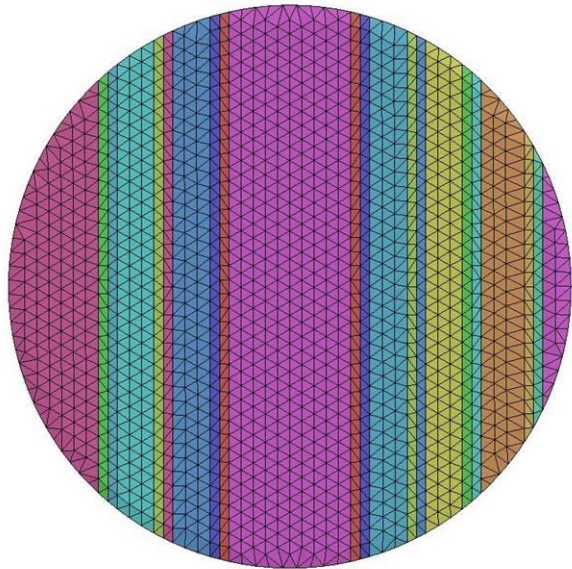
Unfolded View



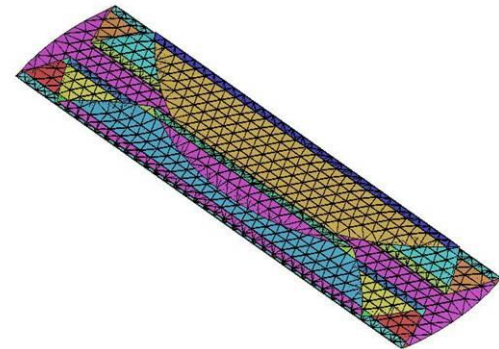
6th Fold: Forward



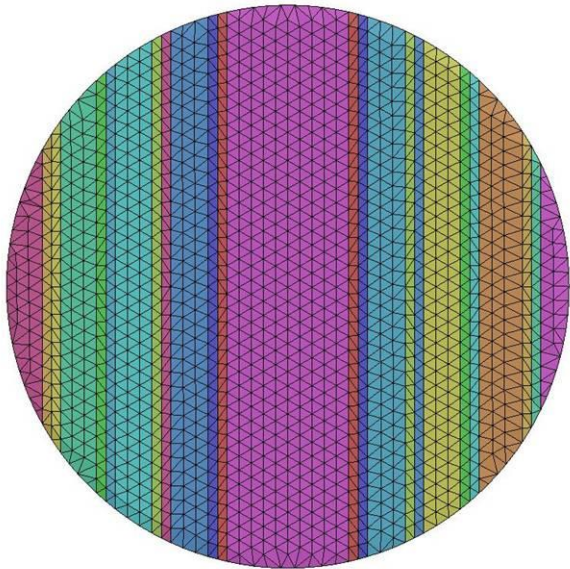
Unfolded View



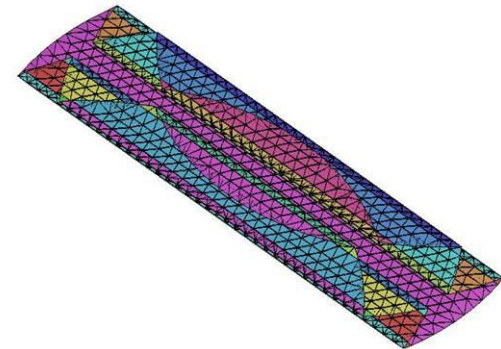
7th Fold: Reverse Subset



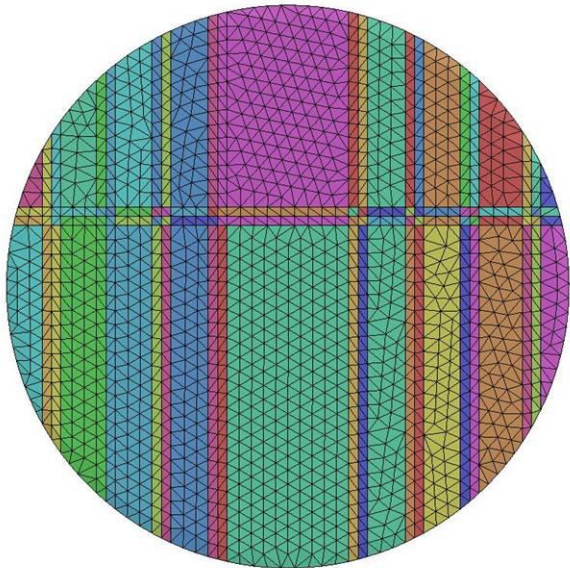
Unfolded View



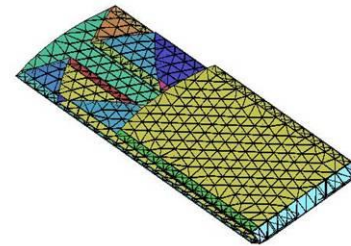
8th Fold: Forward Subset



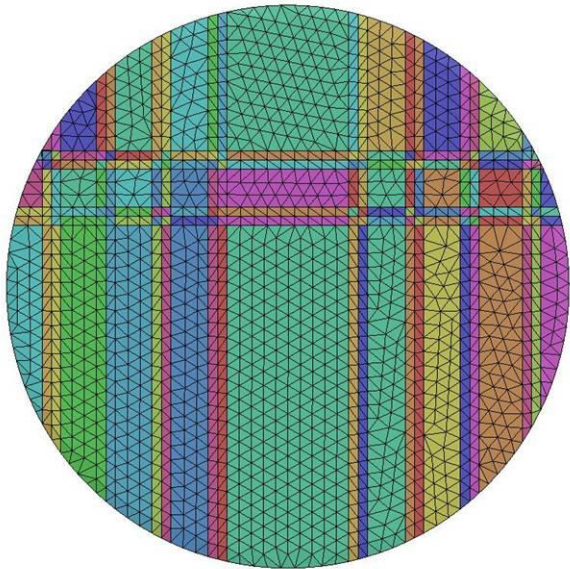
Unfolded View



