

PRIMER 20.1

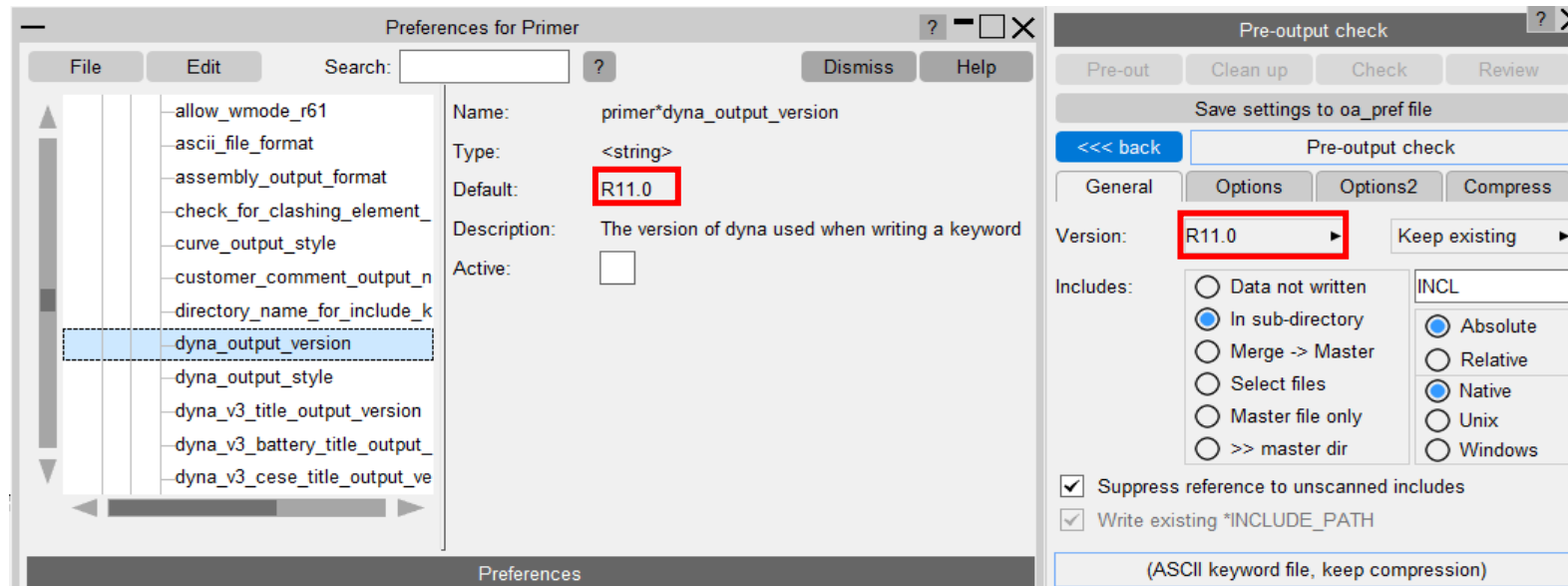
PRIMER 20.1 – Contents

- [Changes to Input and Output](#)
- [Favourite Tools & Keywords](#)
- [Performance improvements](#)
- [IGA enhancements](#)
- [Seatbelt enhancements](#)
- [Seatbelt Anchorage enhancements](#)
- [Higher order element improvements](#)
- [Occupant Positioning](#)
- [Visualisation entities for HBMs](#)
- [HBM Trees](#)
- [Further HBM Tool updates](#)
- [Spotweld Remesh enhancements](#)
- [Connections](#)
- [Assign mass dry run](#)
- [Crossed edge fix](#)
- [Beam-beam/shell automatic general contact](#)
- [Coat with 1D segment/beams](#)
- [User-defined Materials extended](#)
- [Cut section improvements](#)
- [Log option for DEFINE_CURVE and TABLE](#)
- [New Measure Mode: Node to Line](#)
- [Node Selection in the Orient Panel](#)
- [Morphing enhancements](#)
- [JavaScript](#)
- [Encryption Tool enhancements](#)
- [Crash Test Setup enhancements](#)
- [Pedestrian Markup enhancements](#)
- [HIC Area Calculator](#)
- [Analysis Tracking Tool](#)
- [Metal-Forming Tool](#)
- [Workflow Tools](#)
- [Help & Documentation](#)
- [Miscellaneous](#)

Changes to Input and Output

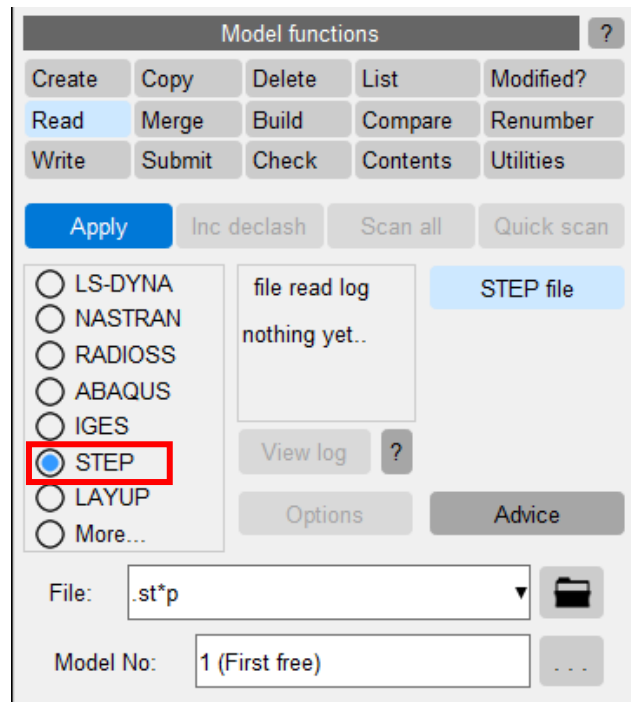
LS-DYNA Keyword Support

- PRIMER 20.0 keywords:
 - LS-DYNA up to and including R13 fully supported.
 - LS-DYNA R14.0 additions and modifications to commonly used keywords supported, including *CONTROL, *DATABASE, *MAT.
- Default output version remains R11.0 (later versions including R14.0 can be selected).



STEP File Reader

- PRIMER 20.0 can read STEP files.
- Select STEP file format in the Read panel, choose the file and click apply.
- This is a beta-version for PRIMER 20.0, please report any issues to the support team.



New 'Post *END data' pre-output option

- Added a new pre-output check option 'Post *END data' that controls whether data can be written after *END.
- Can help with compatibility between softwares.
- This over-rides other options/preferences that control writing specific types of data (e.g., 'write out all connections').
- A preference variable 'write_post_end_data' associated with this new option has also been added with a default value of TRUE. This is the default behaviour in previous versions.

Pre-output check

Pre-out Clean up Check Review

Save settings to oa_pref file

<<< back Pre-output check

General Options Options2 Compress

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

HM comm : on ▶ ☒ Thumbnails

☒ Part colours ☐ Field headers

☒ Emb'd comments ☐ ANSA comment

☒ Timestamp ☒ Xref comments

☐ ZTF output ☒ Post *END data

ZTF opts Fast_full ▶

Writing latent/encrypted parameters

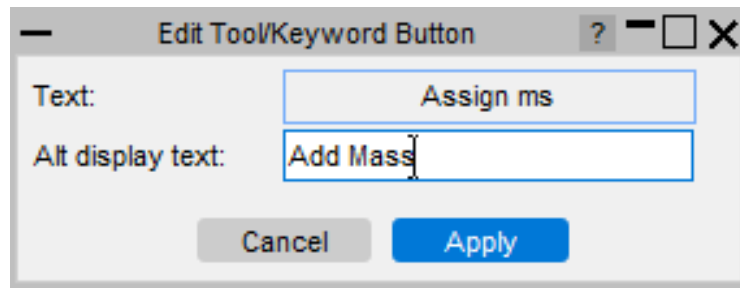
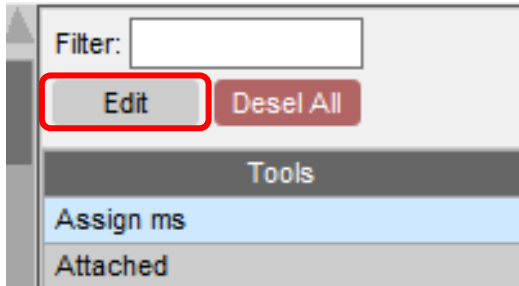
- Fields referring to latent or encrypted parameters are now written as &name (where 'name' is the name of the parameter) instead of the default value of '0' even when the pre-output option 'Write Parameters as values' is set to TRUE.
- This helps prevent unintended model errors.
- The only exception is for the IDEAS output format that does not support the concept of parameters.

Favourite Tools & Keywords

Favourite Tools & Keywords

- The ability to create lists of Favourite Tools and Favourite Keywords was added in PRIMER 19.0.
- Now options from the Mesh Tools list can also be added to your Favourite Tools.
- Additionally, greater granularity has been added to the options available for Favourite Keywords.
- A filter has also been added to make it easier to find the Tools and Keywords you want to add.

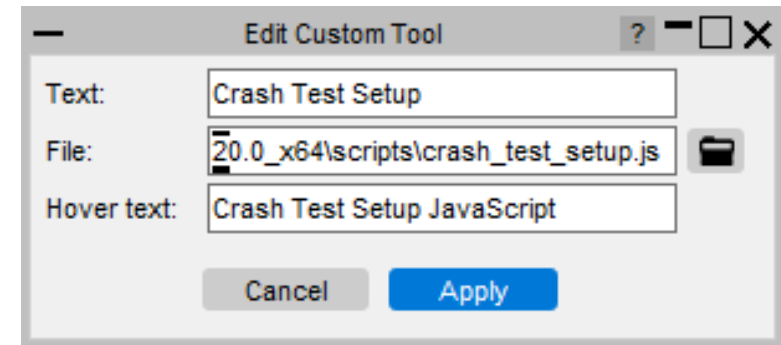
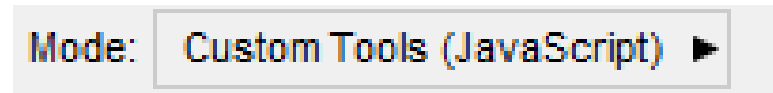
Favourite Tools & Keywords



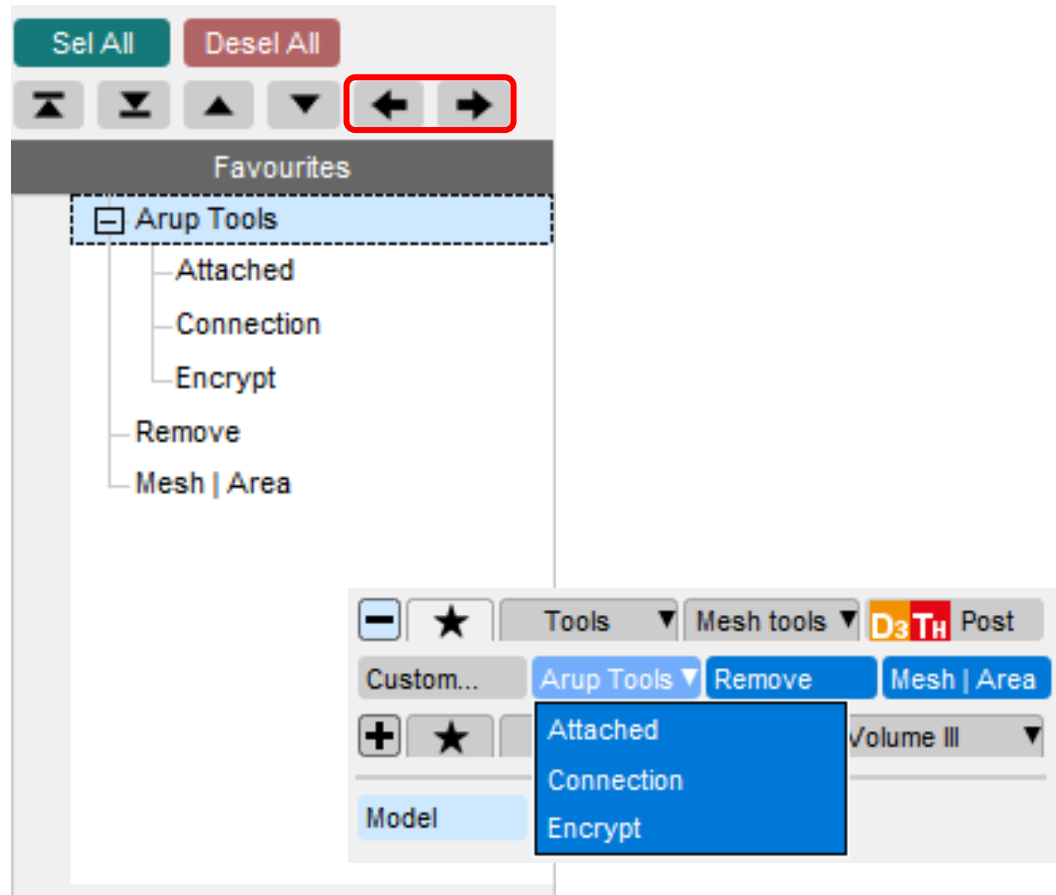
- You can define your own names for standard PRIMER Tools and Keywords buttons and add them to your Favourites to make them easier to find or consistent with other familiar software.

Favourite Tools & Keywords

- Custom JavaScript Tools (created using the PRIMER JavaScript API) can now be added to your Favourite Tools to be accessed with just one click.
- Switch to Custom Tools (JavaScript) mode and add a Tool by defining button text, a JavaScript file, and optional hover text, then add it to your Favourites like any other Tool.



Favourite Tools & Keywords



- Popup buttons can now be created for your Favourite Tools too.
- Control the hierarchy of your Favourite Tools from the Edit Favourite Tools panel using the ← and → buttons.
- Group related Tools together to make them easier to find.

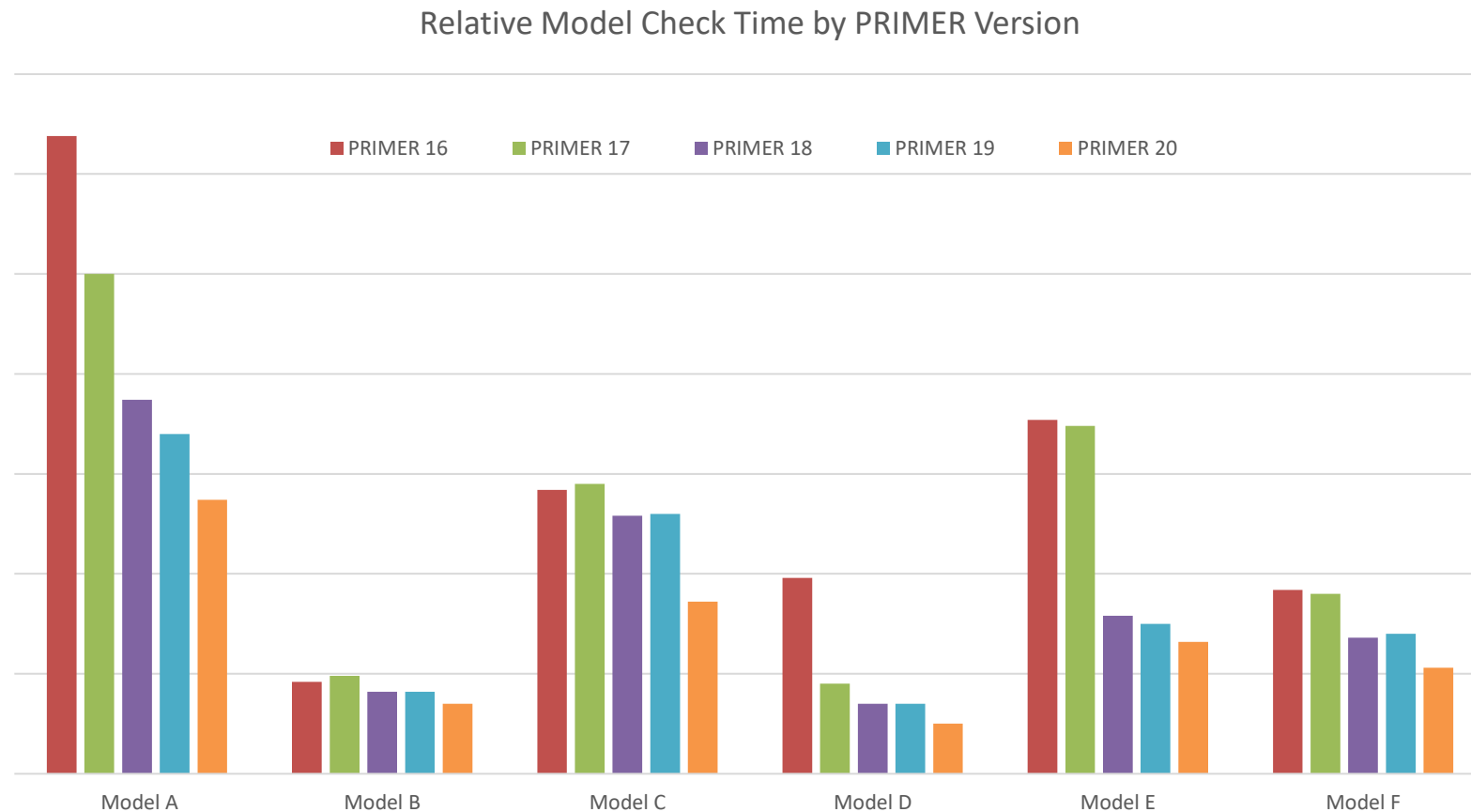
Favourite Tools & Keywords

- To support these additional features, Favourite Tools & Keywords data is now saved to a configuration file, `oa.conf`, instead of the `oa_pref` file as in PRIMER 19.x.
- Favourites options saved in the `oa_pref` file from PRIMER 19.x will still be read into PRIMER 20.0, but won't be saved out as this is now deprecated.
- Options in the `oa.conf` file will take precedence over those in the `oa_pref` file.
- Additionally, options read into PRIMER from an `oa.conf` file in the `OA_ADMIN` directory will be locked for editing and highlighted in magenta on the Edit Favourite Tools/Keywords panel.

Performance improvements

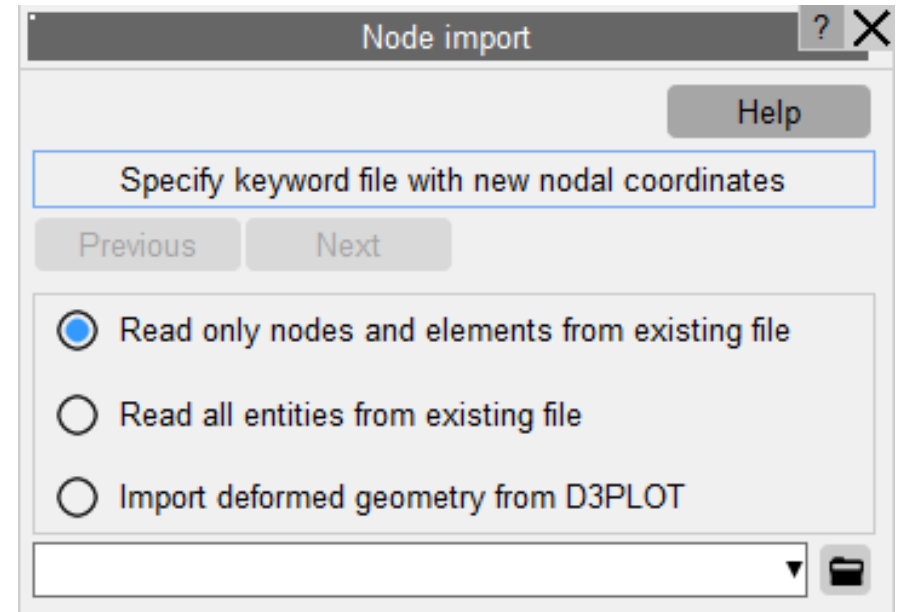
Speed of Model Check

- Model Check has been speeded up by parallelising and optimising calculations.
 - Models A to F in the graph below represent a diverse range of model types and sizes from ~5m to ~44m elements.



Node Import Read-in Speed Up

- You can now speed up the process of reading in a dynain file through 'Node Import'.
- The default mode now only reads in node and element data from the file – this is the faster option.
- If more data is needed, then the 'Read all entities' option should be selected, although this will be slower.
- These options have also been added to other dynain import places (HBM positioning and Seatsquash Import).



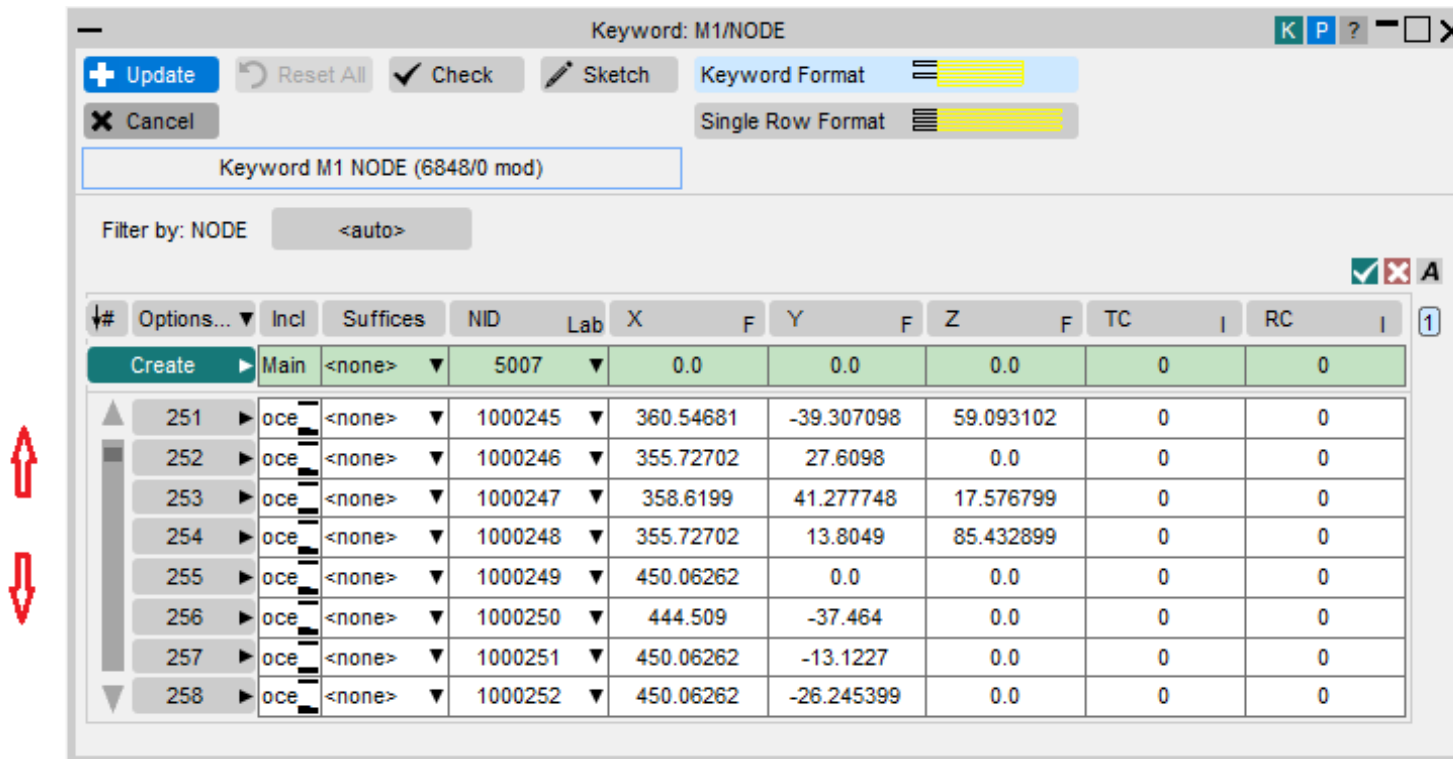
Speed up for *SET_..._GENERAL using BOX

- Test model with 1M shells + 1M solids + 500 SET_..._GENERAL<BOX>.
- This type of set is used for NODE, SEGMENT, SOLID, SHELL.
- Time required to decompose all sets:

	Speed up, PRIMER 20.0 vs 19.x
SET_NODE	1.0x
SET_SEGMENT	2.4x
SET_SHELL	1.9x
SET_SOLID	1.9x

Graphical User Interface speed improvements

- Speed of GUI update is about 2x faster on Windows platforms and about 5x faster on Linux. This will show up most clearly when scrolling lists with many buttons, for example in the keyword editor, but general responsiveness is subjectively better too.



Graphical User Interface Numerical Text Input Improvements

- Numerical text in fields in PRIMER 19.x and earlier was left-justified with leading white space (like a formatted keyout data field). In PRIMER 20.0 it is now centre-justified with no leading white space.
- Centre-justified looks neater with better alignment of data over multiple rows.
- If you click “in front of” the number in the box the cursor position will be immediately in front of the first digit and not some number of spaces to the left.