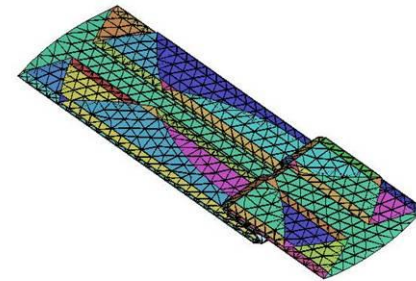
9th Fold: 90deg Reverse



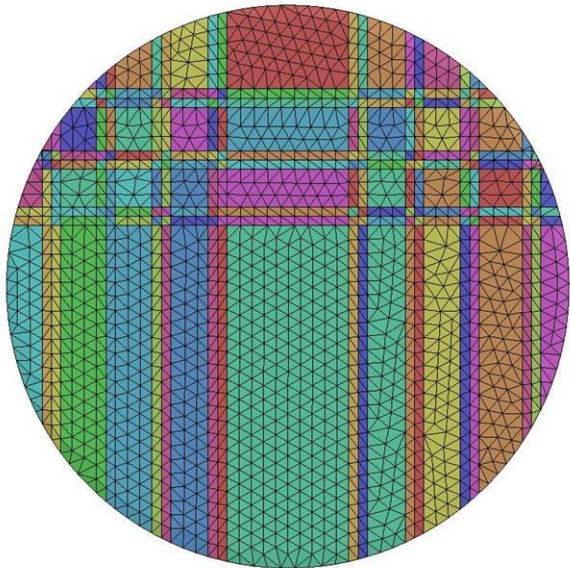
Unfolded View



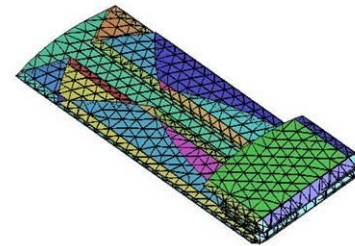
10th Fold: 90deg Forward



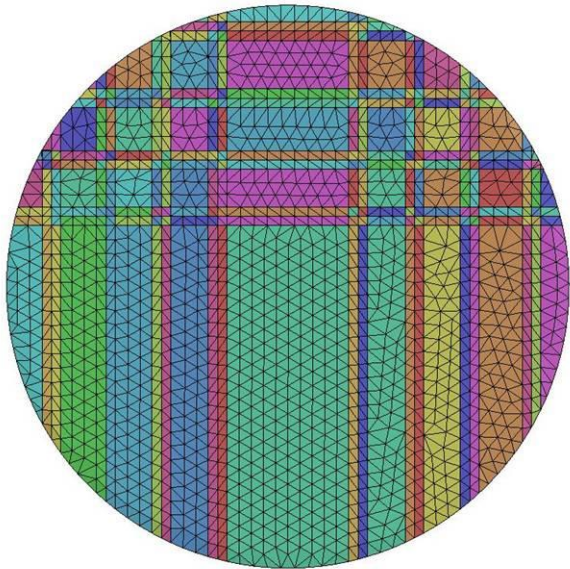
Unfolded View



11th Fold: 90deg Rev. Subset



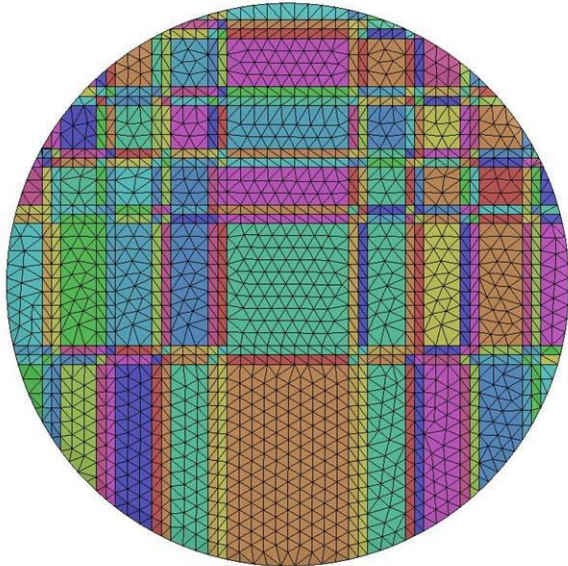
Unfolded View



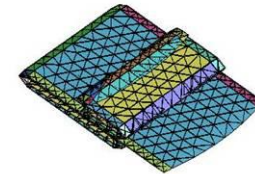
12th Fold: 90deg Fwd. Subset



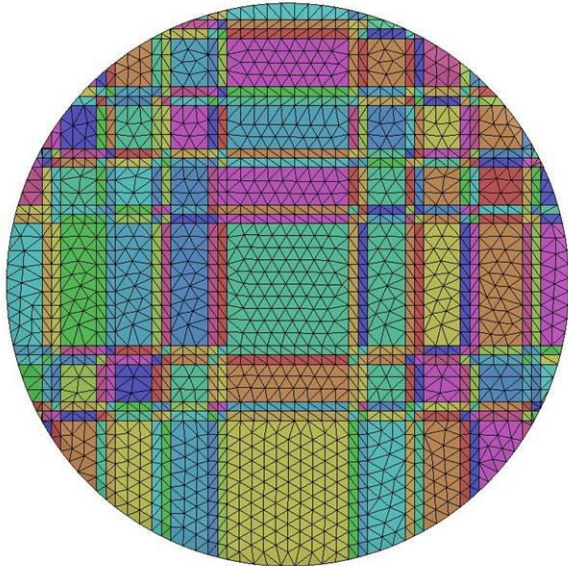
Unfolded View



13th Fold: 90deg Forward



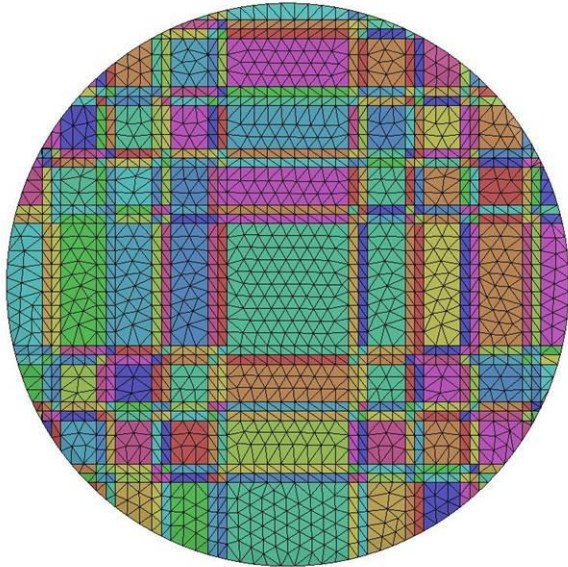
Unfolded View



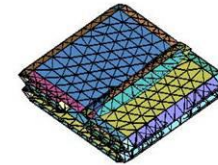
14th Fold: 90deg Reverse



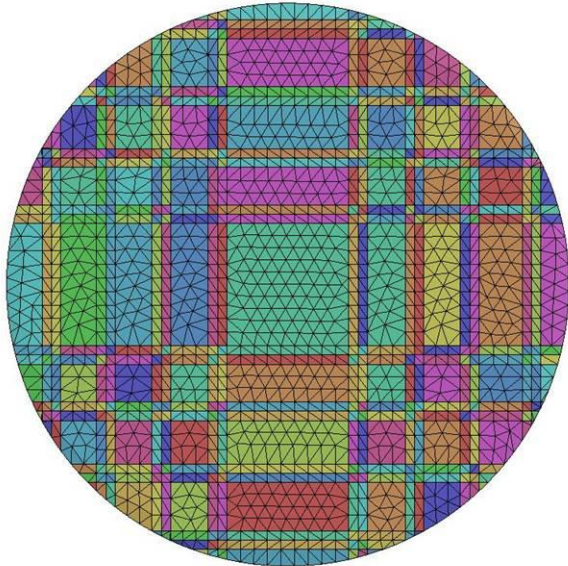
Unfolded View



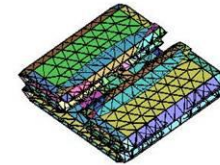
15th Fold: 90deg Fwd. Subset



Unfolded View



15th Fold: 90deg Rev. Subset



Time taken: 20minutes

Oasys PRIMER

A Training Course for New Users

September 2014