Filter by: NODE <auto> **V19: text was left-justified with leading white space** ✓✕ A

#	Options...	Incl	Suffices	NID	Lab	X	F	Y	F	Z	F	TC	I	RC	I
Create		Main	<none>	5007		0.0		0.0		0.0		0		0	
5		Main	<none>	5005		0.0		0.0		0.0		0		0	
6		Main	<none>	5006		0.0		0.0		0.0		0		0	
7		oce	<none>	1000001		455.14261		13.8049		121.8184		0		0	
8		oce	<none>	1000002		438.58182		44.196		95.427803		0		0	

Filter by: NODE <auto> **V20: text centre-justified with no leading white space** ✓✕ A

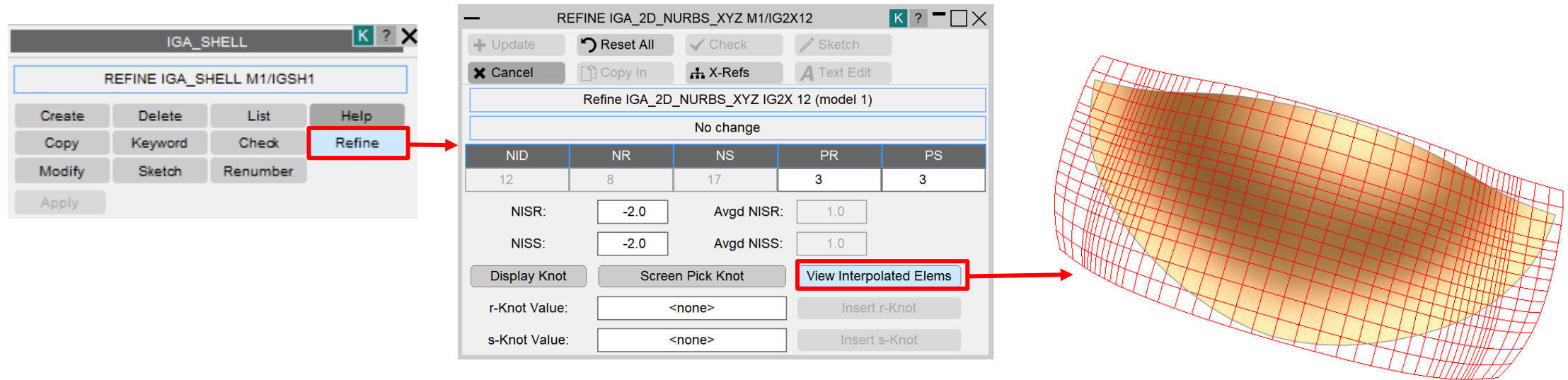
#	Options...	Incl	Suffices	NID	Lab	X	F	Y	F	Z	F	TC	I	RC	I
Create		Main	<none>	5007		0.0		0.0		0.0		0		0	
5		Main	<none>	5005		0.0		0.0		0.0		0		0	
6		Main	<none>	5006		0.0		0.0		0.0		0		0	
7		oce	<none>	1000001		455.14261		13.8049		121.8184		0		0	
8		oce	<none>	1000002		438.58182		44.196		95.427803		0		0	

IGA enhancements

IGA images are courtesy of JSOL

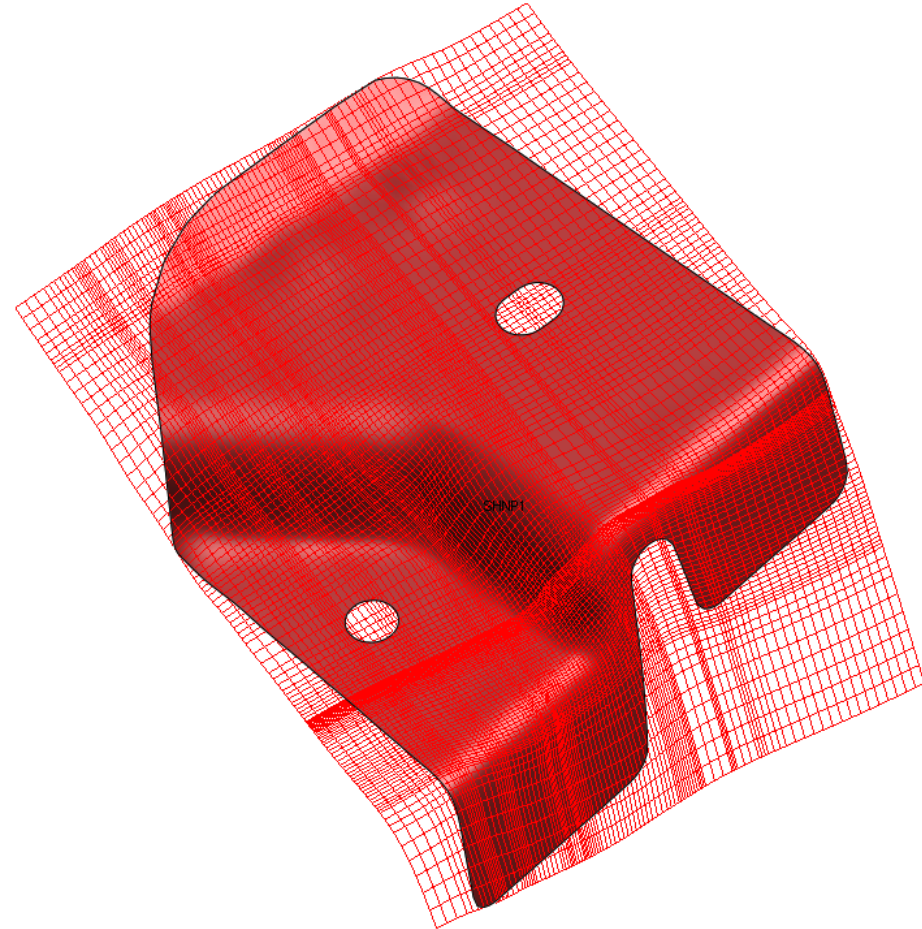
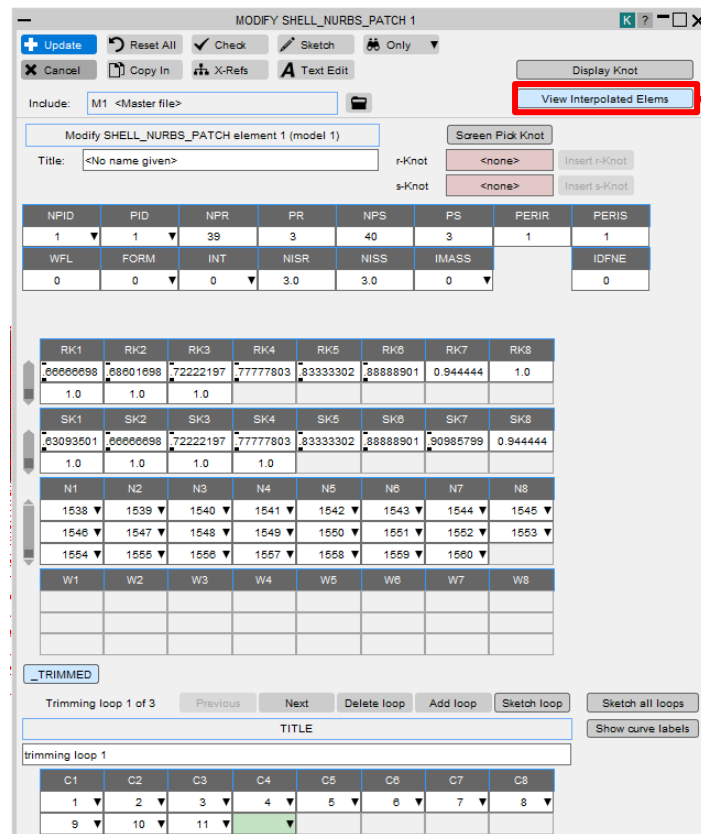
IGA enhancements

- Visualisation of interpolation elements in IGA models is now possible.
- The new option “View Interpolated Elms” has been added in the refine panel. It can be accessed from IGA_SHELL, IGA_FACE_XYZ or IGA_2D_NURBS_XYZ.



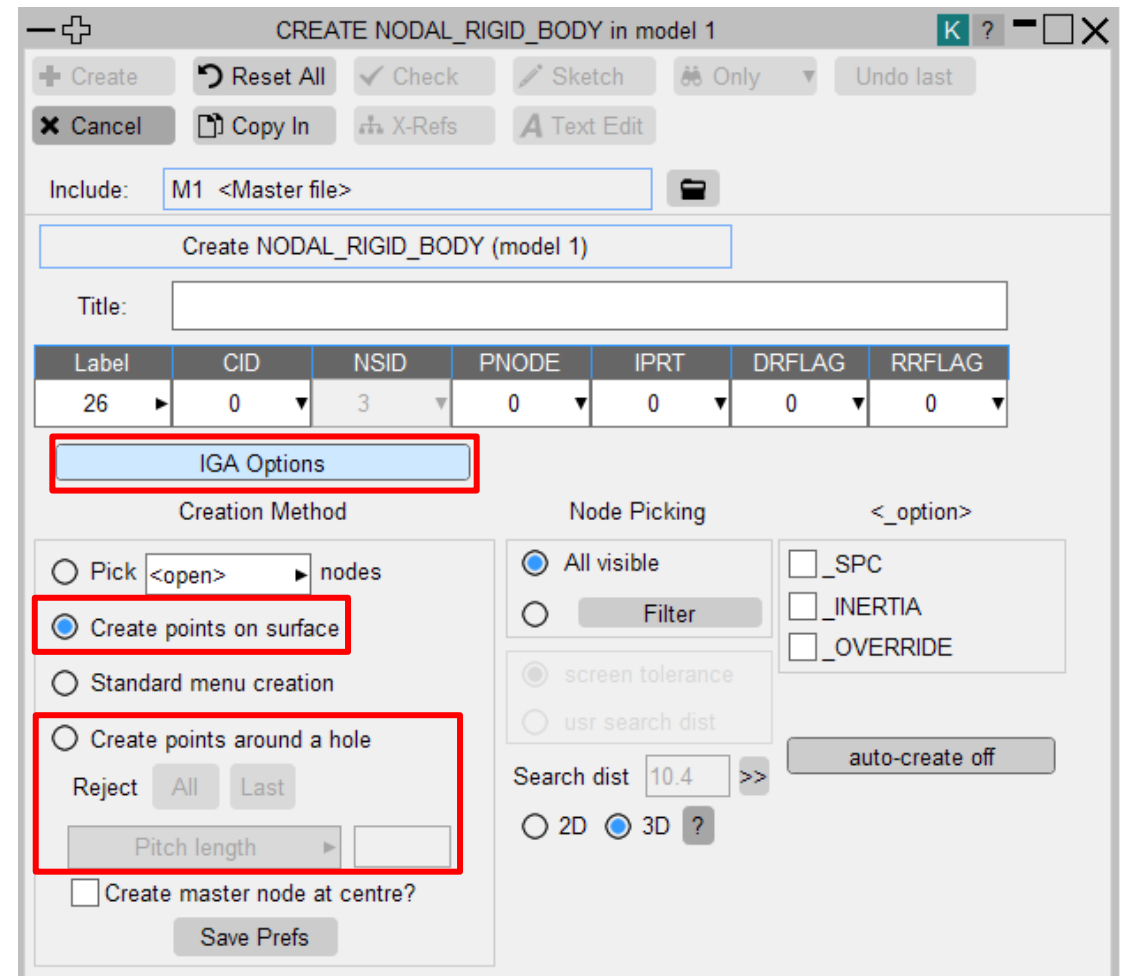
IGA enhancements

- The option “View Interpolated Elms” is also available in
*ELEMENT_SHELL_NURBS_PATCH edit panel



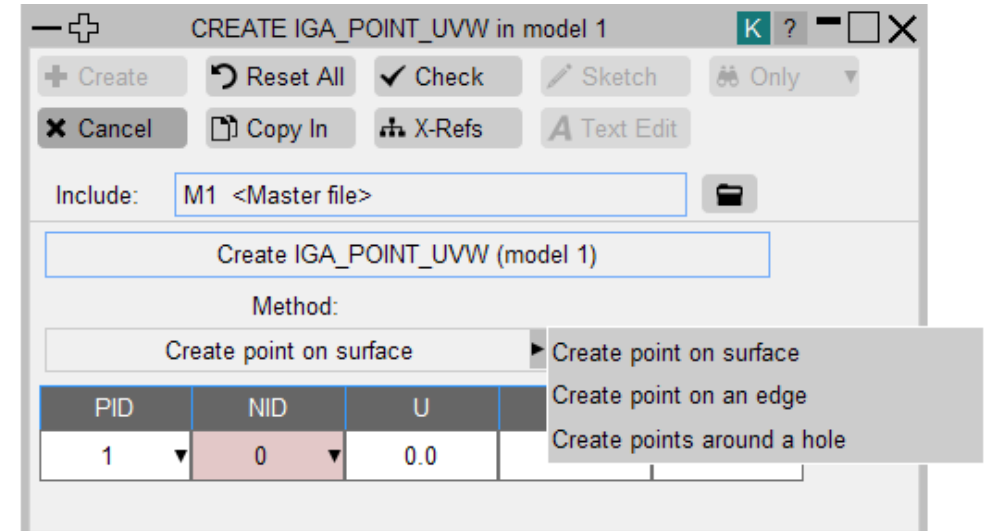
IGA enhancements

- Added “IGA Options” in CNRB create panel.
- You can pick anywhere on IGA_SHELL using “Create points on surface”, it will create IGA_POINT_UVW and NODE automatically and add it on SET_IGA_POINT_UVW on corresponding IGA_FACE_XYZ definition.
- You can also pick IGA_1D_BREP (trimming loop) using “Create points around a hole” option. It will let you create number of nodes and IGA_POINT_UVW on the trimming loop with “Pitch length” or “Number of Nodes” option.



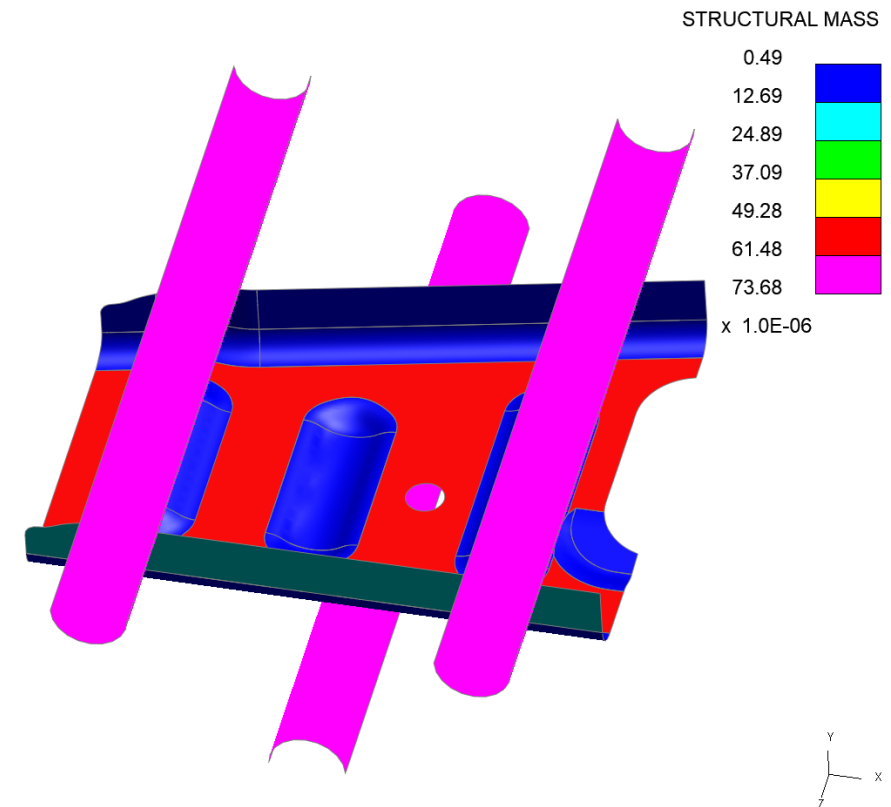
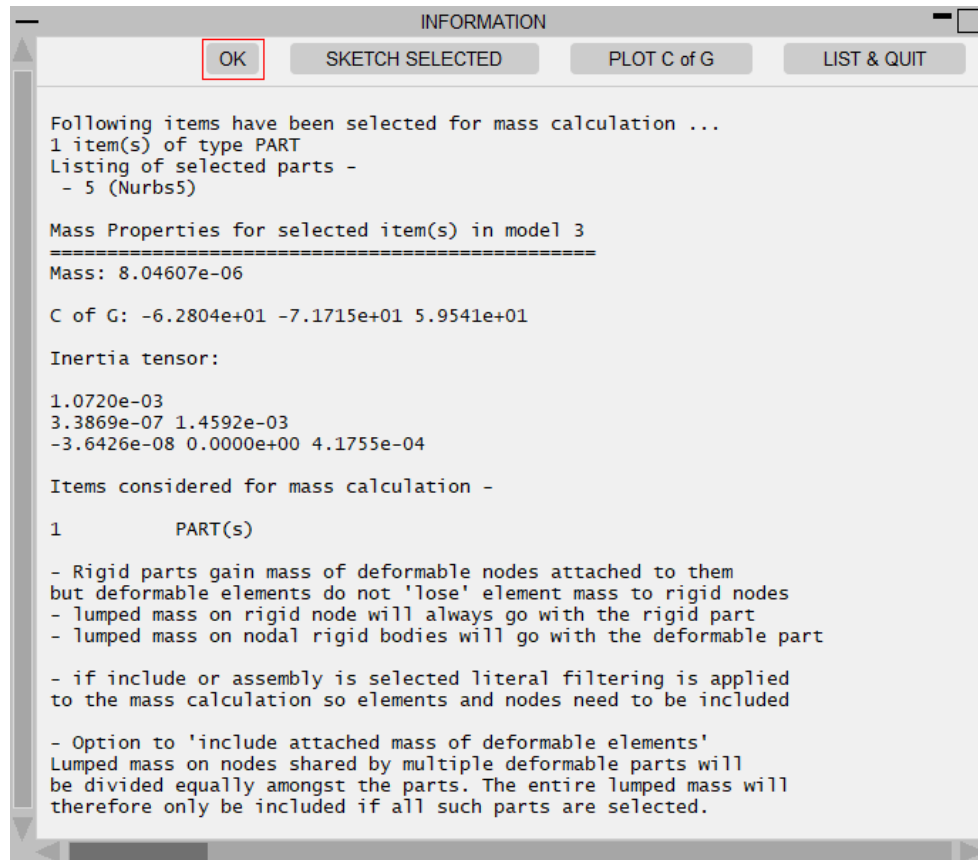
IGA enhancements

- Changed IGA_POINT_UVW create/modify panel for improved user experience.
- Now you can use any of the following methods to create IGA_POINT_UVW:
 1. Create point on surface: Pick and create single IGA_POINT_UVW and NODE on IGA_SHELL;
 2. Create point on an edge: Pick and create single IGA_POINT_UVW and NODE anywhere on IGA_1D_BREP (trimming loop);
 3. Create points around a hole: Pick IGA_1D_BREP (trimming loop) and create nodes and IGA_POINT_UVW on the trimming loop using “Pitch length” or “Number of Nodes” options.



IGA enhancements

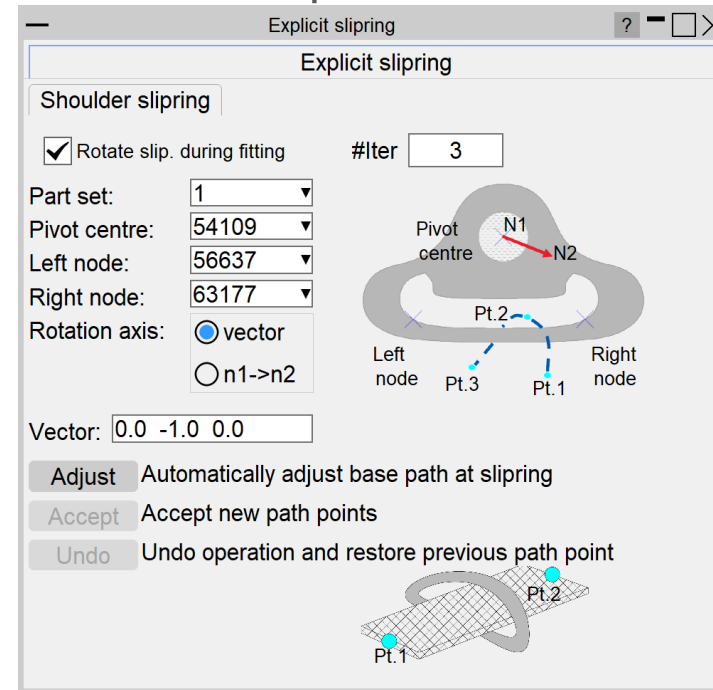
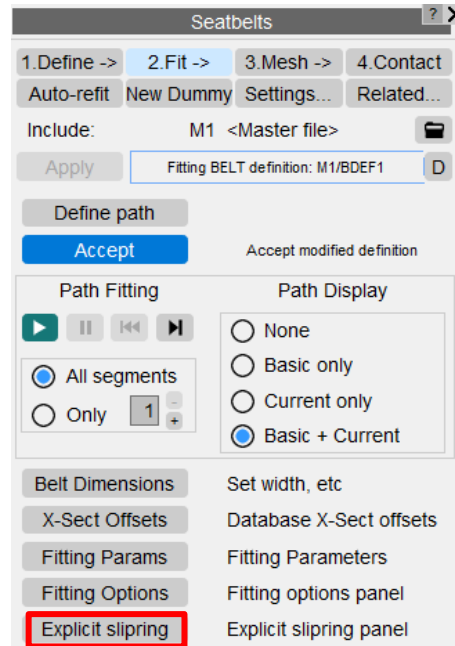
- *IGA_SHELL mass, centre of gravity, and inertial tensors can now be calculated and contoured.



Seatbelt enhancements

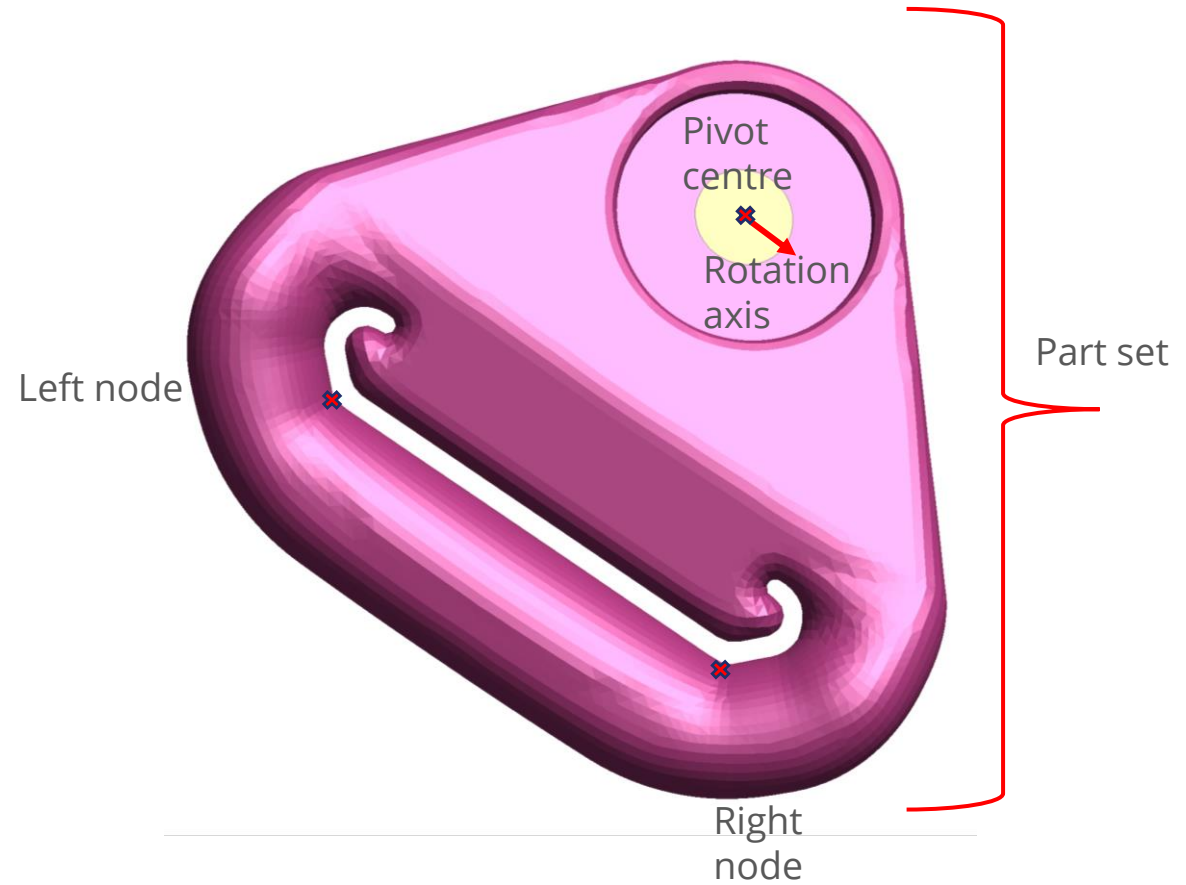
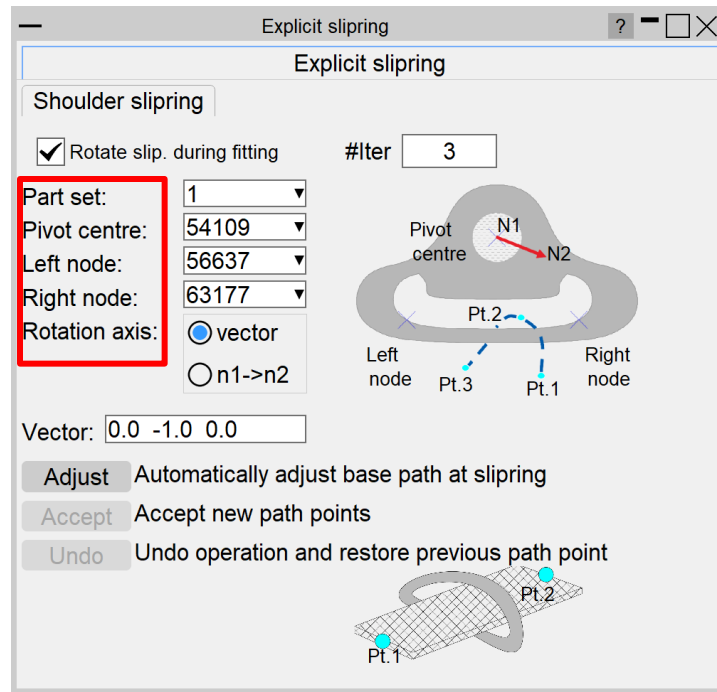
Seatbelt enhancements – Explicitly meshed shoulder slipping

- Added the option to automatically pull an explicitly meshed shoulder slipping (D-ring) into its correct position during the fitting process.
- To turn this option on, you must fill in all the required fields in the 'Explicit Slipping' panel which can be accessed from the main 'Fit' panel.



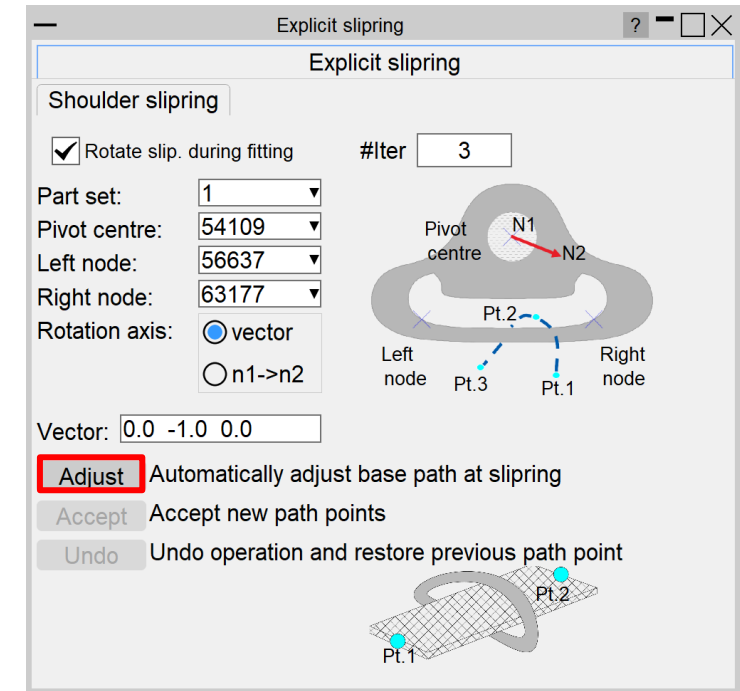
Seatbelt enhancements – Explicitly meshed shoulder slipping

- The entity fields must be defined as below.



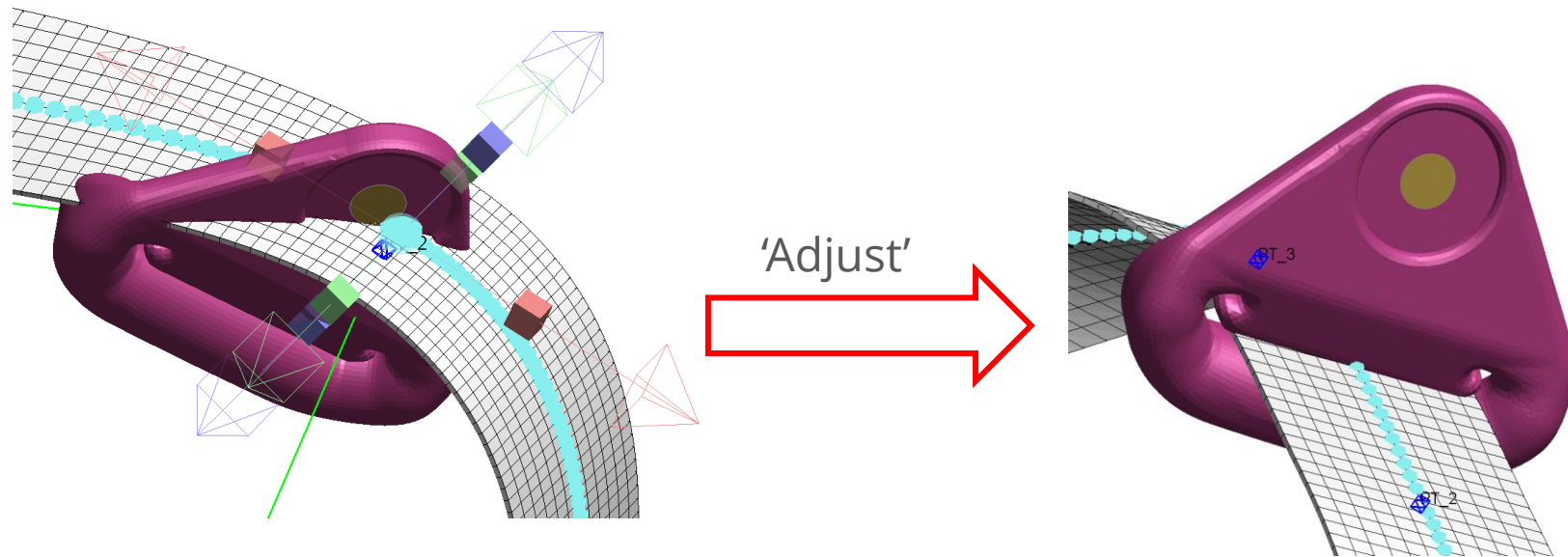
Seatbelt enhancements – Explicitly meshed shoulder slipping

- The 'Adjust' button in the bottom section of this panel can be used to automatically adjust the initial path of the belt within the slot of the shoulder slipping. This will create two base points at each end of the slot and adjust all twist handles in the correct configuration.
- You simply provide the information in this panel and add a single base point anywhere near the bolt of the slipping (it is advised to avoid creating more than one point near the slipping area for this to properly work) and then press the 'Adjust' button.
- You then have the option to accept the new configuration or restore to the previous one.



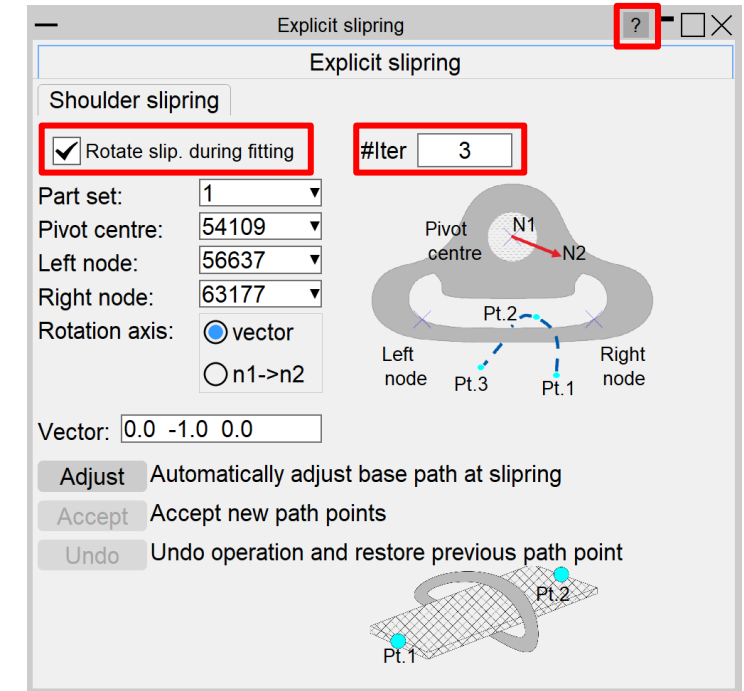
Seatbelt enhancements – Explicitly meshed shoulder slipping

- The below example shows how the 'Adjust' button affects the initial belt path at the shoulder slipping.



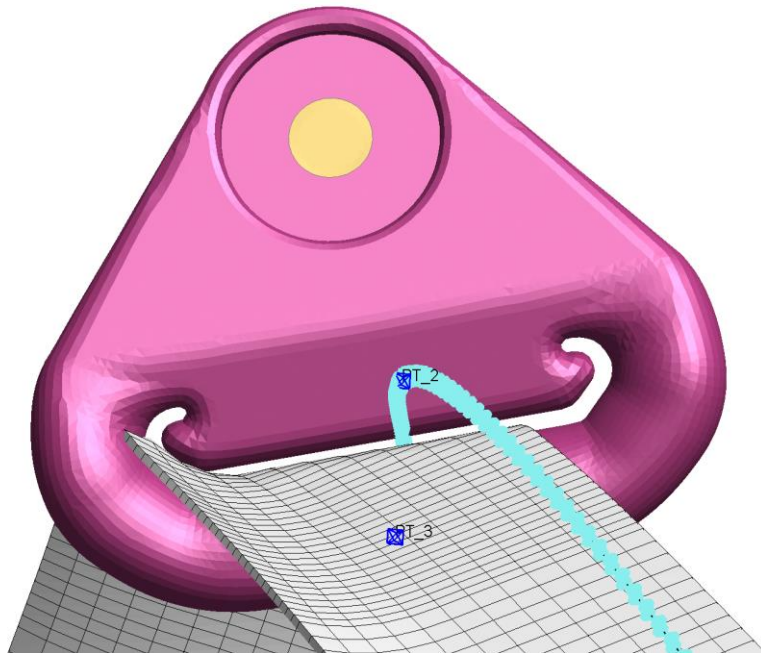
Seatbelt enhancements – Explicitly meshed shoulder slipping

- The 'Rotate slip. during fitting' tick box controls if the option to automatically pull the slipping during the fitting process is turned on.
- Only possible if all required fields are provided.
- The #Iter value gets automatically set to 3 to improve the robustness of this solution in terms of the original orientation of the slipping.
- The 'Help' section describes what each field refers to in more detail.

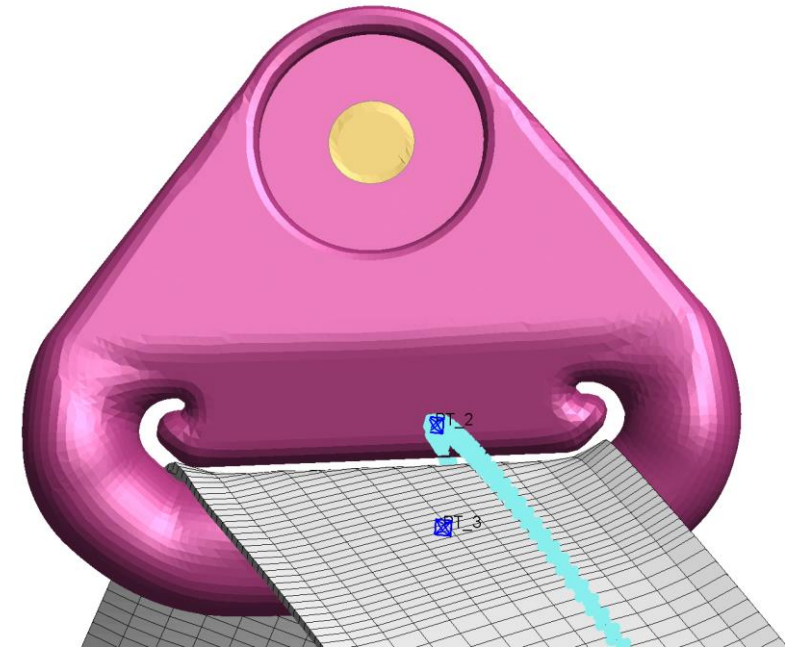


Seatbelt enhancements – Explicitly meshed shoulder slipring

- The below example shows the effect of the 'Rotate slip. during fitting' option after 1000 fit iterations.



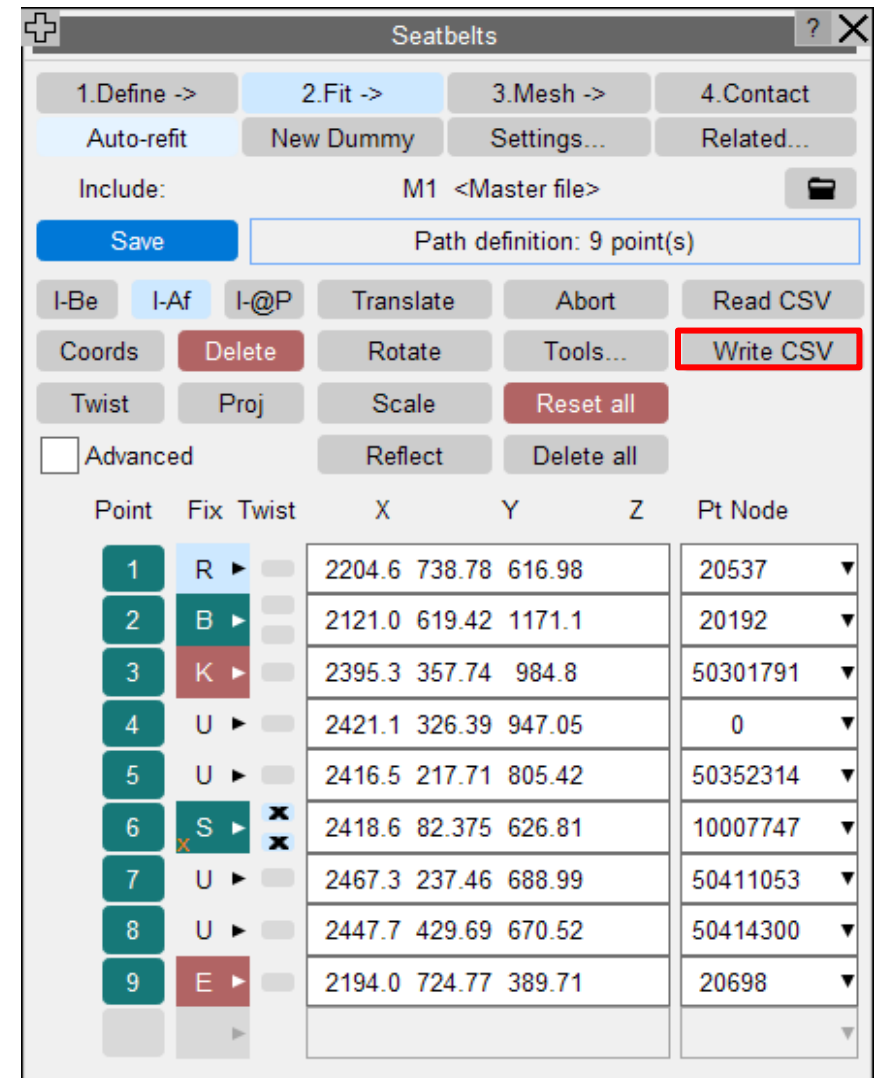
'Rotate slip. during fitting' off



'Rotate slip. during fitting' on

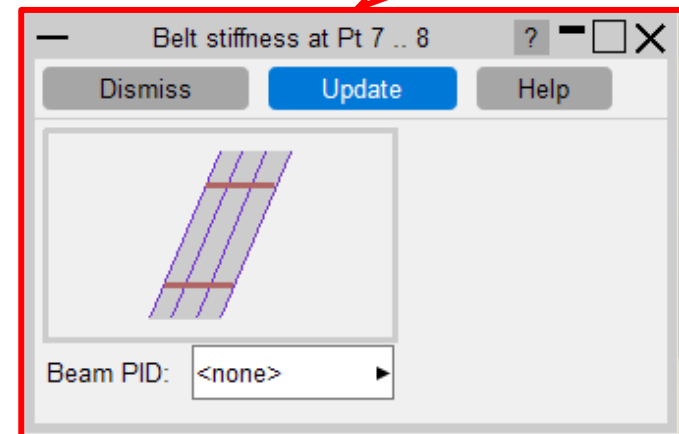
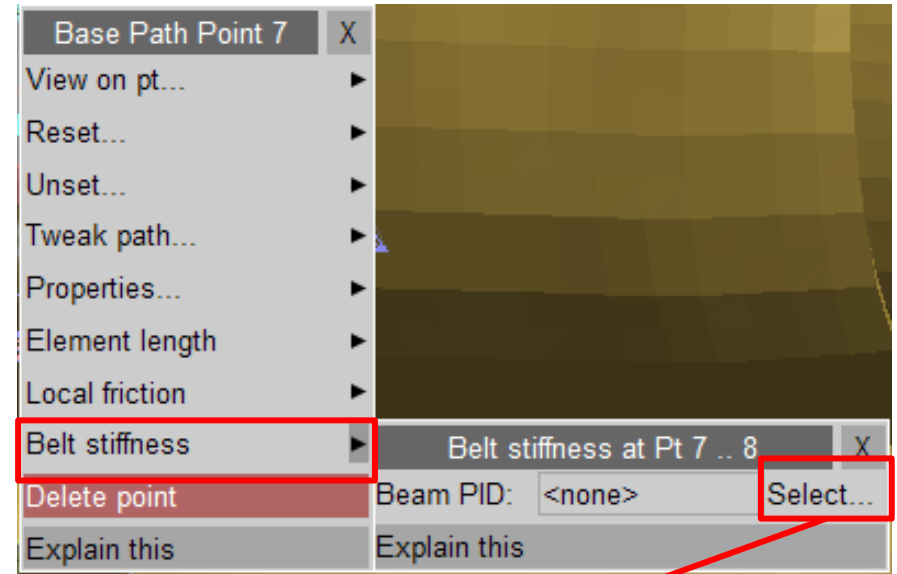
Seatbelt enhancements – Write CSV

- Added Write CSV option for belt path points.
- Also added support for vector along the seatbelt path for read CSV.



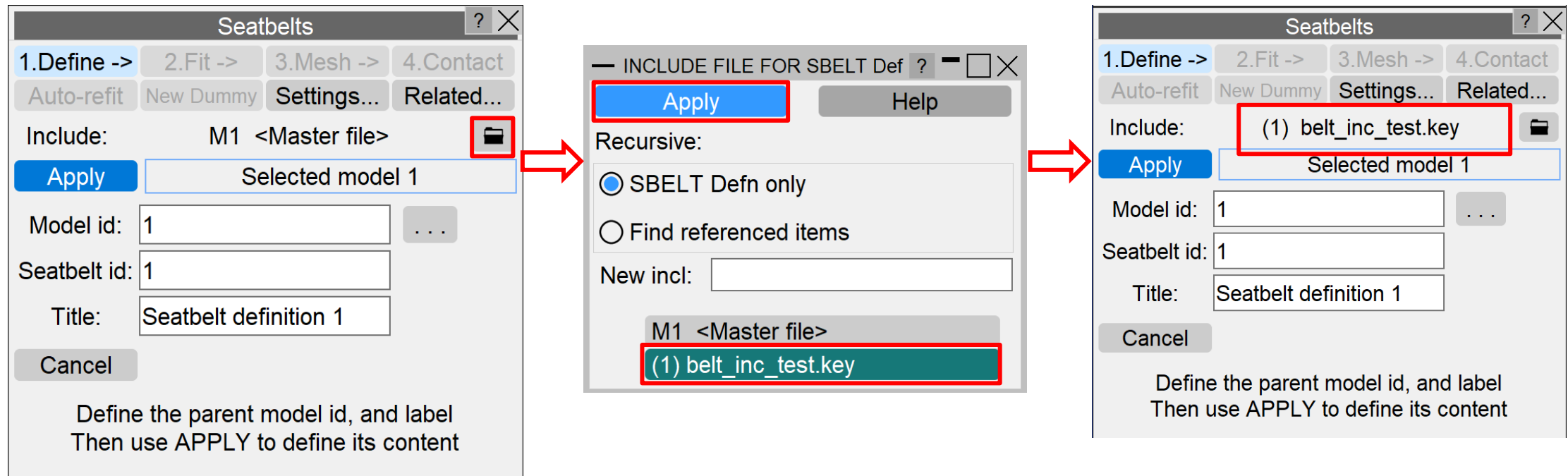
Seatbelt enhancements – Belt stiffness

- Add stiffener beams/strips to prevent belt 'curling'.
- You can access this option by going to Safety->Seatbelts->2.Fit->Define path->Tick Advanced option and right click on any base path point.
- There is an additional popup called "Belt stiffness" where you can give the part id that will be used for creating multiple beam elements between two base path points.
- You can also click on Select... which will open a new panel where there are additional options like Pick, Select, etc. for selecting the part id.



Seatbelt enhancements – Current include layer

- The current include layer can now be set while defining seatbelts ensuring the belt definition goes into the correct file.



Seatbelt enhancements – JavaScript API

- You can now set the existing retractor id present in the model using SetMesh:

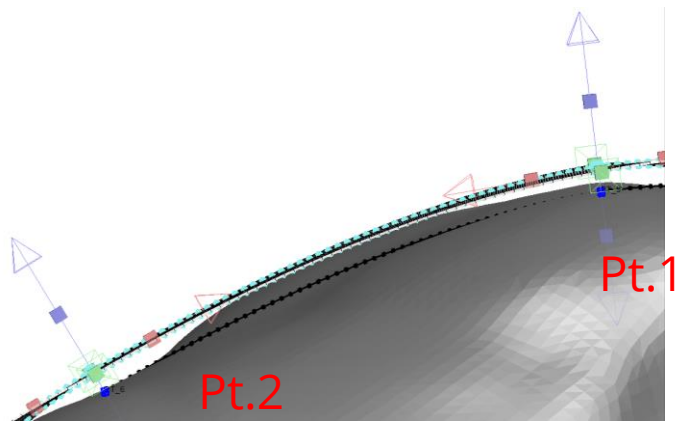
```
var data1 = { retractor: 10000001 };  
b.SetMesh(0, data1);
```

- You can now get and set the number of elements inside a retractor using the new retractor property:

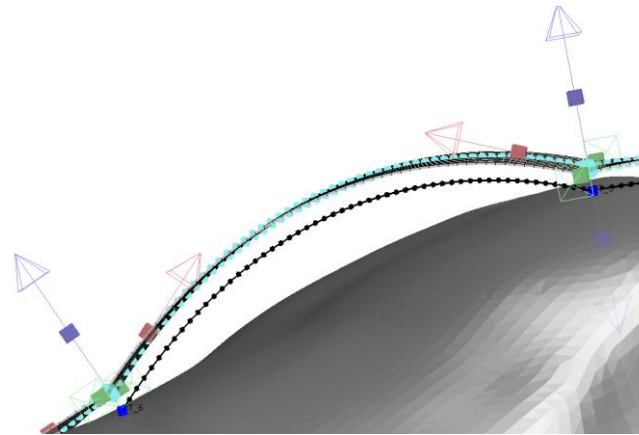
```
var r = Retractor.GetFromID(model, 10000001);  
var nsbi = r.nsbi;  
  
r.nsbi = 10;
```


Seatbelt enhancements – Red twist handle

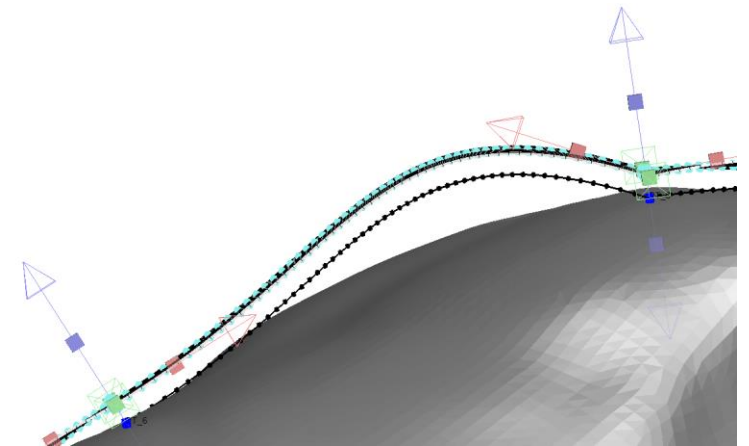
- Changed the behaviour of the red twist handles in the 'Advanced' path definition settings so that any handle can be modified independently without affecting red vectors at nearby path points.
- Below is an example of this effect if a change is applied only to the left-hand side red handle at point 1 in PRIMER 20.0 versus earlier versions.



Initial configuration



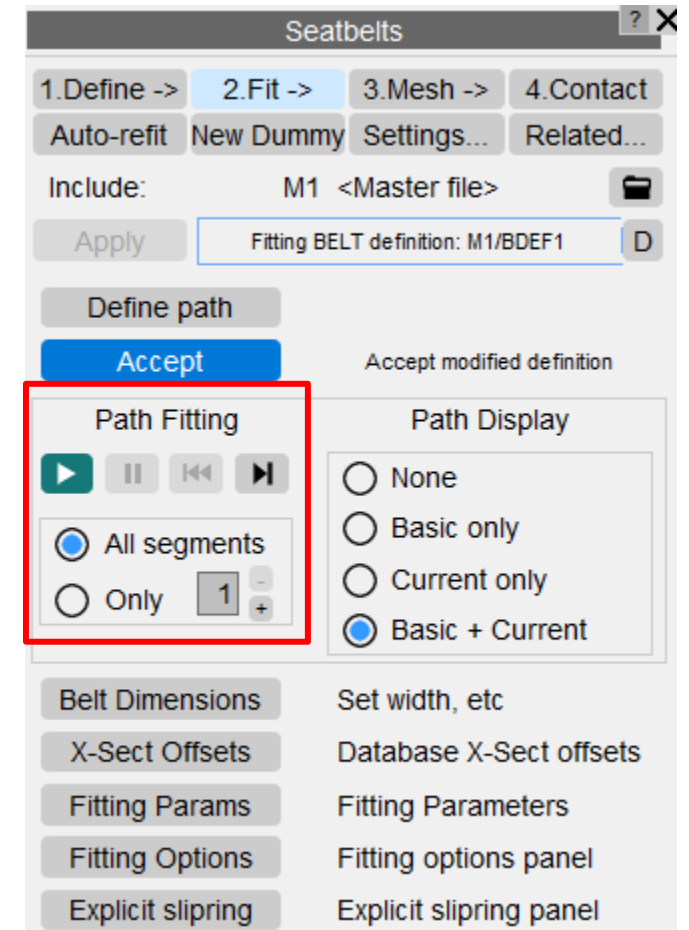
Result prior to PRIMER 20.0



Result in PRIMER 20.0

Seatbelt enhancements – Fit single segment option

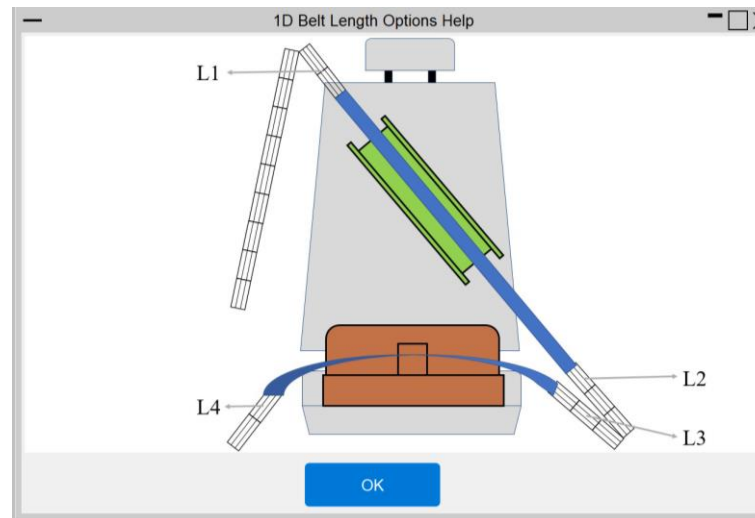
- Added the option to fit belt segments independently.
- This can be done by selecting 'Only' in the 'Path fitting' section and choosing the number of the segment to be fitted before starting the fitting process.
- Note that a 'segment' is defined as the stretch between end and slipring or between sliprings. If 'acute points' are used instead of sliprings these demarcate sliprings as well. Segments are numbered sequentially from 1.



Seatbelt Anchorage enhancements

Seatbelt Anchorage enhancements

- The Seatbelt Anchorage (SBA) tool positions loading devices (lap block and shoulder block) attached with a seatbelt system at the R-point in a vehicle and sets up an analysis according to the ECE R14 specification.
- In PRIMER 20.0 many enhancements have been made to improve the user experience of this tool.



SBA – Create *DATABASE_HISTORY Card For Loading Beams and Belts

Update 1D belt length ☐ ?

Length 1 ? Length 2 ?

Length 3 ? Length 4 ?

Set torso pts to 'Known Positions': ☒ ?

Create *DATABASE cards for loading beams and belt elements: ☒ ?

Change section and material ID's for seatbelt: ☒ ?

Section id for seatbelt 1D: Define ?

Material id for seatbelt 1D: Define ?

Section id for seatbelt shell: Define ?

Material id for seatbelt shell: Define ?

Section id for pulling seatbelt part: Define ?

Material id for pulling seatbelt part: Define ?

Keyword: M1/DATABASE_HISTORY_BEAM

+ Update Reset All Check Sketch Keyword Format

Cancel

Keyword M1 DATABASE_HISTORY_BEAM (4/0 mod)

Filter by: DATABASE_HISTORY_BEAM <auto> <auto>

#	Options...	Incl	Suffices	ID	B	TITLE
Create	Main	<none>		0		
1	Main	ID		120011		Chain SHL Free
2	Main	ID		120012		Chain Lap Free
3	Main	ID		120013		Chain Lap
4	Main	ID		120014		Chain SHL

Keyword: M1/DATABASE_HISTORY_SEATBELT

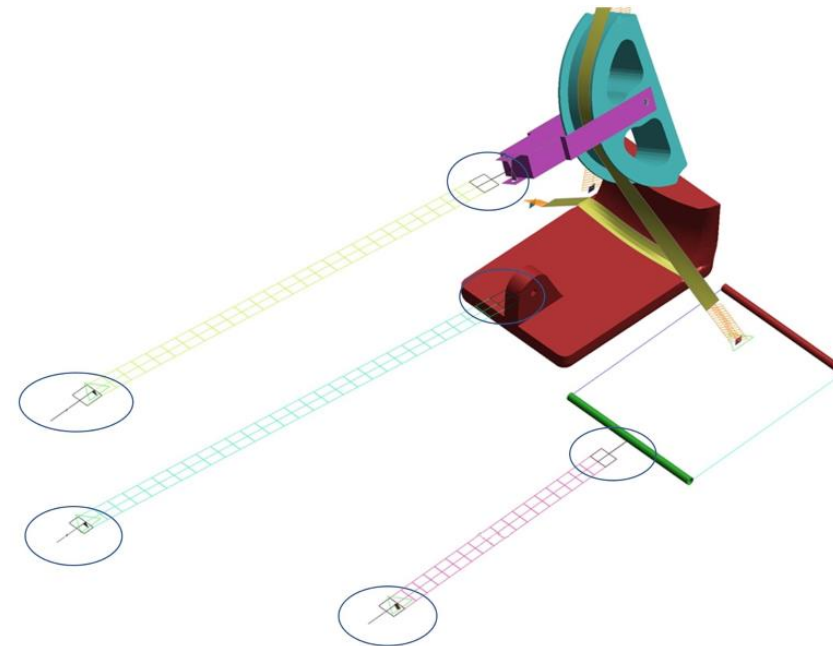
+ Update Reset All Check Sketch Keyword

Cancel

Keyword M1 DATABASE_HISTORY_SEATBELT (4/0 mod)

Filter by: DATABASE_HISTORY_SEATBELT <auto>

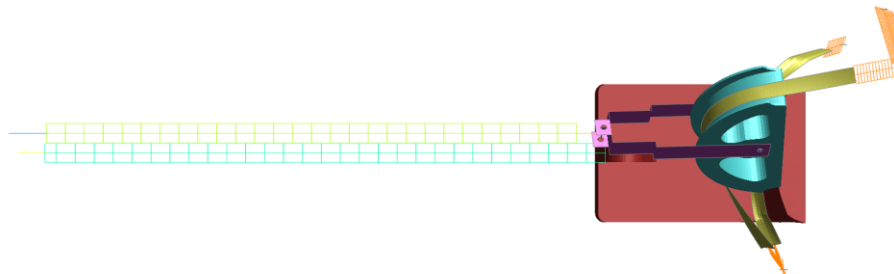
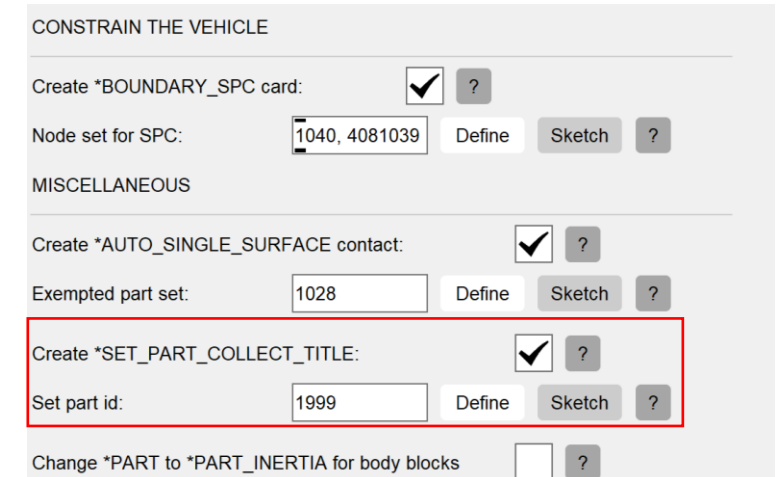
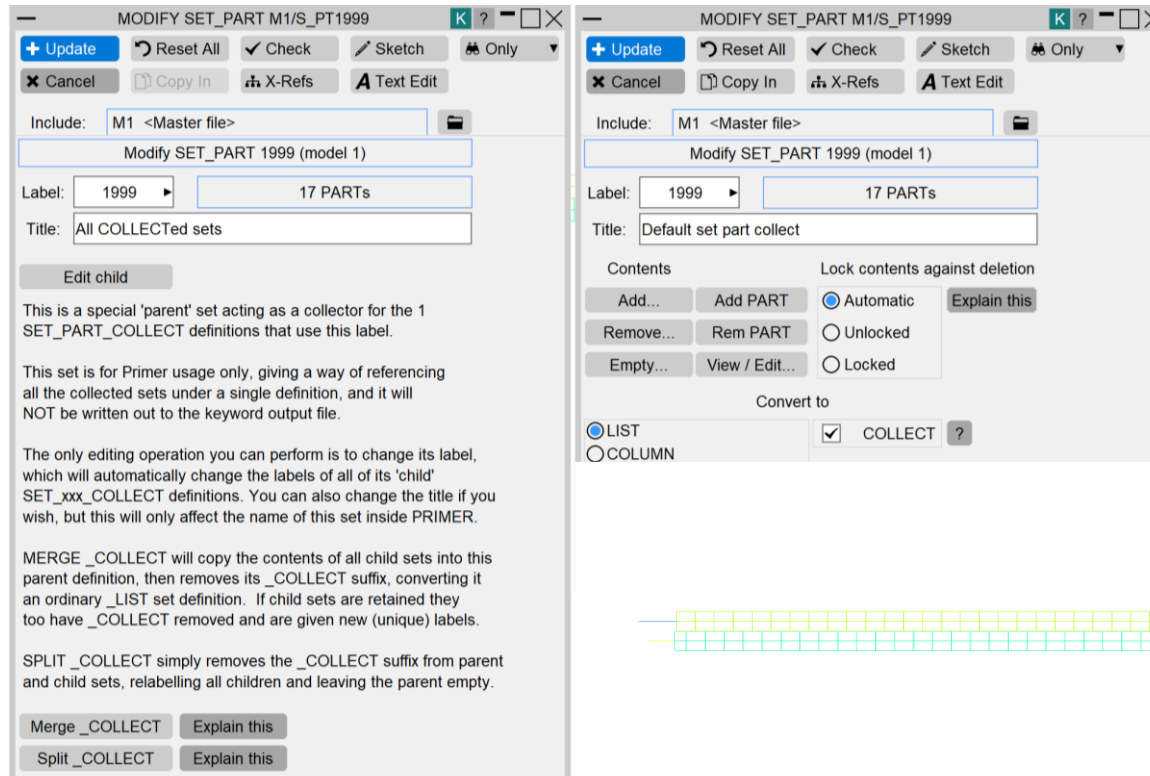
#	Options...	Incl	Suffices	ID	SB
Create	Main	<none>		0	
1	Main	<none>		9794	
2	Main	<none>		9764	
3	Main	<none>		9763	
4	Main	<none>		9793	



- Select this option to create *DATABASE_HISTORY cards for the SBA loading beams and the seatbelt elements connected to them.

SBA – Items created added to *SET_PART_COLLECT_TITLE

- Create a *SET_PART_COLLECT_TITLE to add the body blocks, seatbelt and parts created by the SBA tool so that it is out of global contact.
- Take the next free ID or provide a Set part id to use.



SBA – Options to control belt width, length and thickness

Create *DATABASE cards for loading beams and belt elements: ☐ ?

Change section and material ID's for seatbelt: ☐ ?

Section id for seatbelt 1D: Define ?

Material id for seatbelt 1D: Define ?

Section id for seatbelt shell: Define ?

Material id for seatbelt shell: Define ?

Section id for pulling seatbelt part: Define ?

Material id for pulling seatbelt part: Define ?

Seatbelt elem length: ?

Seatbelt width: ?

Seatbelt thickness: ?

Set your own seatbelt start labels ☐ ?

Node ? Node set ?

Shell ? Shell set ?

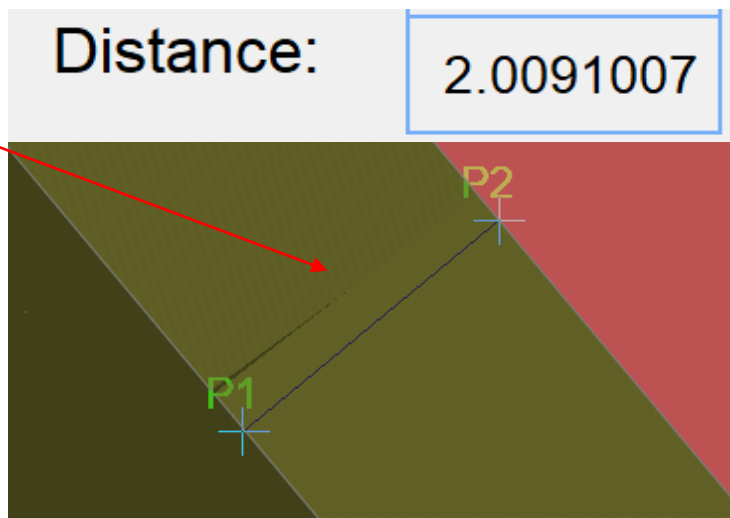
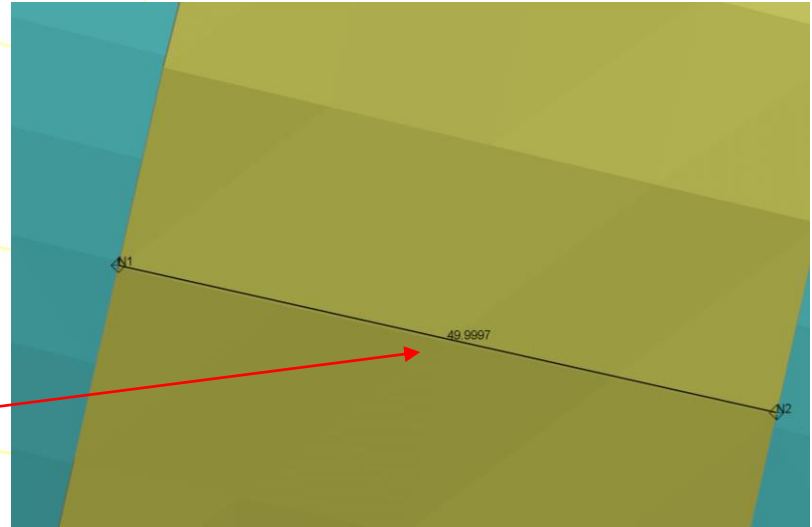
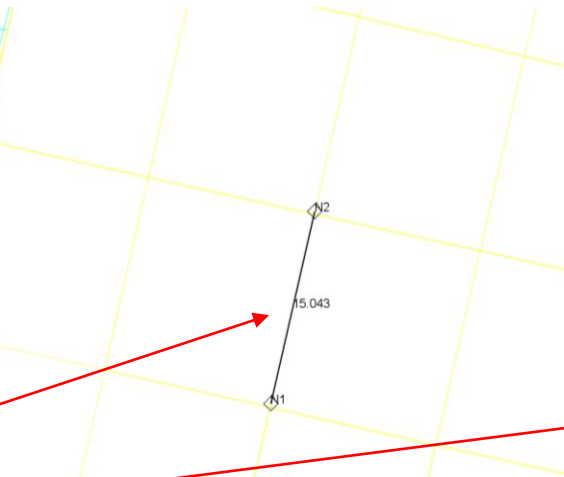
Seatbelt ? Nodal RBs ?

Retractors ? DB xsects ?

Sliprings ? Part sets ?

2D slip nset ?

<<< Input Panel Quit



Distance: 2.0091007

SBA – Inertia Centre of Gravity

- Centre of Gravity can now be specified by inputting your own coordinates.
- Or Centre of Gravity can be calculated on the fly using the seat structure part set selected in 'Seat Part Set'.

Recliner node set: 12200 Define Sketch ?

Seat part set: 12211 Define Sketch ?

Centre of gravity Coordinates ☒ ? Part set ☐ ?

CofG coordinates Sketch ?

Window

Centre of Gravity coordinates for the seat structure, enter a space and/or comma between each value (x, y, z)

OK

Recliner node set: 12200 Define Sketch ?

Seat part set: 12211 Define Sketch ?

Centre of gravity Coordinates ☐ ? Part set ☒ ?

CofG coordinates Sketch ?

SBA – Vehicle Class and Units Load Automation

- Load on blocks and Seat Inertia Loading will now update as per the table below based on the selected Vehicle Category, the Units in use and for Inertia load the seat part set selected.
- You can still manually edit the load values themselves.

Vehicle Category	N1: m< 3.5t	N2:3.5 <m < 12t	N3:m >12t
Should block load	13.5 kN	6.75kN	4.5kN
Lap block load	13.5 kN	6.75kN	4.5kN
Seat inertia loading	20X seat weight	10X seat weight	6.6X seat weight

Vehicle category: N2 ?

Load on blocks: 6.75 ? Units mm,kg,ms ?

APPLY INERTIA LOADING

Apply inertia load: ☒ ?

Default inertia load: 10 x Seat Weight ?

Inertia load: 1.18 ?

Scale inertia loading device Auto ☒ ? Manual ☐ ?

Inertia loading bar length: ?

Gap b/w inertia loading bars: ?

Recliner node set: 12200 Define Sketch ?

Seat part set: 22002 Define Sketch ?

Other Seat Belt Anchorage Enhancements

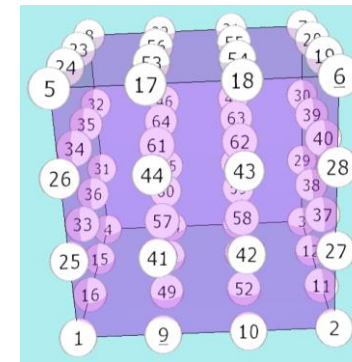
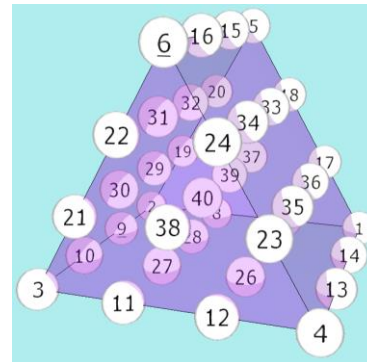
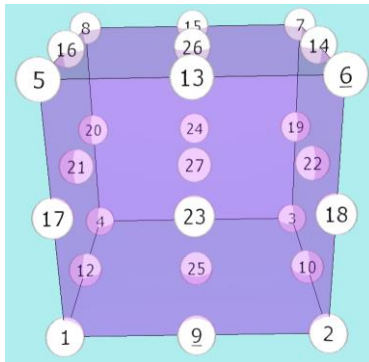
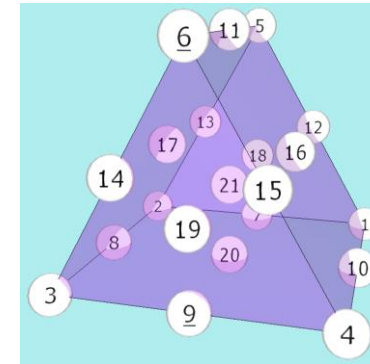
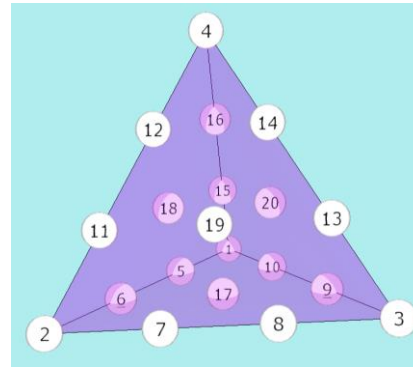
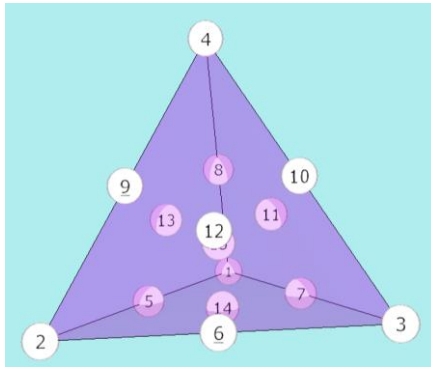
- Change *PART to *PART_INERTIA for body blocks option:
 - The values added into *PART_INERTIA such as total mass, inertia tensor components and coordinates of centre of mass are taken from the Body Blocks themselves using the 'Mass Prop' tool.
- Contacts between body blocks and seatbelt can now be created.
- An option has been added to change the input load keyword from *LOAD_NODE to *BOUNDARY_PRESCRIBED_MOTION_LOAD using a smooth curve.
- Section and Material IDs for seatbelts can be updated to be consistent with the input into the SBA tool.
- Option to re-number meshing start labels in seat belt settings panel.
- Option to use your own formers in main input panel.
- Default formers models improved.
- Some panel redesign and help images improved.

Higher order element improvements

Parabolic and cubic elements, shells and solids with interpolation nodes

High order elements – Orient > Reflect

- When mirroring these elements, the interpolation nodes are now re-ordered, so they conform to the LS-DYNA manual, which will avoid warnings and errors.



High order element – Coating

- New option when coating high order solids.
- Now the interpolation nodes can be used to define the coating shells:

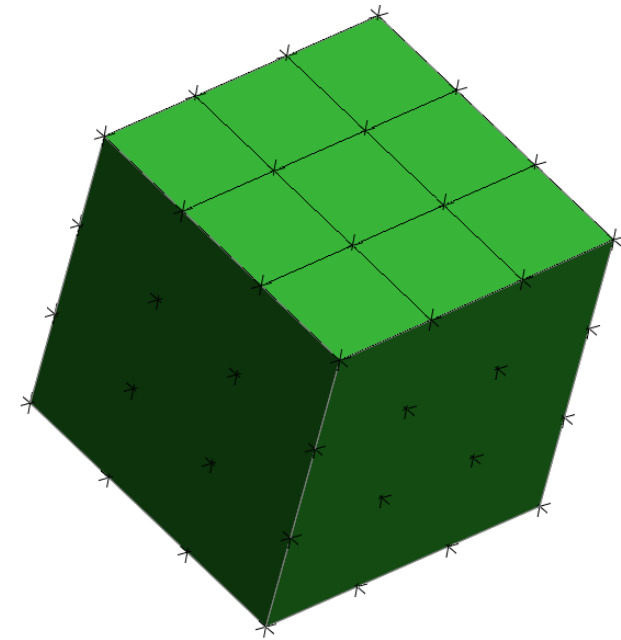
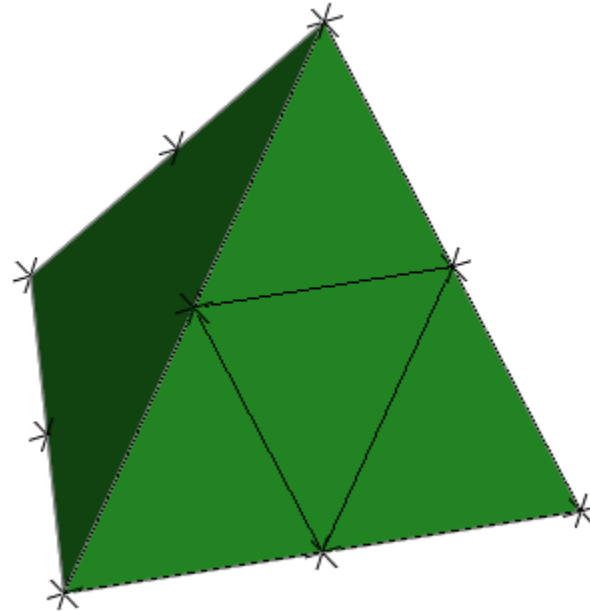
High order options ?

☐ Ignore int.nodes
☒ Use int. nodes

Tetra 10 options:

No of trias per face ?

☒ Four 1st order trias
☐ One 2nd order tria



Occupant Positioning

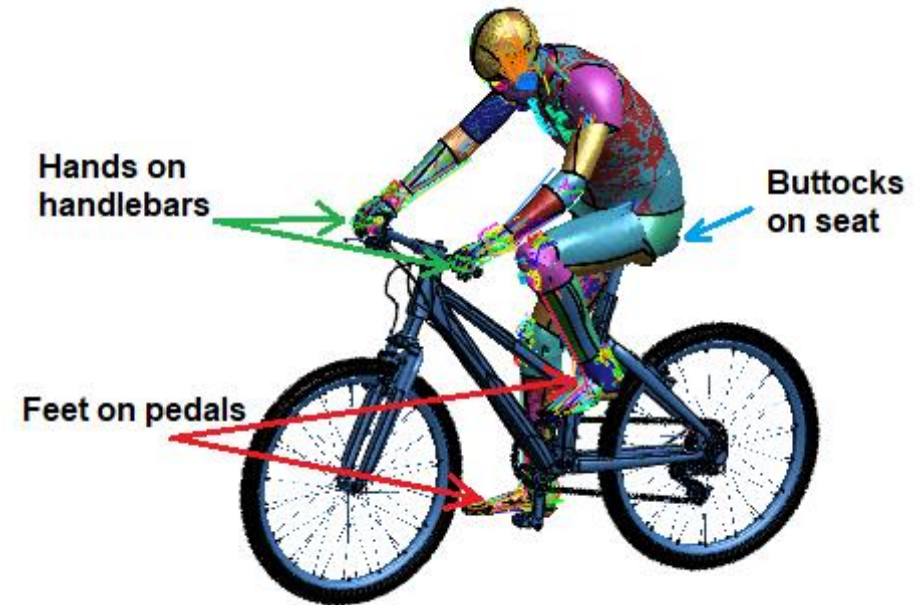
Image acknowledgements:

THUMS - Toyota Motor Corporation

GHBMCM - Elemance

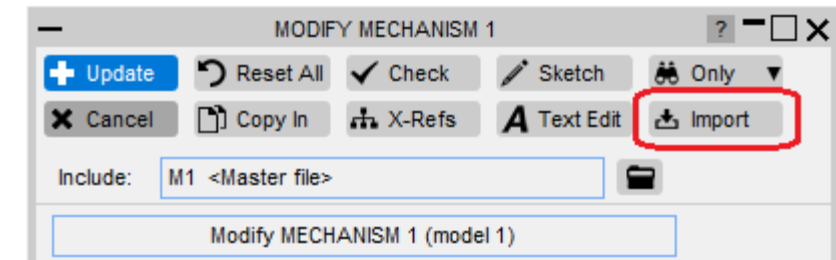
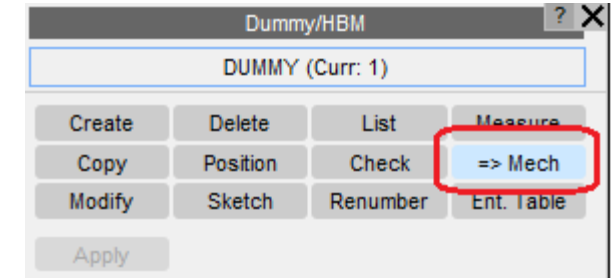
Export Dummy/HBM as Mechanism

- Historically it is possible to incorporate a dummy or human body model (HBM) as a child of a Mechanism.
- This is a limited one-way treatment in which the mechanism can drive the dummy/HBM only at a single point and is not adequate when mechanism and dummy/HBM need to interact fully.
- For example to position a dummy/HBM on a bicycle such that pedals and feet move together and hands move with handlebar it is necessary for the dummy/HBM to be a fully integrated mechanism whose assemblies are siblings with, rather than children of, those in the bicycle mechanism.



Export Dummy/HBM as Mechanism (ctd)

- There are two ways of doing this:
 1. Export using the “=> Mech” button on the Dummy/HBM panel
This creates a new free-standing Mechanism definition
 2. Import into a new or existing Mechanism
This converts the Dummy/HBM to a mechanism and adds it to the existing mechanism assemblies and joints. (It is also possible to import existing Mechanisms in the same way.)



- When first imported the Dummy/HBM assemblies will co-exist with existing Mechanism ones but no connections between the two will exist, these need to be added manually.

Export Dummy/HBM as Mechanism (ctd)

- A Dummy/HBM is converted to a Mechanism as follows:
 - The Parts and Nodes making up a Dummy/HBM Assembly are preserved verbatim as the equivalent Mechanism Assembly. Restrained degrees of freedom are also preserved.
 - Connections permitting rotation about 2 or 3 axes (typically hip and ankle joints) are converted to Mechanism Pin joints. If stop angles are defined via a Constrained Joint Stiffness card this will also apply to the Mechanism Pin joint (a new capability for Mechanisms in PRIMER 20.0).
 - Connections which only allow rotation about a single axis are converted to Mechanism Hinge joints. If stop angles are defined these will be converted to the existing Hinge rotation limiting angles.
 - The “parent/child” relationship of assemblies defined in the Dummy/HBM tree is lost: once it becomes a Mechanism all assemblies are equal siblings.
- Conversion leaves the original Dummy/HBM definition intact, a new Mechanism is created. The new definition can only be positioned within the Mechanism positioner.

Visualisation entities for Human Body Models (HBMs)

Image acknowledgements:

THUMS - Toyota Motor Corporation

GHBM - Elemance

Visualisation entities for HBMs

- The ability to create “visualisation entities” for Dummy/HBMs has been introduced to make it easier to view and manipulate these models.
- These visualisation entities are created to break up HBM entities into:
 - Dummy/HBM tree assemblies: pelvis, thorax, head, legs, arms etc.
 - HBM anatomy: skin, skeleton, flesh, organs, NULL-shells etc.
- The visualisation entities are created from the “Visualisation table”, available from within the Safety->Dummies tools panel.
- The table can be used to manipulate the visibility, colour, transparency etc. for these entities.
- Visualisation entities are not LS-DYNA entities and are temporary that are valid only in the ongoing PRIMER session.

Visualisation entities for HBMs

5. Specify the type of Dummy/HBM

Create Entities for:
THUMS Model
GHBMC Model
Standard Dummy

6. Press this to create HBM visual entities

1

2

3

4

7. Select to show the list of assembly or anatomy entities

8. Only/Blank/Unblank the visual entities you wish to see in the graphics area

Title	Colour	Transparency	Plot Mode	Stippled	Actions	Only	Blank	Unblank
Outer Skin Shells	Orange	Trans 70%	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Solid Skeleton	Grey	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Position Beams	Red	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Inner Skin Shells	Green	Trans 70%	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Shell Skeleton	Blue	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Outer Solid Flesh	Red	Trans 70%	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Inner Solid Flesh	Blue	Trans 70%	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Muscle Beams	Green	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Seat-Belt and Slings	Black	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
Foam	Yellow	Trans 70%	Shaded	Solid (def)	Actions	Only	Blank	Unblank

Visualisation entities for HBMs

Switch from model parts view to the "Assembly" or the "Anatomy" view

Save the view in the "View properties" (.prp) format

Use these popup menus to change visual properties of visual entities

PRIMER 20.0 - 64 bit

File Keywords Tools Display Images Viewing Options Help

PRIMER: M1: THUMS AM50 Occupant Model Version 6.1.2.2003

Visualisation entities table of DUMM1 in Model1

Dismiss Save View Properties

Dummy/HBM type: THUMS Model Create visual entities

☐ Enable parts view
☒ Enable assemblies view
☐ Enable anatomy view

☐ Show assembly list
☐ Show anatomy list

Visualisation entities of DUMM1 in Model1	Colour	Transparency	Plot Mode	Stippled	Actions	Only	All	Blank All
<input checked="" type="checkbox"/> PELVIS Assembly	Blue	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> T8-T1 THORAX Assembly	Red	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> HEAD Assembly	Blue	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> R UPR LEG Assembly	Cyan	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> R LWR LEG Assembly	Magenta	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> L UPR LEG Assembly	Yellow	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> L LWR LEG Assembly	Pink	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> L FOOT Assembly	Green	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank
<input checked="" type="checkbox"/> R UPR ARM Assembly	Purple	Opaque	Shaded	Solid (def)	Actions	Only	Blank	Unblank

When you drag an assembly, all the entities associated with the assembly are dragged

Assigning H point of Dummy 1 to assembly 1: PELVIS
Successfully copied model 2 to 1
Dimension tests determine model 1 units to be MM. See units panel for test configuration

Manual CT SI Node plot AC Zoom CN Save P Lock
Stop Timestep Init Vels (Tr) AC Zoom CN Save P Lock
Tidy +XY +YZ +XZ +ISO -XY -YZ -XZ -ISO R Views Rev Ent

Tools Mesh tools Data Post
Volumes I & II Volume III
Model Part tree Options Dummy/HBM
M1:THUMS_AM50_V61_Occupant_202003_data_for_primer_v3.key
Dummy/HBM
Curr: M1/DUMM1. H-pt: -102.0 0.0 -60.0
Position parts
Accept Reject Global Accuracy 1.0 ?
Rotate angles Drag assembly Move points Explain Options...
Visualisation table ?
Assembly Cont Lock Lock for
1: PELVIS Tall x y z Rall x y z
2: T8-T1 THORAX Tall x y z Rall x y z
4: HEAD Tall x y z Rall x y z
5: R UPR LEG Tall x y z Rall x y z
6: R LWR LEG Tall x y z Rall x y z
7: R FOOT Tall x y z Rall x y z
8: L UPR LEG Tall x y z Rall x y z
9: L LWR LEG Tall x y z Rall x y z
10: L FOOT Tall x y z Rall x y z
11: R UPR ARM Tall x y z Rall x y z
12: R LWR ARM Tall x y z Rall x y z
13: R HAND Tall x y z Rall x y z
14: L UPR ARM Tall x y z Rall x y z
15: L LWR ARM Tall x y z Rall x y z
16: L HAND Tall x y z Rall x y z
18: L SHOULDER Tall x y z Rall x y z
19: R SHOULDER Tall x y z Rall x y z
20: L5 Tall x y z Rall x y z
21: L4 Tall x y z Rall x y z
22: L3 Tall x y z Rall x y z
23: L2 Tall x y z Rall x y z
24: L1 Tall x y z Rall x y z
25: T12 Tall x y z Rall x y z

HBM Trees

Image acknowledgements:

THUMS - Toyota Motor Corporation

GHBMC - Elemance

Supported GHBMC models – detailed

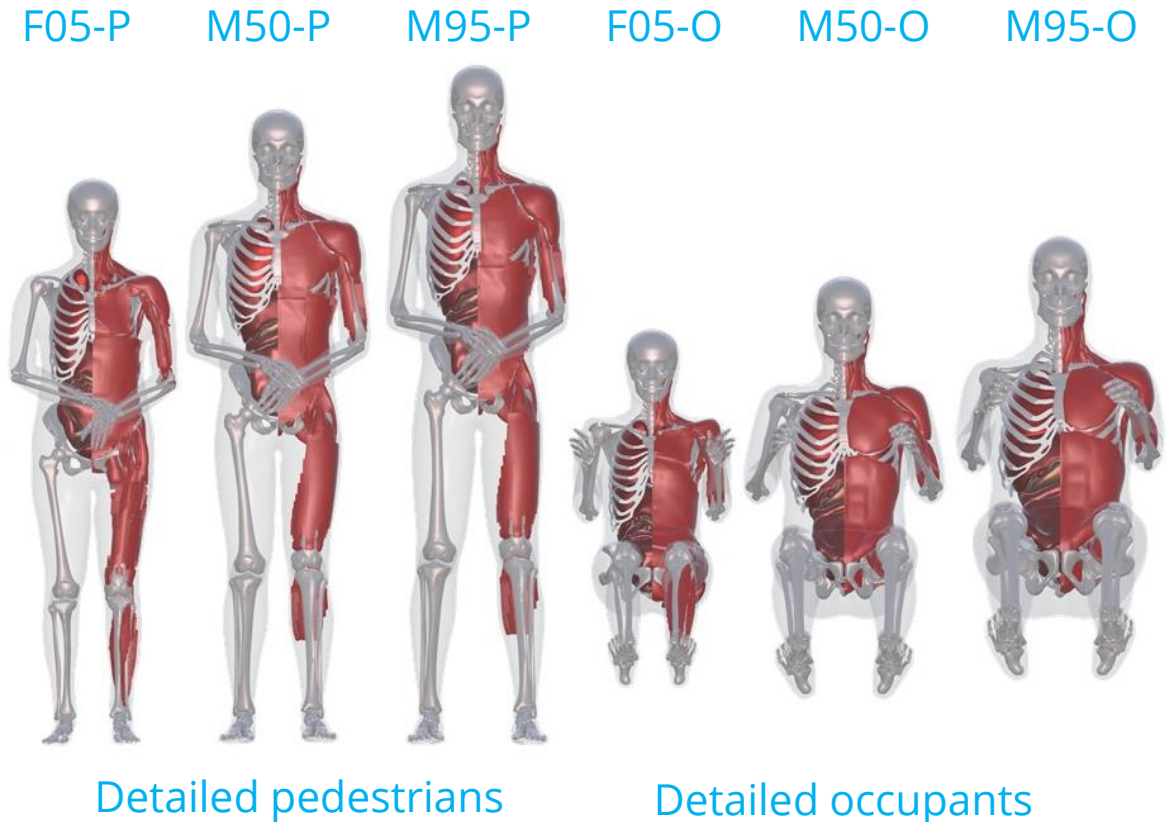
- Tree files help position and prepare HBMs for LS-DYNA analysis.
- These are available for the most popular GHBMC models.
- Tree files are free to PRIMER users and available from your local Oasys distributor.

- **Detailed occupants:**

- F05-O v5.1/**v6.0**
- M50-O v5.1/v5.1.1/**v6.0**
- M95-O v5.1/**v6.0**

- **Detailed pedestrians**

- F05-P v1.2/**v5.3.4**
- M50-P v1.6/**v5.3.1/v5.3.4**
- M95-P v1.2/**v5.3.4**



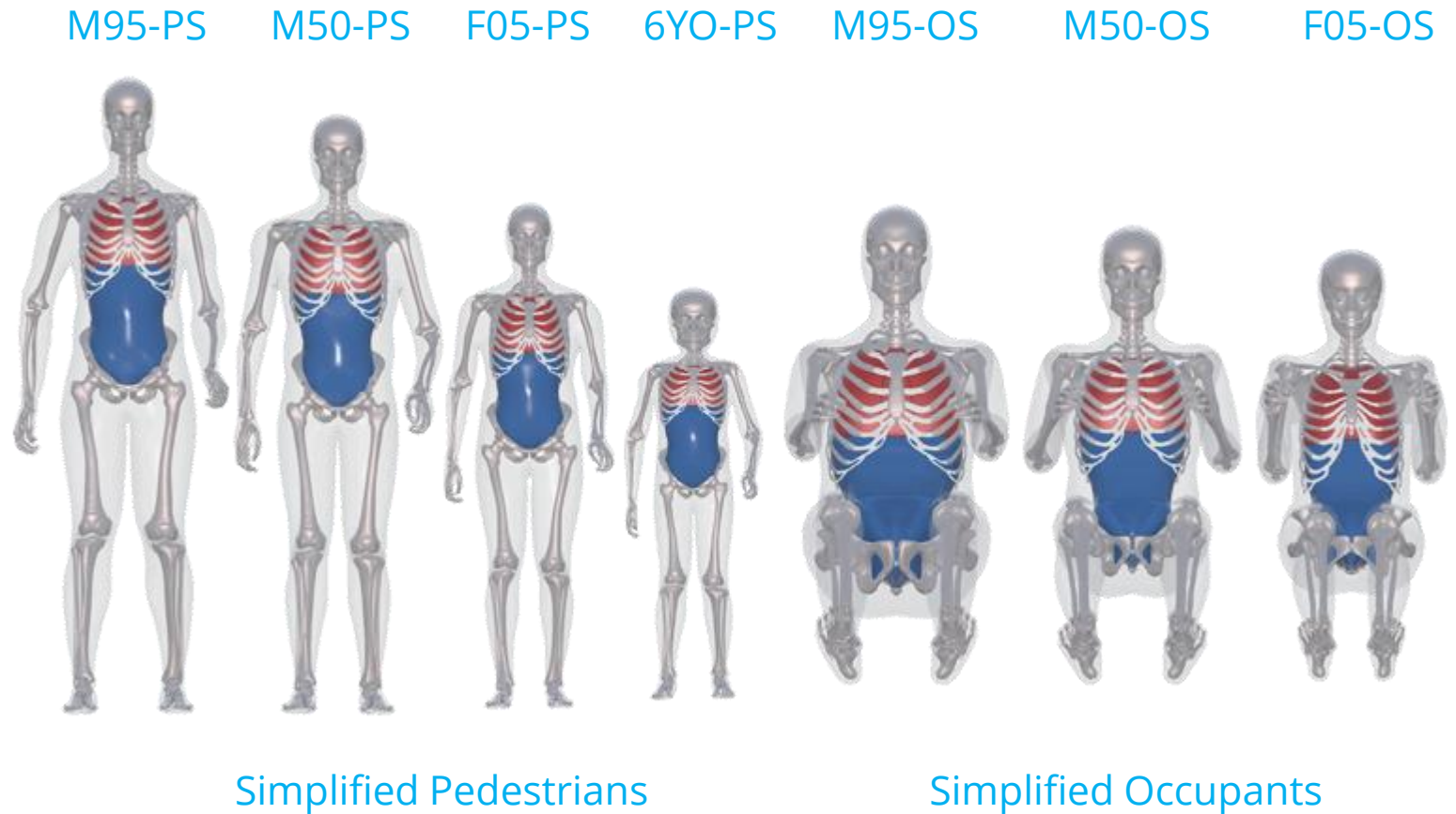
Supported GHBMC models – simplified

- **Simplified occupants:**

- F05-O v2.3
- M50-O v2.3
- M95-O v2.3

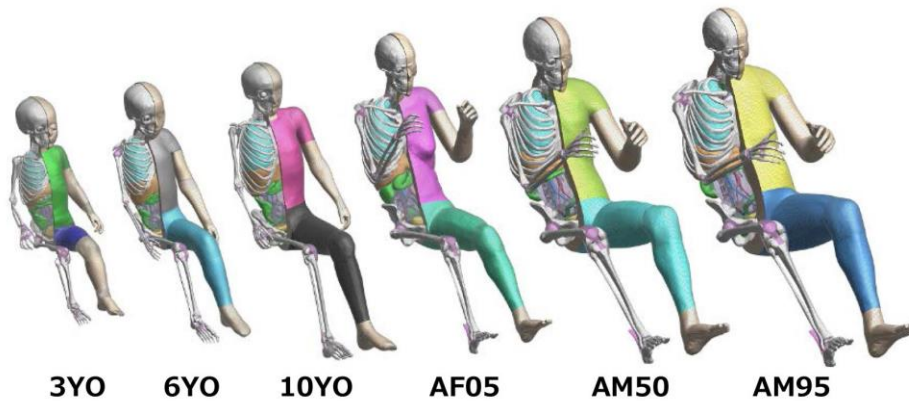
- **Simplified pedestrians:**

- F05-P v1.8/**v5.3.4**
- M50-P v1.8/**v5.3.4**
- M95-P v1.8/**v5.3.4**
- 3YO-P v1.8
- 6YO-P v1.8
- 10YO-P v1.8



Supported THUMS models

- Tree files are available for the most popular THUMS models.
- Tree files are free to PRIMER users and available from your local Oasys distributor.

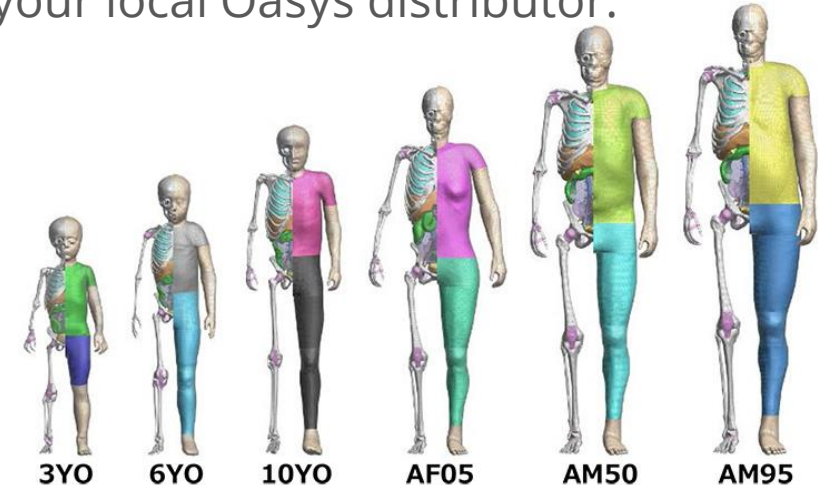


• Version 4 Pedestrians:

- AF05-P v4.0.2
- AM50-P v4.0.2
- AM95-P v4.0.2
- 3YO/6YO/10YO v4.0

• Version 4 Occupants:

- AF05-O v4.1
- AM50-O v4.1
- AM95-O v4.1
- 3YO/6YO/10YO v4.0



• Version 5:

- AF05-O v5.0.3
- AM50-O v5.0.3
- AM95-O v5.0.3

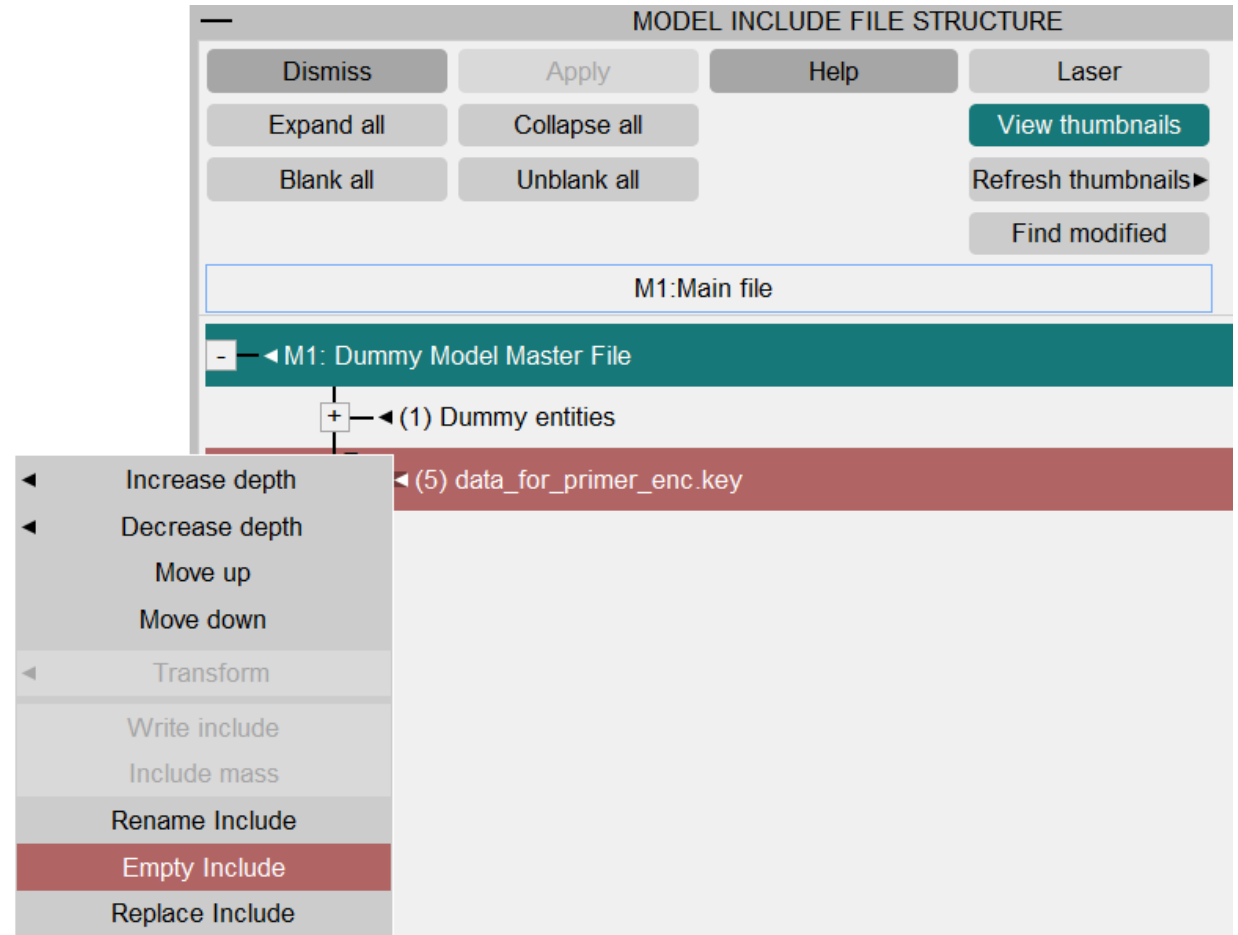
• Version 6:

- AF05-O v6.1
- AM50-O v6.1
- AM95-O v6.1

Further HBM Tool updates

Add encrypted include files into the keyword deck from PRIMER

- It is now possible to add “Encrypted” dummy/HBM tree include files into the keyword deck from within the PRIMER "Include Tree" panel.
- We have also enabled certain useful popup menus for encrypted include files in this panel.



Add part/node sets to encrypted HBM assemblies

- It is now possible to add in additional *SET_PART or *SET_NODE objects to an existing HBM assembly, even for an encrypted model.
- The panel shown can be accessed through Tools -> Safety -> Dummies -> Modify -> Edit (Assemblies).
- Already encrypted information will not be shown, but new sets will be displayed.
- Add to the assembly by using the green dropdown boxes displayed on the right.

MODIFY DUMMY MECH ASSEMBLY 1 (for DUMMY)

+ Update Reset All Check Sketch Only

Cancel Copy In X-Refs Text Edit

Modify DUMMY MECH ASSEMBLY 1 (model 1)

Label	Title
1	PELVIC

Part Sets	S_PT 1	S_PT 2	S_PT 3	S_PT 4
Add...	<encrypt>			
Remove				
Sketch...				

Parts	P 1	P 2	P 3	P 4
Add...				
Remove				
Sketch...				

Node Set	S_NO 1	S_NO 2	S_NO 3	S_NO 4
Add...	<encrypt>			
Remove				
Sketch...				

List of part sets contacted during positioning

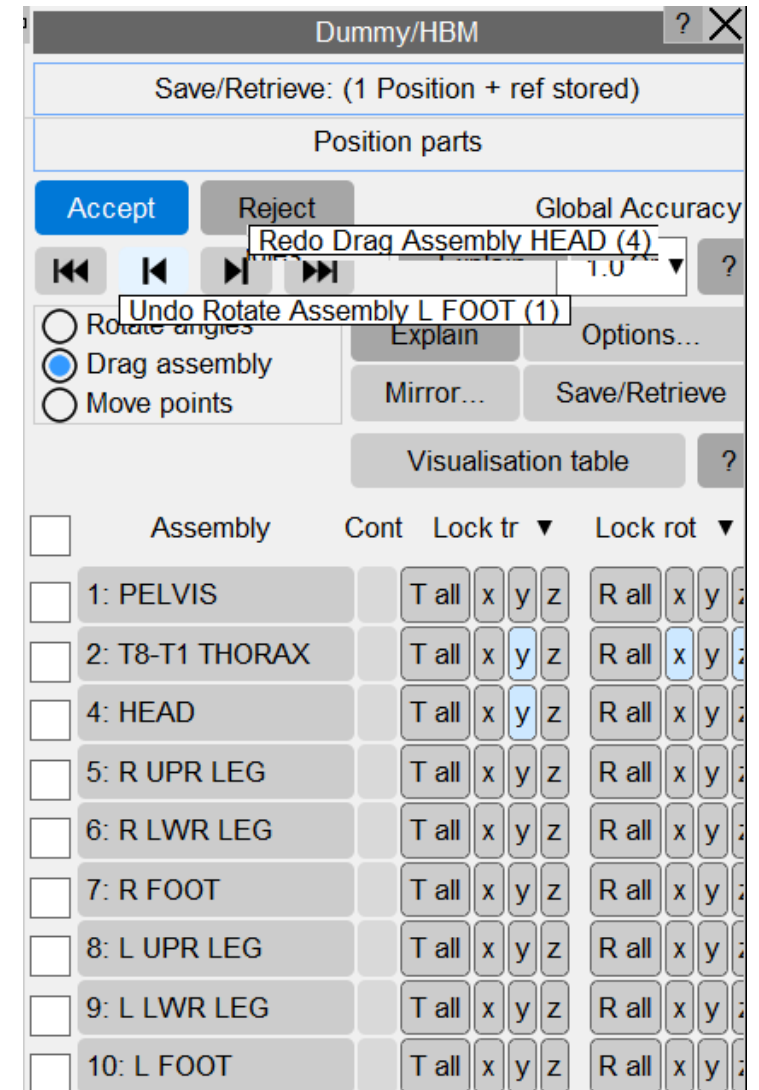
Del	Part set	Box	Tk Factor	Active	Explain
X					Sketch
X					Sketch
X					Sketch

List of child assemblies attached to this one

Del	Assembly	Jnt Stiff	Node A	DoF codes	Node B
X	<encrypt>	<encrypt>	<encrypt>	<encrypt>	<encrypt>
X	<encrypt>	<encrypt>	<encrypt>	<encrypt>	<encrypt>
X	<encrypt>	<encrypt>	<encrypt>	<encrypt>	<encrypt>

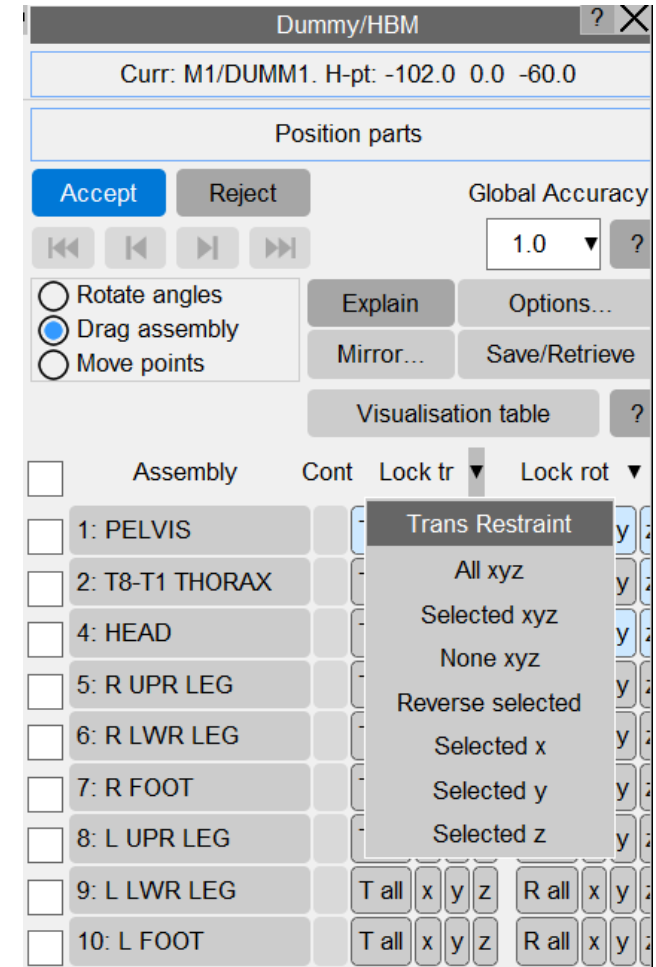
Undo/Redo the dummy assemblies positions

- You can now perform Undo/Redo for assembly positions while positioning the Dummy/HBM in the “Position parts” panel.
- You can also perform “**Undo all**” to go back to the original position or “**Redo all**” to the last positioned orientation of the Dummy/HBM.
- The Undo/Redo of the assembly positions work for all these positioning modes:
 - Rotate angles;
 - Drag assembly;
 - Move points.



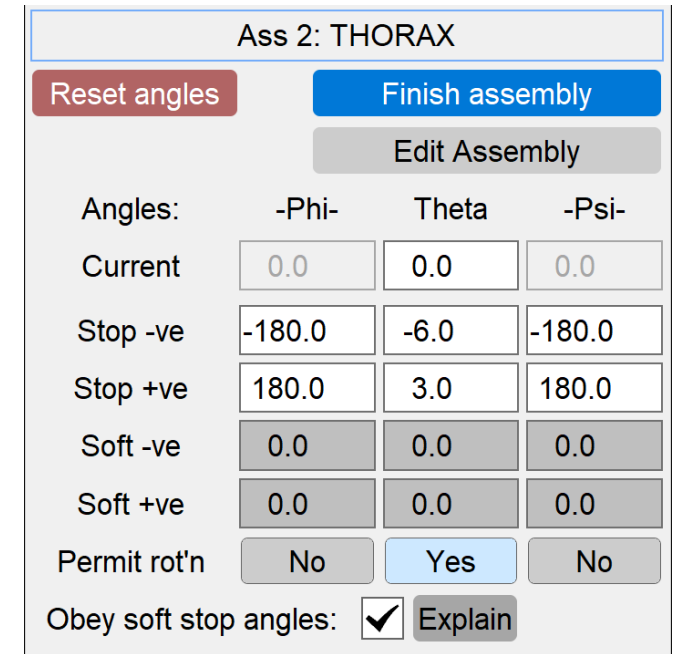
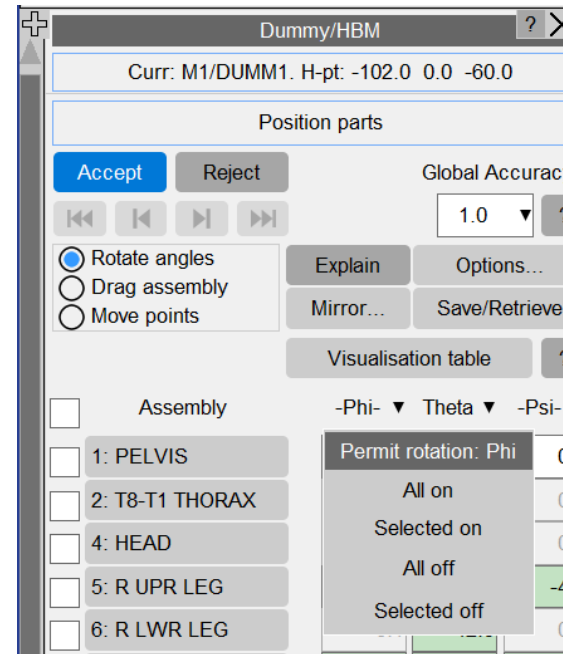
Select and modify assemblies restraints

- You can now select assemblies via tick buttons and manipulate the translation and rotation constraints for multiple assemblies via the “Trans Restraint” and “Rotate Restraint” popup menu buttons.
- Popup menus found by clicking on the ‘Lock tr’ and ‘Lock rot’ text boxes.
- These popups can also be found in the Mechanism positioning panel.



Select and modify assemblies rotation

- You can now select assemblies via tick buttons and manipulate the rotation axes through the “Permit rotation” menu buttons.
- To control these options for **all/selected** assemblies, new popups have been added in the main positioning panel under the Phi, Theta, Psi buttons.



Select and modify cable attachments attributes for assemblies

- You can now select assemblies via tick buttons and do the following on selected assemblies:
 - Sketch assembly with cable nodes;
 - Mark assembly to 'Ignore cable';
 - Mark assembly as rigid;
 - Remove cable attachment nodes.

The screenshot shows the 'Dummy/HBM' dialog box with the title bar containing a question mark and a close button. The main title is 'Select nodes and rigidify assemblies'. Below the title are buttons for 'Cancel', 'Previous', and 'Next'. There is a section for 'Automatically determine nodes' with a 'Help' button. Below that are two checkboxes: 'Ignore parts below mass:' (checked) with a value of '1.0E-5', and 'Consider encrypted materials' (unchecked) with a question mark. To the right of these are three radio buttons: 'FREE' (selected), '1node', and '2node'. Below these are three dropdown menus: 'Assembly actions', 'Node 1', and 'Node 2'. The 'Assembly actions' dropdown is open, showing options: 'Sketch', 'All', 'Selected', 'Ignore cables', 'All', 'Selected', 'None', 'Reverse selected', 'Rigidify assemblies', 'All', 'Selected', 'None', and 'Reverse selected'. The 'Node 1' dropdown is set to '51185' and the 'Node 2' dropdown is set to 'Remove node 2'. Below these are three columns of node IDs. The first column has values: 51185, 05982, 26655, 103644, 071687, 000007, 103633, 071478, 000007, 007055, 002911. The second column has values: 209002, 031655, 098991, 005893, 031945, 098845, 005692, 006469, 003980. The third column has values: 204791, 072673, 002005, 008645, 072508, 001834, 006952, 006189, 002446.

Assembly actions	Node 1	Node 2	Node 3
Sketch	51185	Remove node 2	
All	05982	All	
Selected	26655	Selected	
Ignore cables	103644	031655	072673
All	071687	098991	002005
Selected	000007	005893	008645
None	103633	031945	072508
Reverse selected	071478	098845	001834
Rigidify assemblies	000007	005692	006952
All	007055	006469	006189
Selected	002911	003980	002446
None			
Reverse selected			

Mirror Dummy/HBM Position

- We have added the option to mirror the position of a dummy/HBM, making it easier to have a reflected position in the arms and legs.
- Accessed via Safety -> Dummies -> Position -> Move Parts -> Mirror...
- This will mirror assemblies which have an opposite matching 'sibling' assembly (more on 'siblings' in following slides).
- The axes for reflection vary between different model setups, a few common configurations have been placed under the 'Dummy/HBM types' dropdown button.
- Select 'Custom' type to access the axis tick boxes and create a custom setting, see manual for guidance.
- Default settings for the axes options can be saved via preferences.

+

Dummy/HBM

Curr: M1/DUMM1. H-pt: 1000.0 0.0 0.0

Save to oa_pref file Done ?

Dummy/HBM types: Default

Select axes to negate angles in:

Phi: ☒

Theta: ☐

Psi: ☒

☒ Mirror All Assemblies

☐ Mirror Selected Assembly

Source: <none>

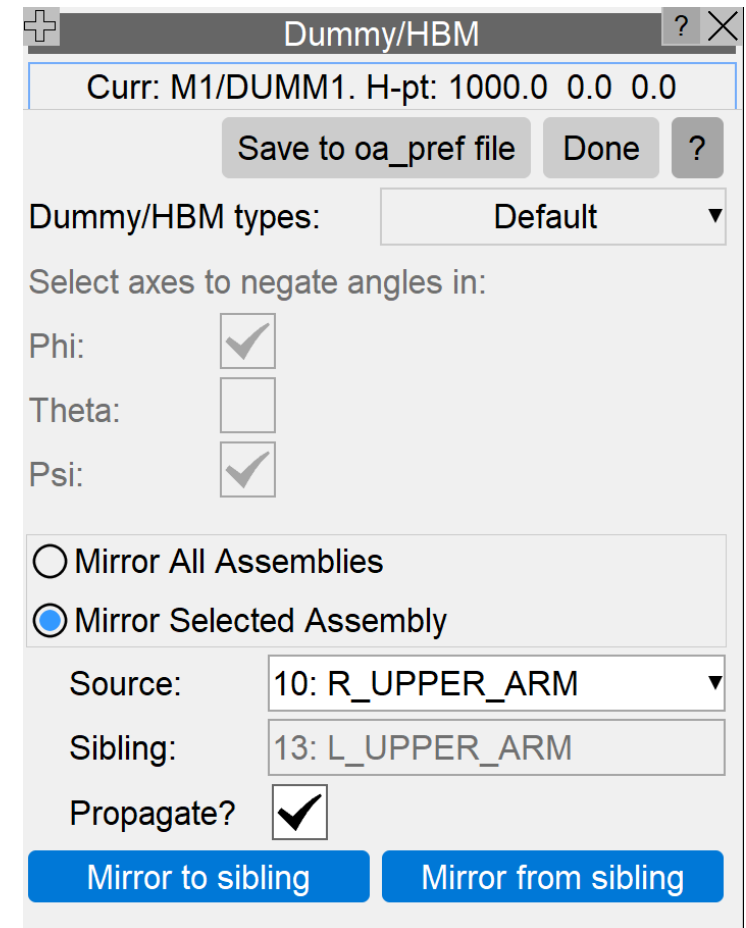
Sibling:

Propagate? ☒

Mirror L->R Mirror R->L

Mirror Dummy/HBM Position

- In this panel, you can either:
 - Mirror the whole assembly (any assembly with an opposing 'sibling');
 - Mirror a selected assembly.
- The latter option allows you to select a source assembly; here you can either mirror the angles from the source's sibling to itself, or vice versa.
- The 'Propagate' button determines whether the source's child assemblies are also mirrored. For example, if ticked for an upper arm assembly, then the lower arm and hand assemblies will also be mirrored.



The screenshot shows a dialog box titled "Dummy/HBM". At the top, it displays "Curr: M1/DUMM1. H-pt: 1000.0 0.0 0.0". Below this are buttons for "Save to oa_pref file", "Done", and a help icon "?". The "Dummy/HBM types:" dropdown is set to "Default". Under "Select axes to negate angles in:", the checkboxes for "Phi:" and "Psi:" are checked, while "Theta:" is unchecked. There are two radio button options: "Mirror All Assemblies" (unselected) and "Mirror Selected Assembly" (selected). Below these, the "Source:" dropdown is set to "10: R_UPPER_ARM" and the "Sibling:" dropdown is set to "13: L_UPPER_ARM". The "Propagate?" checkbox is checked. At the bottom, there are two blue buttons: "Mirror to sibling" and "Mirror from sibling".

Mirror Dummy/HBM Position

- Opposing assemblies are automatically matched up, with the pairings accessible through the individual assembly panels (accessed in Safety -> Dummies -> Position -> Move parts).
- Here you can change the sibling pairings if needed and mirror individual assemblies.
- In this panel you can also perform the mirroring of selected assemblies, similar to the main Mirror panel.

Ass 4: R_UPPER_LEG

Reset angles

Finish assembly

Edit Assembly

Angles:	-Phi-	Theta	-Psi-
Current	0.0	0.0	0.0
Stop -ve	-40.0	-70.0	-45.0
Stop +ve	40.0	60.0	45.0
Soft -ve	0.0	0.0	0.0
Soft +ve	0.0	0.0	0.0
Permit rot'n	Yes	Yes	Yes
Obey soft stop angles:	<input checked="" type="checkbox"/>	Explain	
Sibling:	7: L_UPPER_LEG ▾		
Propagate?	<input type="checkbox"/>		
Mirror to sibling		Mirror from sibling	

Updates for after a Node Import

- If processing a node import through a dynain file read-in, then some updates may be required to the:
 - Sliprings (if the HBM contains seatbelts);
 - Parameters (for THUMS v5 onwards).
- Tick boxes to give you the option to automatically process these updates can be found in 3 places:
 - Node Import;
 - Safety -> Seatsquash -> Import;
 - Safety -> Dummies -> Position -> LS-DYNA -> Import.
- More details can be found in the following slides.

Node import

Help

Which data do you want to import (if present)?

Previous Apply

	Import	Delete
New nodal coordinates	<input checked="" type="checkbox"/>	
*INITIAL_STRESS_SOLID	<input type="checkbox"/>	<input type="checkbox"/>
*INITIAL_STRESS_SHELL	<input type="checkbox"/>	<input type="checkbox"/>
*INITIAL_STRESS_BEAM	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	<input type="checkbox"/>
	<input type="checkbox"/>	<input type="checkbox"/>
*INITIAL_VELOCITY_NODE	<input type="checkbox"/>	

☐ (Re)create *INITIAL_FOAM_REF_GEOM ?

☐ Remove existing *INITIAL_FOAM_REF View OFF

☐ Remove any IALEGP reference when importing *INITIAL_STRESS_SOLID card

☐ Update slipring belt IDs

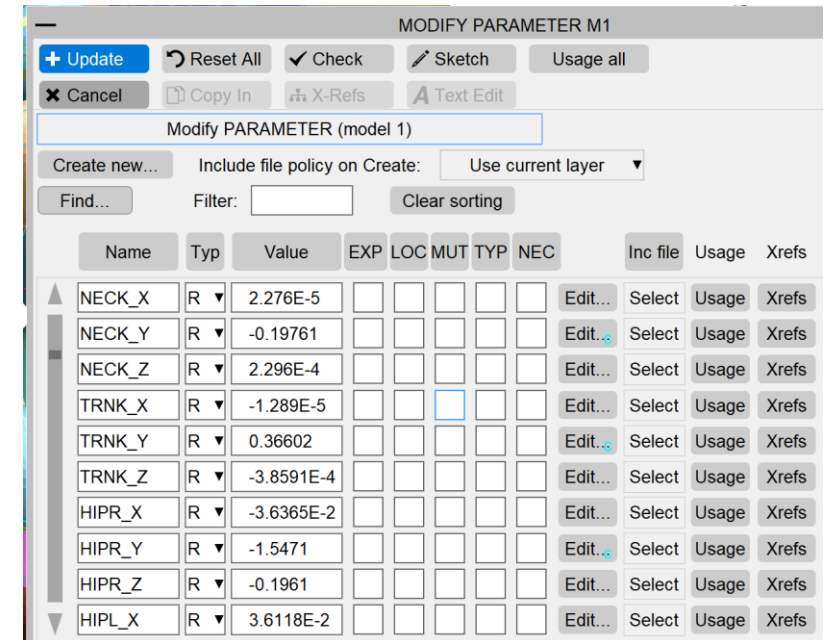
☐ Update THUMS joint angle *PARAMETERS

Updating Sliprings Seatbelts after a Node Import

- For a dummy/HBM that contains slipring and seatbelt elements, processing a node import (or reading a dynain file) can make the seatbelts pass through the slipring. The slipring has two variables referring to the seatbelts connected to it (sbid1 and sbid2), and previously these would not be updated, meaning that the connections were out of sync.
- Processing an update through the tick box displayed in the previous slide will fix all sliprings in the model. A preference has also been added for the default mode of this box: *dynain_fix_seatbelt_sliprings*.
- Singular sliprings can be updated via a Model Check by right clicking on the errors and selecting 'Autofix'.

Update Parameters after a Node Import

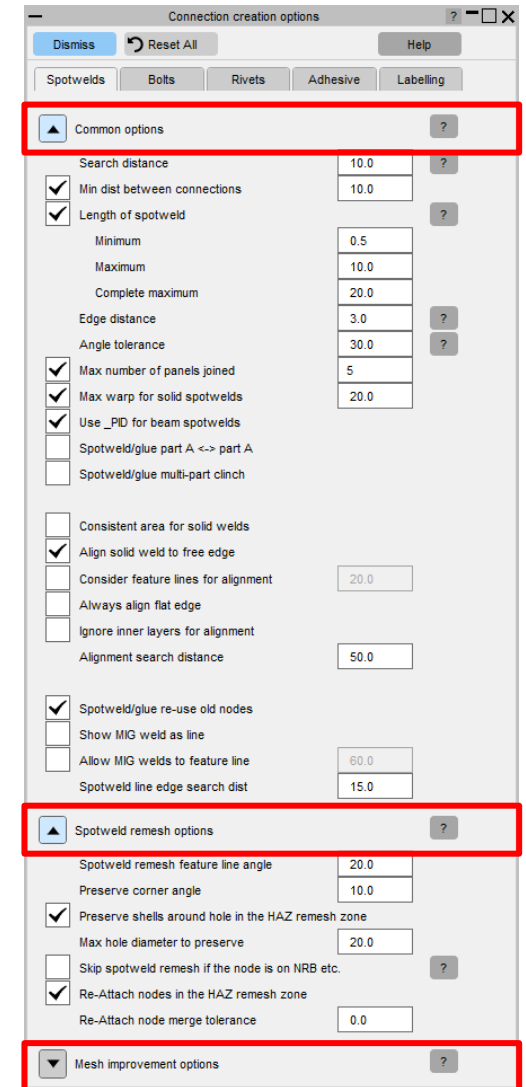
- Similarly, to the slipping updates, an update may be required to the parameters of an HBM after processing a node import.
- With THUMS models (v5 onwards) there are 48 parameters which represent the angles between joints on the HBM.
- If a node import is done (through a dynain read in), then the body parts may have moved, meaning the angles need to be updated.
- This can be updated through the tick box displayed in the previous slide. A preference has been added for the default option for this: *dynain_fix_thums_angle_params*.



Spotweld remesh enhancements

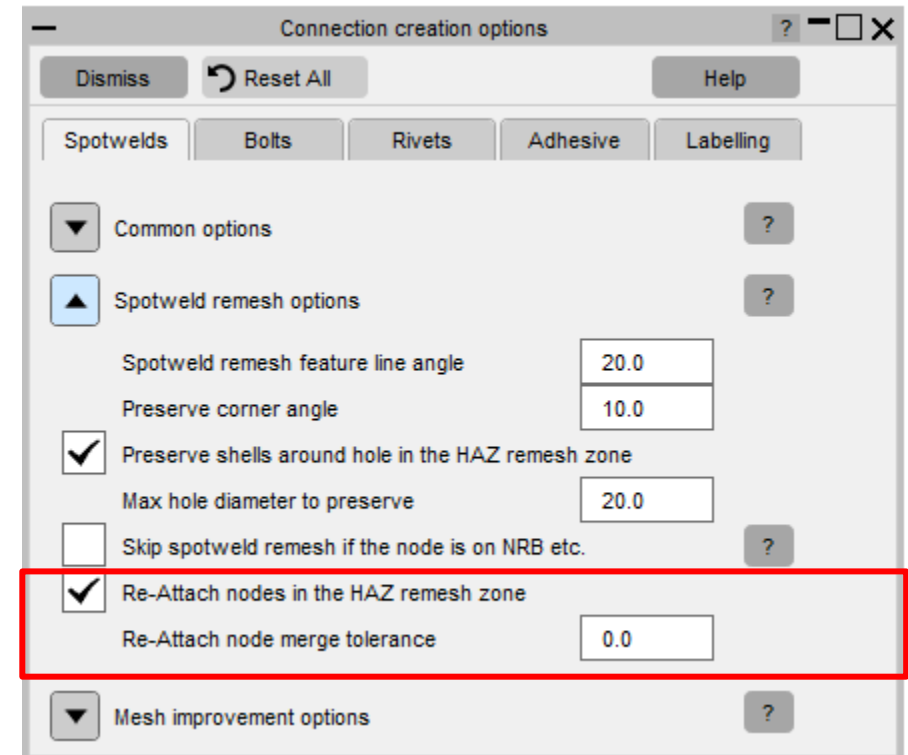
Reorganised spotweld options panel

- Spotweld options panel has grown significantly due to addition of mesh improvement options. Now, the options have been segregated and put under three expander toggle buttons "Common options", "Spotweld remesh options", and "Mesh improvement options".
- These toggle buttons can be used to expand or minimise the options improving user experience.
- Spotweld remesh related options have moved under "Spotweld remesh options" and mesh improvement related options have moved under "Mesh improvement options". All the remaining options are put under common options.



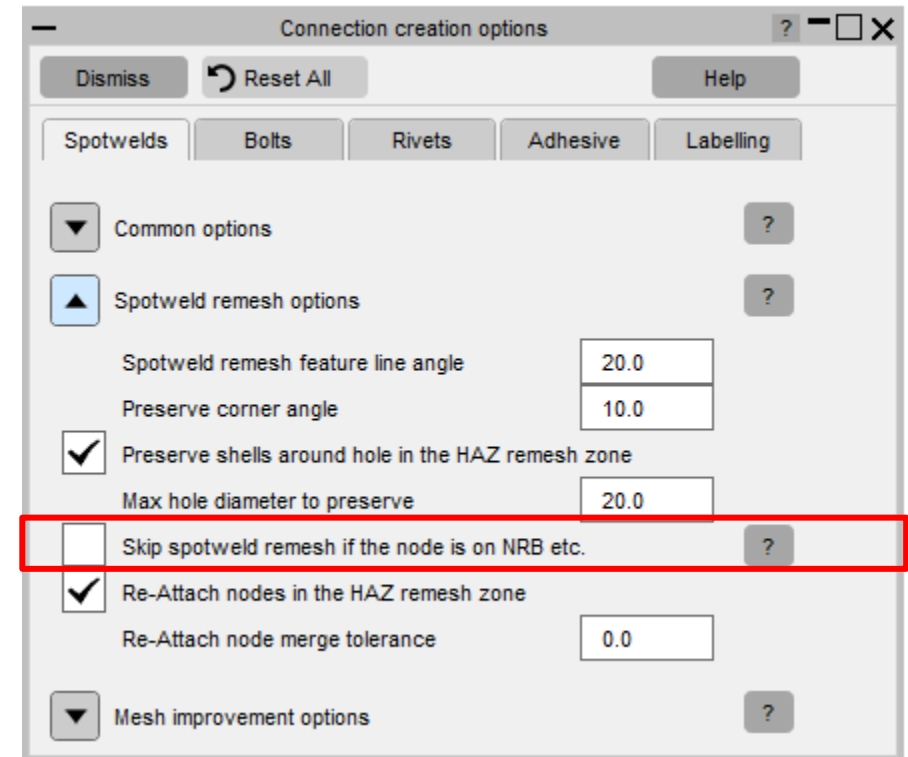
Re-attach nodes

- Constrained NRBs, Initial Load, Masses etc. were not preserved after performing the spotweld remeshing if close to spotwelds.
- Now an option “Re-Attach nodes in the HAZ remesh zone” is provided to re-attach nodes after remeshing to preserve Nodal Rigid Bodies, Initial Load, and masses etc.
- Enter tolerance value for node merge in “Re-Attach node merge tolerance” textbox. By default, it is set to zero.



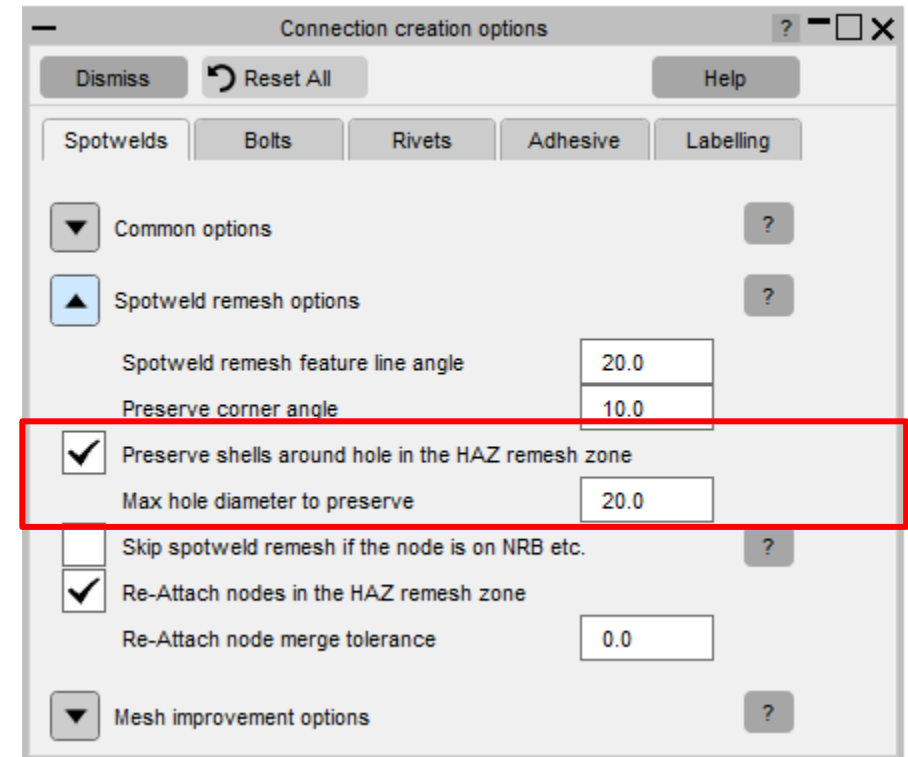
Skip spotweld remesh

- An option “Skip spotweld remesh if the node is on NRB etc.” is provided to prevent remeshing the spotwelds when too close to something like Nodal Rigid Body, Boundary SPC etc. which would lose nodes.



Preserve shells around hole

- In earlier versions, the elements around a hole were not preserved if they are in vicinity of a spotweld after remeshing.
- Now an option “Preserve shells around hole in the HAZ remesh zone” is added, and if turned ON, it will preserve the elements around a hole.
- Enter the hole diameter to be preserved in “Max hole diameter to preserve” textbox.



Preserve elements around hole

- When 'Preserve elements around hole' option is turned ON, PRIMER will preserve all the elements while remeshing which are present immediately around the hole. By specifying 'Max hole size' you can control which hole elements should be preserved. Edges of all such elements will be drawn in a cyan colour to show that they will be preserved.

Remesh area

Restart

Apply

Settings

Options

Add shells

Remove shells

Element size:

5.0

Ave

Feature line limit:

20.0

Fixed point angle:

45.0

Fixed points

Node

Edge

Free

Feature lines

Single

Feature

Lock

Preserve internal part boundaries :

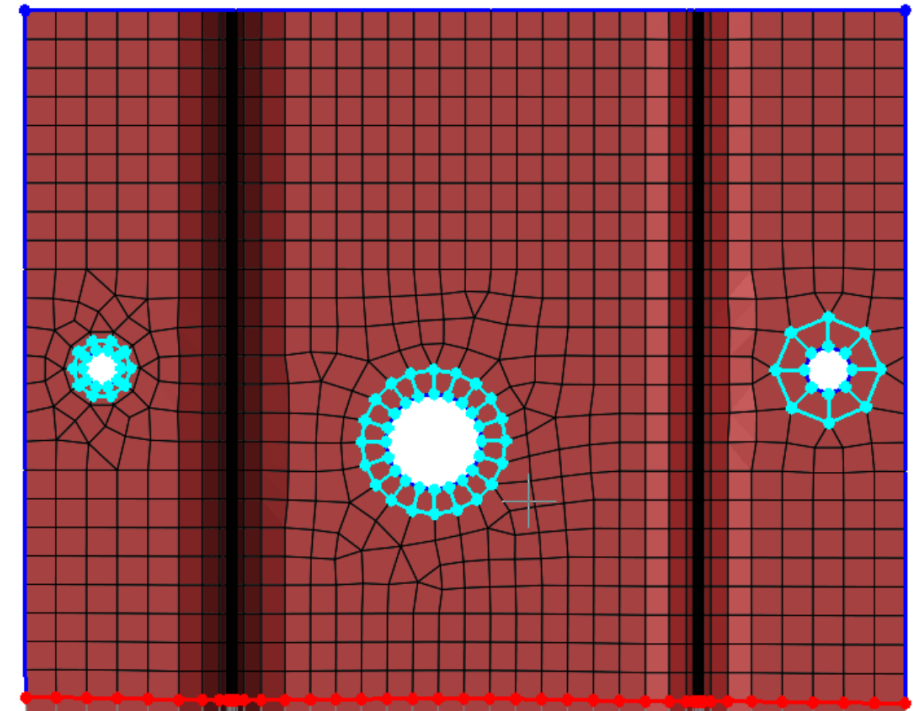
☒

Preserve elements around hole:

☒

Max hole size:

20.0



Mesh size for adhesive patch extruded from quad biased grid

- When 'Mesh size for adhesive patch' option is turned on, PRIMER will use the mesh size provided for this option for creating the adhesive patch, otherwise the average size will be used.

The screenshot shows the 'Connection creation options' dialog box with the 'Adhesive' tab selected. The 'Mesh size for adhesive patch' option is checked and highlighted with a red box, with a value of 5.0. Other options include 'Patch from quad biased grid' (checked), 'Spotweld/glue re-use old nodes' (checked), 'Max warp for adhesive solids' (checked), 'Max number of panels joined' (5), 'Max height of complete adhesive' (20.0), 'Minimum height' (0.5), 'Min dist between connections' (10.0), 'Angle tolerance' (30.0), 'Edge distance' (3.0), 'Maximum thickness' (10.0), 'Adhesive percentage made check' (50.0), 'Patch adhesive source angle tolerance' (30.0), 'Break angle' (30.0), 'Soft aspect ratio' (3.0), 'Hard aspect ratio' (5.0), 'Max adhesive layers' (2), 'Patch cohesive penta elements' (unchecked), 'Spotweld/glue multi-part clinch' (unchecked), 'Spotweld/glue part A <-> part A' (unchecked), and 'Max warp for adhesive solids' (checked).

Option	Value
Maximum thickness	10.0
Edge distance	3.0
Angle tolerance	30.0
<input checked="" type="checkbox"/> Min dist between connections	10.0
<input checked="" type="checkbox"/> Minimum height	0.5
Maximum height	10.0
Max height of complete adhesive	20.0
<input checked="" type="checkbox"/> Max number of panels joined	5
<input checked="" type="checkbox"/> Max warp for adhesive solids	20.0
<input type="checkbox"/> Spotweld/glue part A <-> part A	
<input type="checkbox"/> Spotweld/glue multi-part clinch	
<input checked="" type="checkbox"/> Spotweld/glue re-use old nodes	
Adhesive percentage made check	50.0
<input type="checkbox"/> Patch adhesive source angle tolerance	30.0
Break angle	30.0
Soft aspect ratio	3.0
Hard aspect ratio	5.0
Max adhesive layers	2
<input type="checkbox"/> Patch cohesive penta elements	
<input checked="" type="checkbox"/> Patch from quad biased grid	
<input checked="" type="checkbox"/> Mesh size for adhesive patch	5.0

Add nodes around spotweld to improve mesh quality

- When 'Add nodes around spotweld to improve mesh quality' option is turned ON, PRIMER tries to improve the quality of the mesh produced around spotweld by adding new nodes around it.

The screenshot displays the PRIMER software settings window. The 'Add nodes around spotweld to improve mesh quality' option is checked and highlighted with a red border. Below this option, a dropdown menu is open, showing various configurations for adding new nodes. The settings for 'HAZ mesh reconstruct' and 'HAZ tria reduction' are also visible and checked.

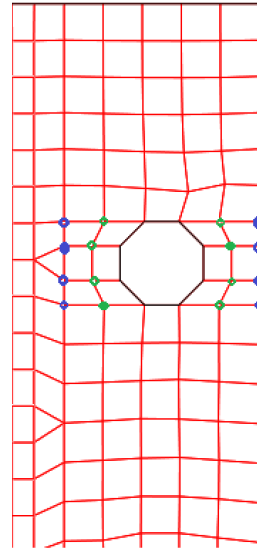
Option	Value
HAZ mesh reconstruct	0.0
HAZ mesh reconstruct element size	35.0
HAZ mesh reconstruct search dist	20.0
HAZ mesh reconstruct feature line	
HAZ tria reduction	
HAZ part replace options:	
Simple element replace	<input checked="" type="radio"/>
Full part replace	<input type="radio"/>
Add nodes around spotweld to improve mesh quality	<input checked="" type="checkbox"/>
Second feature line angle	2.0
Max flange width to skip adding nodes	25.0
Options for adding new nodes	4 nodes both

Options for adding new nodes:

- 4 nodes both
- 4 nodes only edges
- 4 nodes only middle
- 2 nodes both
- 2 nodes only edges
- 2 nodes only middle

Add nodes around spotweld to improve mesh quality

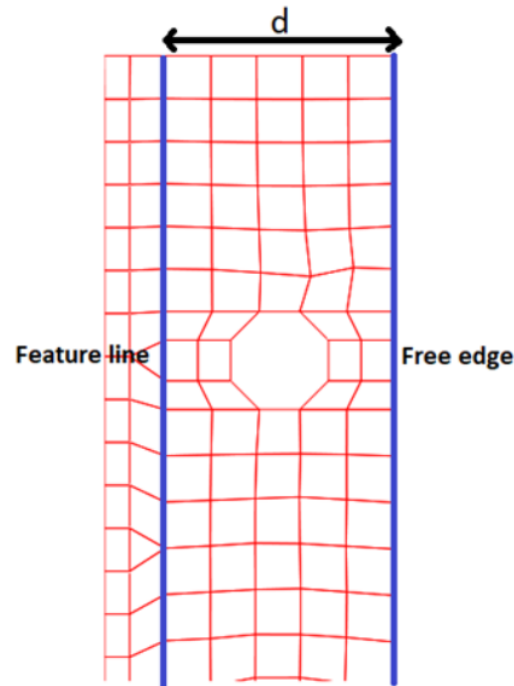
- The number of new nodes added per spotweld depends upon the local geometry and the type of spotweld remeshed. For example, in the image below, nodes marked in green, and blue have been added by PRIMER.



- There are various parameters associated with this option:
 - a) **Second feature line angle:** By specifying a suitable value for this option a larger area for remesh can be achieved leading to a better mesh quality;

Add nodes around spotweld to improve mesh quality

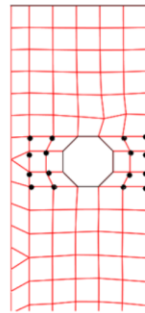
b) Max flange width to skip adding nodes: If distance specified here is less than the 'd' (flange width where spotweld is located.) then PRIMER will skip using this option for that spotweld. PRIMER calculates the flange width near each spotweld and if that is greater than the value specified here then it skips using this option for that spotweld. We recommend using 'Add nodes around spotweld to improve mesh quality' option for flanges which are less than 20mm wide.



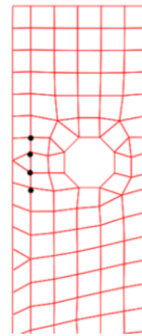
Add nodes around spotweld to improve mesh quality

c) Options for adding new nodes: There are six options by which you can specify how many nodes and at what location around the spotweld they will be added. Nodes marked in black below are the nodes which have been added while using the corresponding option.

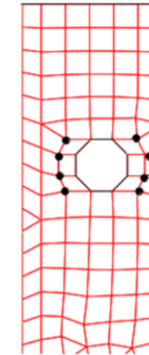
1) 4 nodes both:



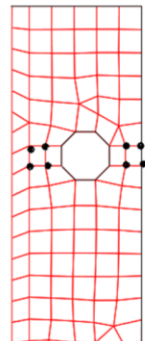
2) 4 nodes only edges:



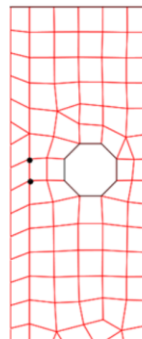
3) 4 nodes only middle:



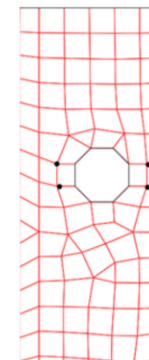
4) 2 nodes both:



5) 2 nodes only edges:



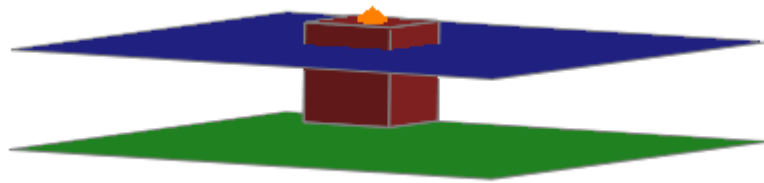
6) 2 nodes only middle:



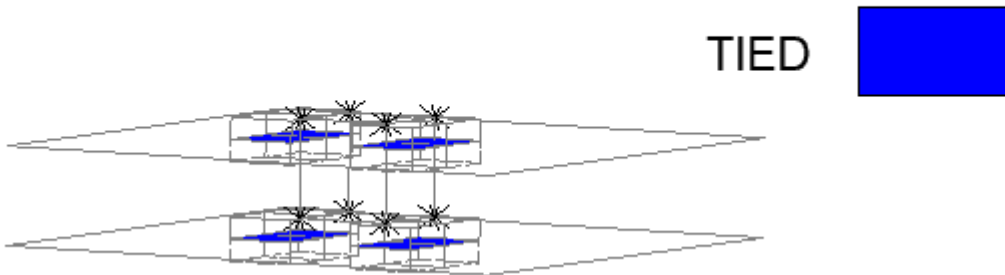
Connections

New out of plane check for weld/adhesive connection nodes

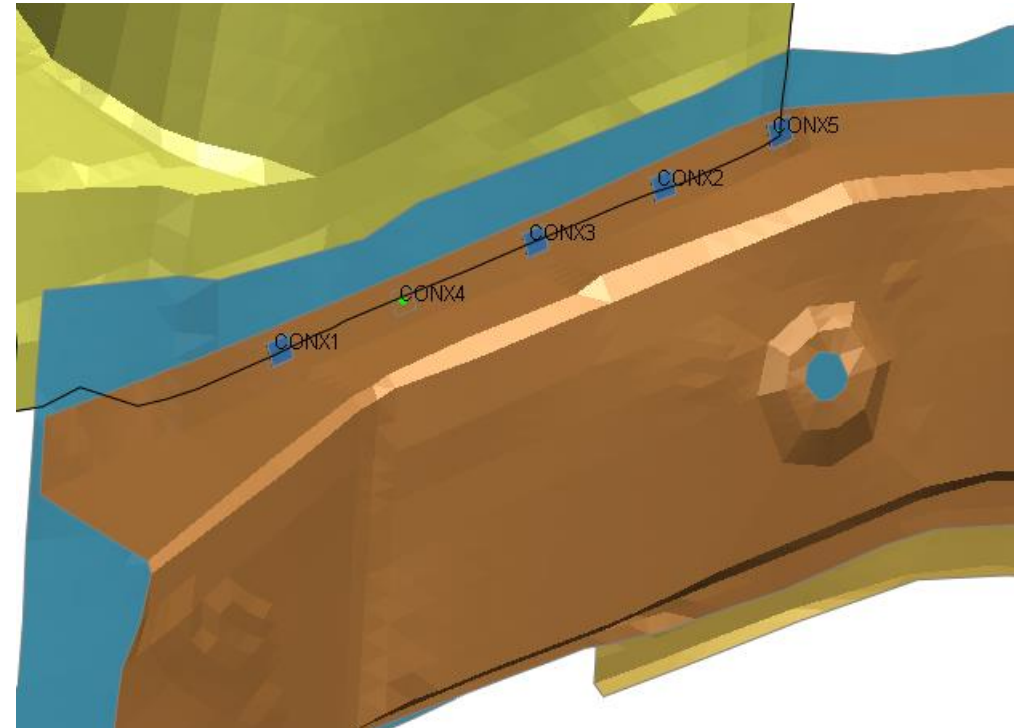
- Spotweld contact will allow considerable tolerance on the node to plane tied distance.



Contact connectivity



- Panel replacement without remake of attached welds can leave welds protruding.



New out of plane check for weld/adhesive connection nodes

- PRIMER 19.x conferred realized status if contact tied node to layer:

ID	Type	Status	Error	Details	Contact id
CNX1	SPOTWELD	Realized			C1
CNX2	SPOTWELD	Realized			C1
CNX3	SPOTWELD	Realized			C1
CNX4	SPOTWELD	Realized			C1
CNX5	SPOTWELD	Realized			C1

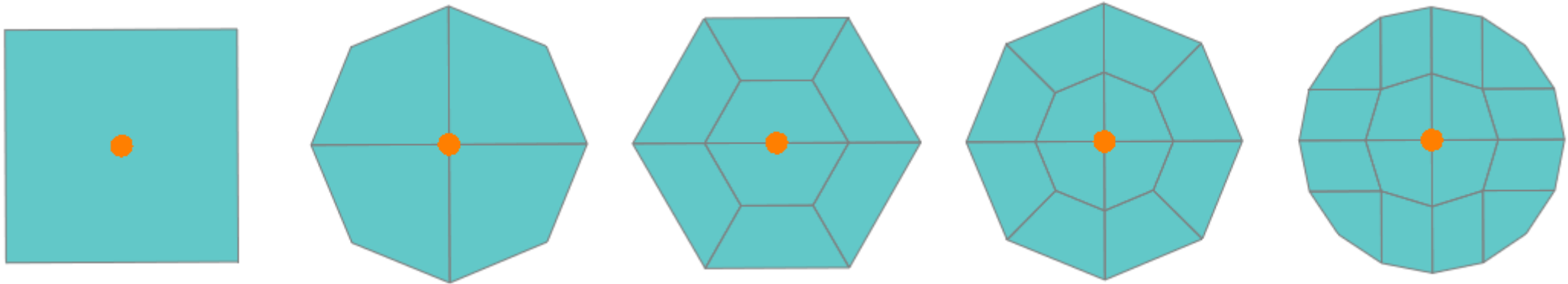
- PRIMER 20.0 requires BOTH contact tied AND nodes within layer plane:

ID	Type	Status	Error	Details	Contact id
CNX1	SPOTWELD	Invalid	OUT OF PLANE - nod	node 800019 is 0.995176 off plane of layer (exceeds tolerance of 0.1)	C1
CNX2	SPOTWELD	Invalid	OUT OF PLANE - nod	node 800033 is 0.993898 off plane of layer (exceeds tolerance of 0.1)	C1
CNX3	SPOTWELD	Invalid	OUT OF PLANE - nod	node 800044 is 0.993577 off plane of layer (exceeds tolerance of 0.1)	C1
CNX4	SPOTWELD	Invalid	OUT OF PLANE - nod	node 800077 is 0.996454 off plane of layer (exceeds tolerance of 0.1)	C1
CNX5	SPOTWELD	Invalid	OUT OF PLANE - nod	node 800064 is 0.996385 off plane of layer (exceeds tolerance of 0.1)	C1

- Tolerance is 0.01 x maximum thickness setting (default 0.01x10=0.1).

Area of multi-solid spotwelds

- Multi-solid welds have been updated in PRIMER 20.0 such that the definition of area is consistent with single solid welds:
 - The area is $\pi \cdot D \cdot D / 4$, where D = weld diameter.
- In PRIMER 19.x areas of multi-solid welds can vary.



Area of multi-solid spotwelds

- PRIMER 20.0 detects that an earlier version deck has been read and offers to auto-fix.

```
REMAKE ALL WELDS WITH CONSISTENT AREA SETTING  WELDS->TABLE

Model 1: taurus_orig
=====

Setting 'consistent area for spotwelds' is ON (default)

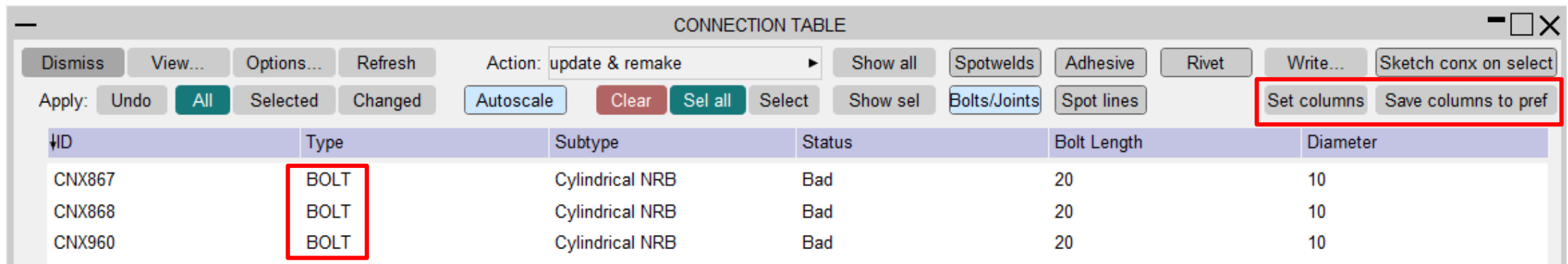
This old format input deck has 950 solid multi-hex spotwelds which have cross-sectional area
less than the expected value of  $\pi \cdot d^2 / 4$ . Worst case is reduction of 17%.
These connections may require remake with consistent area flag set TRUE.

REMAKE ALL WELDS will remake all multi-hex welds with the consistent area setting (recommended)
WELDS->TABLE      will display the relevant connections if there is a single active model
DONT WARN AGAIN   no action, this panel will not appear again for this model, setting remains true
                  and will apply to any newly created spotwelds
SETTING OFF       no action and the setting is false for this and future sessions (home pref)

You can update these connections manually on the table at any time.
Any connection that is remade will lose this warning (whether saved setting is true or false)
```

New options for display of columns on connection table

- Requires that all displayed connections are of same type (spotwelds, bolts, etc).
- 'Save columns to pref' tool creates/updates home pref for displayed type.
- 'Set columns' retrieves saved column settings (if extant) for displayed type.
- If multiple types are displayed or no saved pref, 'Set columns' calculates columns as before.



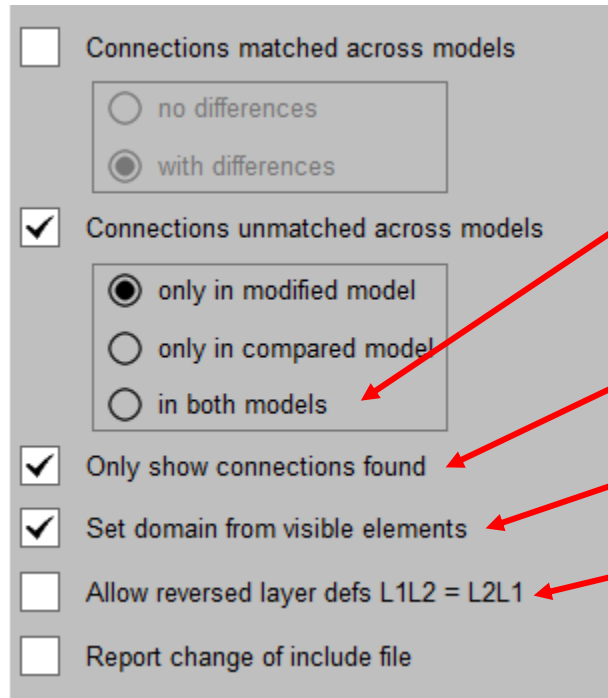
CONNECTION TABLE

Dismiss View... Options... Refresh Action: update & remake Show all Spotwelds Adhesive Rivet Write... Sketch conx on select

Apply: Undo All Selected Changed Autoscale Clear Sel all Select Show sel Bolts/Joints Spot lines Set columns Save columns to pref

ID	Type	Subtype	Status	Bolt Length	Diameter
CNX867	BOLT	Cylindrical NRB	Bad	20	10
CNX868	BOLT	Cylindrical NRB	Bad	20	10
CNX960	BOLT	Cylindrical NRB	Bad	20	10

Connection compare – new options



The screenshot shows a settings panel for 'Connection compare'. It contains several checkboxes and radio button groups. Red arrows point from the following list items to their corresponding settings in the panel:

- Show in table unmatched connections in both models. (points to 'in both models' radio button)
- Display in model only connections that are found. (points to 'Only show connections found' checkbox)
- Show in table only connections in the visible domain. (points to 'Set domain from visible elements' checkbox)
- Suppress the difference if reversed layers match. (points to 'Allow reversed layer defs L1L2 = L2L1' checkbox)

Options visible in the panel:

- ☐ Connections matched across models
 - ☐ no differences
 - ☒ with differences
- ☒ Connections unmatched across models
 - ☒ only in modified model
 - ☐ only in compared model
 - ☐ in both models
- ☒ Only show connections found
- ☒ Set domain from visible elements
- ☐ Allow reversed layer defs L1L2 = L2L1
- ☐ Report change of include file

- Show in table unmatched connections in both models.
- Display in model only connections that are found.
- Show in table only connections in the visible domain.
- Suppress the difference if reversed layers match.

Assign mass dry run

Assign mass dry run

- *Calculate* for assign mass with current targets now displays the achievable mass properties before applying the solution, saving time.

Set Mass ▶ 0.44 Adding: 1.867952E-2

Set CofG: ▶ X: 2440.0 Y: 100.0 Z: 640.0

Set Inertia ▶ XX: * XY: * XZ: *

Reset Props YY: * YZ: *

Error tolerance (percent) 5.0 ZZ: *

☐ Current mass properties ☒ Achievable mass properties

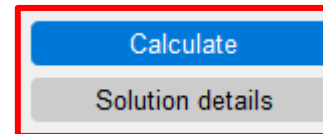
Actual mass: 0.44 Plot mass Show CofG

Actual CofG: X: 2438.356 Y: 75.01173 Z: 643.0315

Inertia tensor: IXX: 213527.8 IXY: 4535.681 IXZ: 19420.64

IYY: 661245.5 IYZ: -783.0419

IZZ: 777294.9

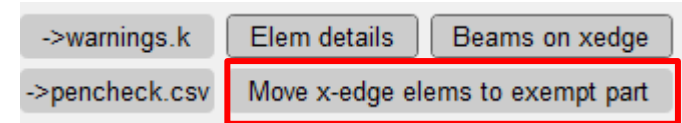


Achievable mass property values to be implemented on Update		
ACHIEVABLE MASS	TARGET MASS	ERROR
0.44	0.44	+0
ACHIEVABLE COGX	TARGET COGX	ERROR
2438.36	2440	-1.64412
ACHIEVABLE COGY	TARGET COGY	ERROR
75.0117	100	-24.9883
ACHIEVABLE COGZ	TARGET COGZ	ERROR
643.031	640	+3.03147
ACHIEVABLE INERTIA TENSOR:		
2.1353e+05		
4.5357e+03 6.6125e+05		
1.9421e+04 -7.8304e+02 7.7729e+05		
===== Max Error in solution: 0.57% =====		

Crossed edge fix

Crossed edge fix

- Contact automatic single surface with exempt part set offers fix of crossed edges by moving elements to exempt part – see PenCheck panel.



- (Another) Part set with title PR_PROTECT_FROM_ELEMENT_SWAP ensures that elements of these parts are not swapped.



Beam-beam/shell automatic general contact

Beam-beam automatic general contact

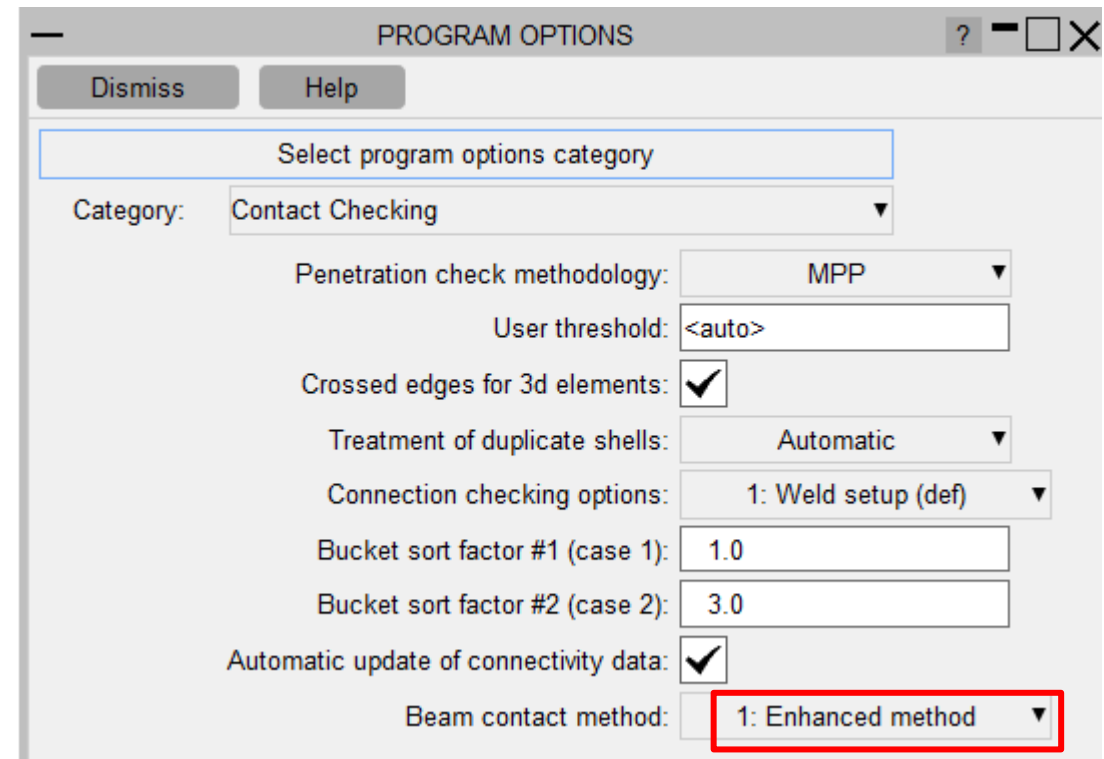
- Escape end condition can lose penetrations in PRIMER 19.x.



- In PRIMER 20.0 these are captured.

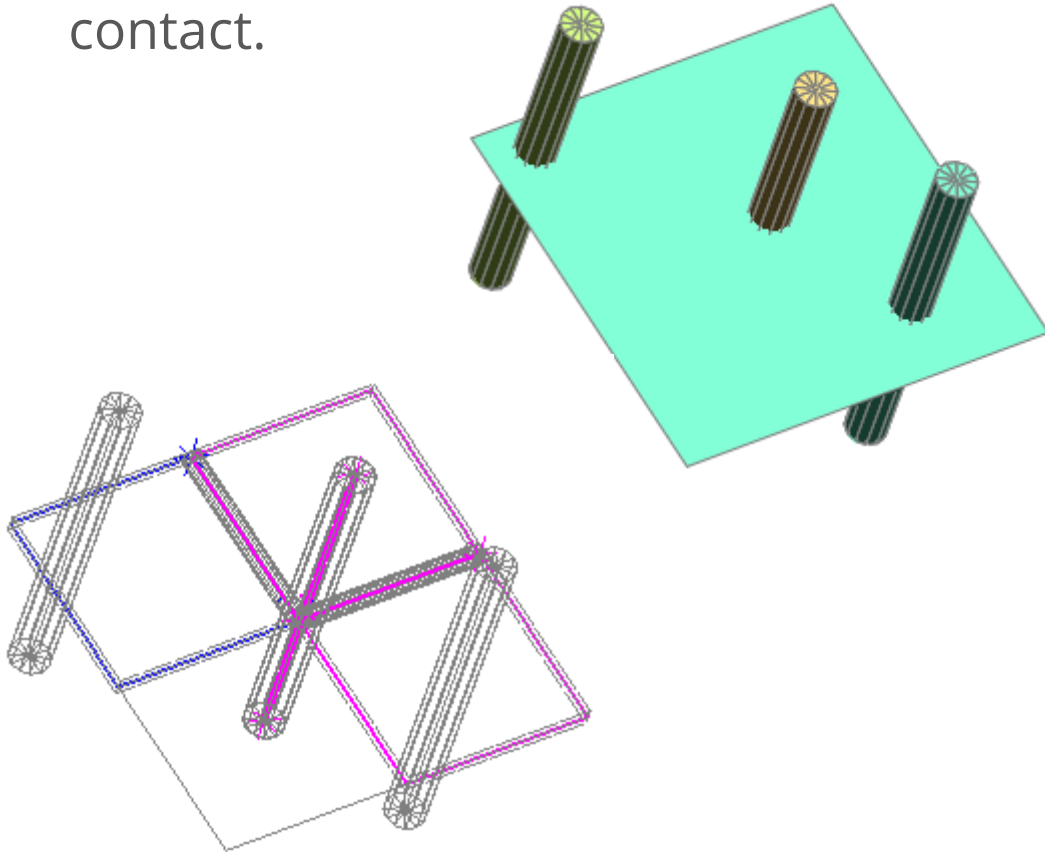


- In PRIMER 20.0 default option is enhanced method.



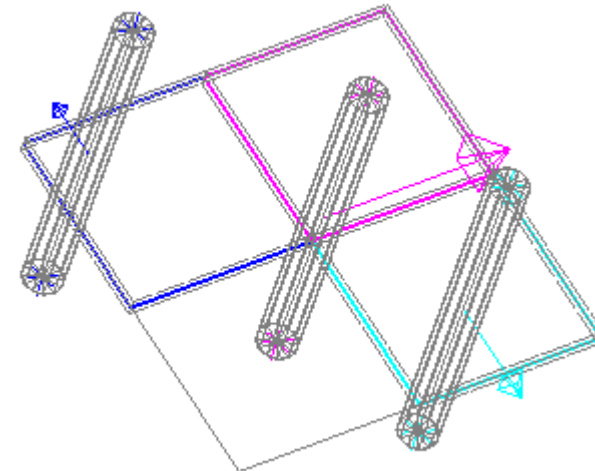
Beam-shell automatic general contact

- In PRIMER 19.x beam penetrating through shells may miss edge contact.



- In PRIMER 20.0 default option is enhanced method.
- This calculates the escape vector to the nearest edge and displays penetration more clearly.

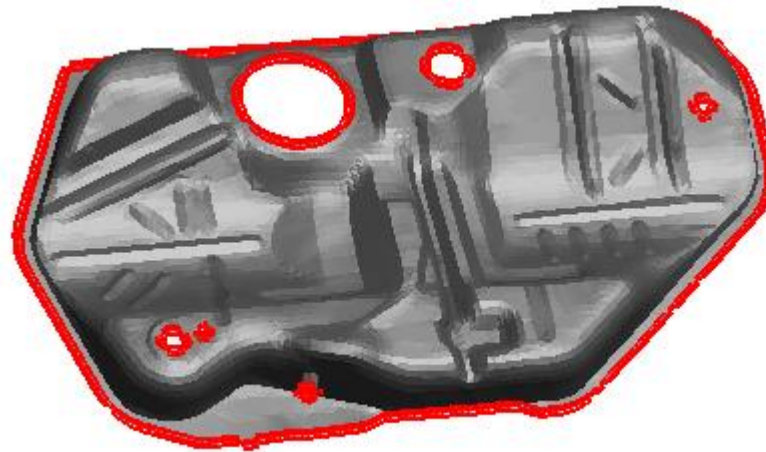
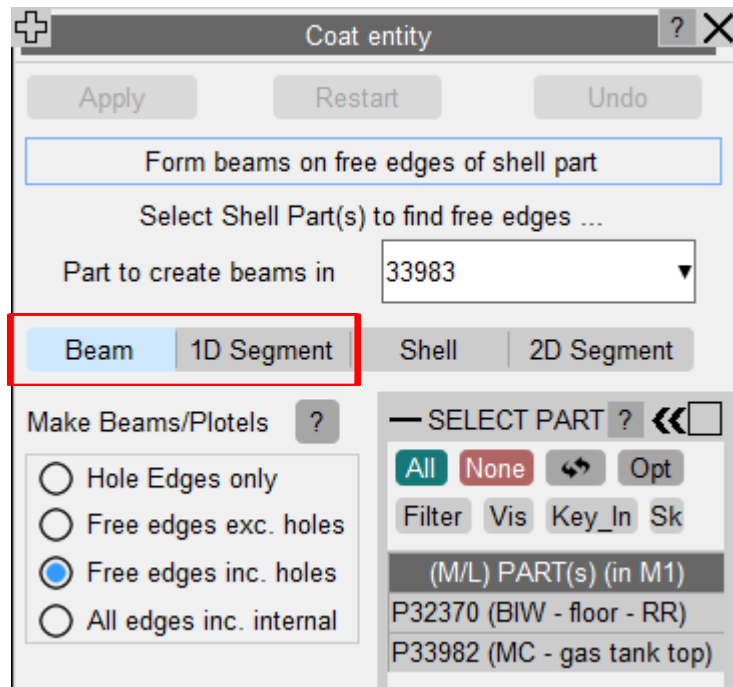
Beam contact method: 1: Enhanced method ▼



Coat with 1d segments/beams

Coat with 1d segments or beams

- Option to add 1d segments or beams to free edges and/or hole edges introduced.



User-defined Materials extended

USER_DEFINED_MATERIAL_MODELS

- User defined material models allow you to implement your own material models, or use 3rd party material models in LS-DYNA.
- Any number of material parameters can be defined on the cards that are used by the external material subroutines.
- These parameters can be anything – for example real numbers (moduli, stresses) or integers (references to *DEFINE_CURVES).

CREATE MATERIAL in model 1

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

Include: M1 <Master file>

Create material in model 1

Label: 1 Etypes: H, S, B, TS, SPH, IGA

Type: T_281-290: USER_DEFINED_MATERIAL_MODELS FLTR

Suffix: <none>

Title: <No material name given>

Suffixes:

Row\Col	1	2	3	4	5	6	7	8
1	<Label>	RO F	MT I	LMC I	NHV I	IORTHO/SP	IBULK I	IG I
	1	0.0	281	10	0	0	0	0
2	IVECT I	IFAIL I	ITHERM I	IHYPER I	IEOS I	LMCA I		
	0	0	0	0	0	4		
3	PARAM F	LCK1 I	LCT1 I	LCK2 I			SLIM I	LCK3 I
	0.0	0	0	0			0	0
4	E F	PR F						
	0.0	0.0						
5	LCKA1 I	CP F		NUMINT I				
	0	0.0		0				

USER_DEFINED_MATERIAL_MODELS CSV

- As the material subroutines are external, PRIMER does not know what the various material parameters mean.
- You can use CSV configuration files within PRIMER to make it easier to use user-defined materials, for example for maintaining references to entities in the model.

LMC1	PARAM	FLOAT	
LMC2	LCK1	TABLE	
LMC3	LCT1	LCUR	
LMC4	LCK2	TABLE	
LMC5			
LMC6			
LMC7	SLIM	INTEGER	
LMC8	LCK3	LCUR	
LMC9	E	FLOAT	YM
LMC10	PR	FLOAT	PR
LMCA1	LCKA1	TABLE	
LMCA2	CP	FLOAT	
LMCA3			
LMCA4	NUMINT	INTEGER	

USER_DEFINED_MATERIAL_MODELS CSV

- You can specify field titles and the type of the field for user-defined materials. The CSV file should contain the field name, field type and optionally whether the field is Young's modulus or Poisson's ratio.
 - 1st column determines whether the field is of LMC type or LMCA type;
 - 2nd column should contain the field header name for each field;
 - 3rd column should contain the field type: Currently, we support float, integer, load curves and tables;
 - 4th column (optional) whether the field is Young's modulus or Poisson's ratio.
- This CSV file is then read into PRIMER using preferences. Define the path of the CSV file in the oa_pref (mat_xx_user_defined), e.g.
 - primer*mat_285_user_defined: C:\Users\xx\my_materials\mat285.csv;
 - primer*mat_45_user_defined: C:\Users\xx\my_materials\mat45.csv.

LMC1	PARAM	FLOAT	
LMC2	LCK1	TABLE	
LMC3	LCT1	LCUR	
LMC4	LCK2	TABLE	
LMC5			
LMC6			
LMC7	SLIM	INTEGER	
LMC8	LCK3	LCUR	
LMC9	E	FLOAT	YM
LMC10	PR	FLOAT	PR
LMCA1	LCKA1	TABLE	
LMCA2	CP	FLOAT	
LMCA3			
LMCA4	NUMINT	INTEGER	

New Addition for User Defined Materials

- USER_DEFINED_MATERIAL_MODELS have been extended to include the materials in the range 281-290.

CREATE MATERIAL in model 1

Include: M1 <Master file>

Create material in model 1

Label: 1 Etypes: H, S, B, TS, SPH, IGA

Type: T_281-290: USER_DEFINED_MATERIAL_MODELS

Suffix: <none>

Title: <No material name given>

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

Row\Col	1	2	3	4	5	6	7	8
1	<Label>	RO F	MT I	LMC I	NHV I	IORTHO/SP	IBULK I	IG I
1	1	0.0	281	10	0	0	0	0
2	IVECT I	IFAIL I	ITHERM I	IHYPER I	IEOS I	LMCA I		
2	0	0	0	0	0	4		
3	PARAM F	LCK1 I	LCT1 I	LCK2 I			SLIM I	LCK3 I
3	0.0	0	0	0			0	0
4	E F	PR F						
4	0.0	0.0						
5	LCKA1 I	CP F		NUMINT I				
5	0	0.0		0				

Cut section improvements

Cut section improvements – Panel redesign

- The cut section panel has been redesigned to be more user friendly.
- All the relevant buttons have been grouped together to improve user experience.
- More frequently used buttons are towards the top and other buttons are included in tabs below.

- Element Capping tab includes settings related to shell and beam cut-face capping.
- Display Settings tab includes remaining settings.

Element Capping

☐ No 2D element capping **Explain**

☒ Use true thickness x

☐ Use fixed thickness:

☐ Use Part_Contact values:

☒ Wire draw for beams **Explain**

☐ Use true thickness of beam

☐ Use Part_Contact <OPTT>

Display Settings

More drag options... **Save/Retrieve**

Plane Display: **Off** Wireframe Transp

New Panel

Cut Sections

Cutting switch: **OFF** ☒ ☐ ☐

Direction: **D1** **D2** **D3**

Drag (D) **<= ?**

Constant X

Use node: **Pick Node** X coordinate:

Properties...

Positive Action

☒ Omit

☐ Outline

☐ Normal

Swap <==>

Negative Action

☒ Omit

☐ Outline

☐ Normal

☐ Thick cut **?**

☐ Multiple cuts **?** Spacing:

☐ Custom spacing +/- Np:

Parts to exclude: 0 items selected

Start Select **Cancel All**

☒ Element Capping

☒ Display Settings

Old Panel

Cut Sections

Cutting switch: **ON** **Save/Retrieve**

Direction: **D1** **D2** **Add** **Remove**

Definition method: **Constant Y**

Use node: **Pick Node**

or

Y coord:

Move Section

Drag (D key) **<= ?**

More drag options...

Properties...

Positive Action

☒ Omit

☐ Outline

☐ Normal

Swap <==>

Negative Action

☒ Omit

☐ Outline

☐ Normal

Plane Display: **Off** Wireframe Transp

☐ Thick cut **?** ☐ Multiple cuts **?**

Thickness: ☐ Custom spacing

☒ Union **?** Spacing:

☐ Intersection +/- Np:

☐ No 2D element capping **Explain**

☒ Use true thickness x

☐ Use fixed thickness:

☐ Use Part_Contact values:

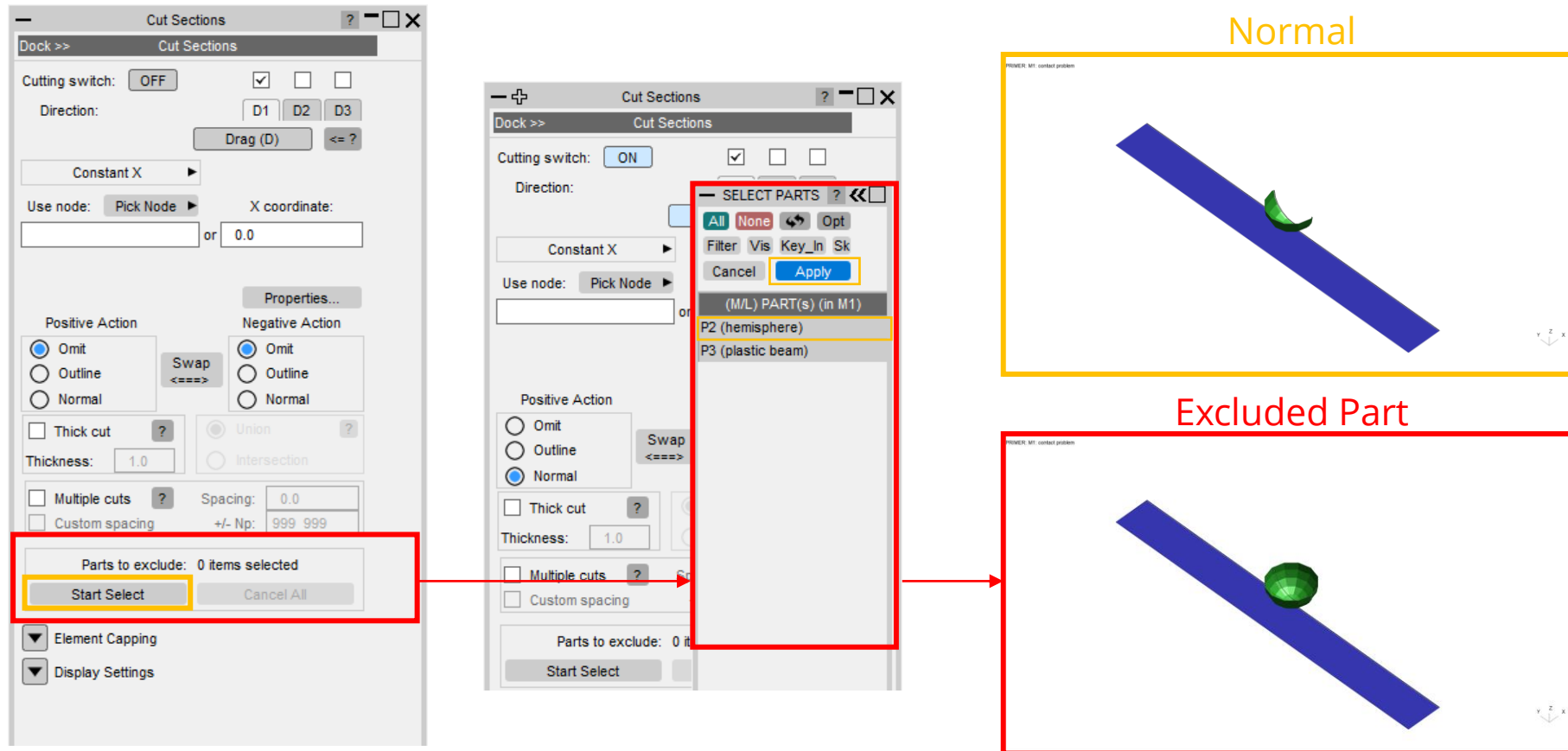
☒ Wire draw for beams **Explain**

☐ Use true thickness of beam

☐ Use Part_Contact <OPTT>

Cut section improvements – Parts to exclude

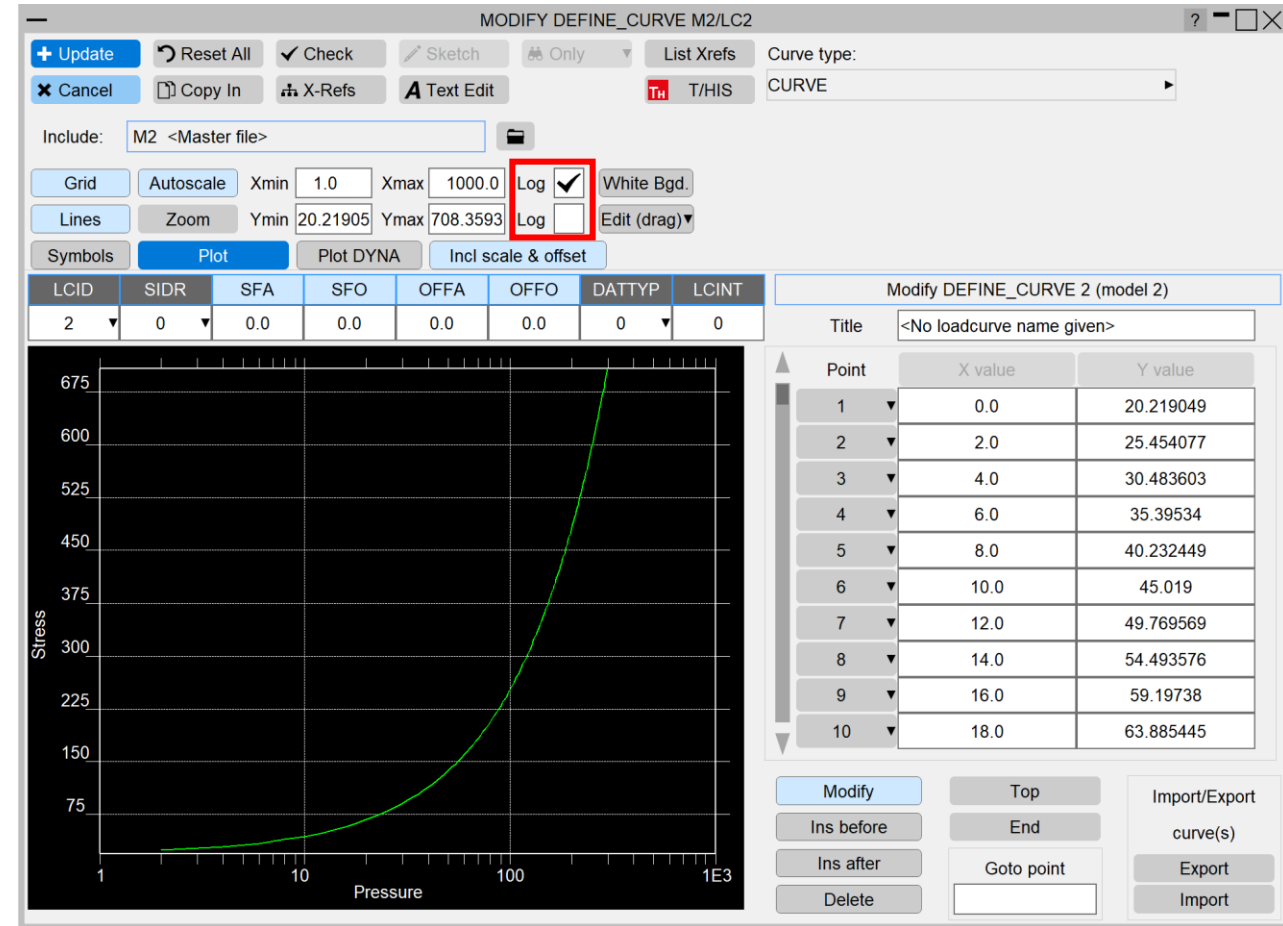
- This option excludes individual Part/Part Sets from the cutting plane so they can be seen on either side of the plane regardless of the +ve and -ve action settings.



Log option for DEFINE_CURVE and TABLE

Log option in DEFINE_CURVE and TABLE

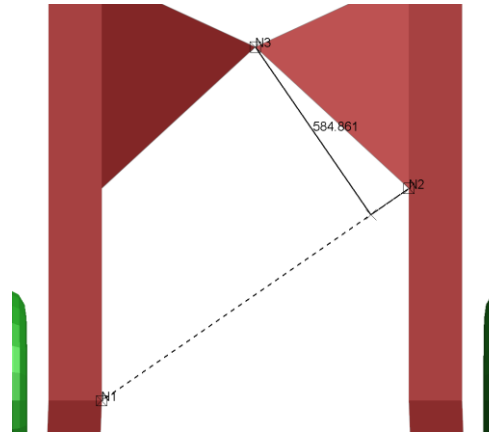
- Added two 'Log' buttons to the DEFINE_CURVE and DEFINE_TABLE editors that can be used to set the graph x and/or y axis to a logarithmic scale.
- When a log button is selected, the axis min gets set to the closest power of 10 less than the positive min value and the max is set to the closest power of 10 greater than the positive max value. The same applies when the 'Autoscale' button is pressed when in log mode. The axis min/max can be changed to any value in the corresponding textboxes.
- All other options in the editors can be used when in log mode (e.g. Symbols, Plot DYNA, Edit, Compare).



New Measure Mode: Node To Line

New Measure Mode: Node To Line

- Added a new Measure mode that calculates the shortest distance from a node (N3) to a line defined by two other nodes (N1 and N2).
- All nodes can be either selected/picked through the popup menu or through the automatically activated QuickPick.
- The projection vector from line N1N2 to N3 is calculated, as well as the projected distances on the coordinate planes.



Measure

Point to Point

Point angle

Node to Node

Node angle

Nodal coord

Curve length

Node to Part

Part to Part

Node Curve len

Distance Plotte

Node to Plane

Node to Line

Node to Line - pick nodes

Done

Reject last

Refresh

N42	►	-434.95459	1002.2917	398.38446
N57	►	435.02783	1602.2916	398.38446
N70	►	4.68766E-2	2002.2919	498.4389

Vector:

-327.1546

474.36462

100.05444

Number Format:

Automatic

Decimal Places

3

Distance:

584.86139

Projected distance on coordinate planes:

XY:

576.2395

YZ:

484.8017

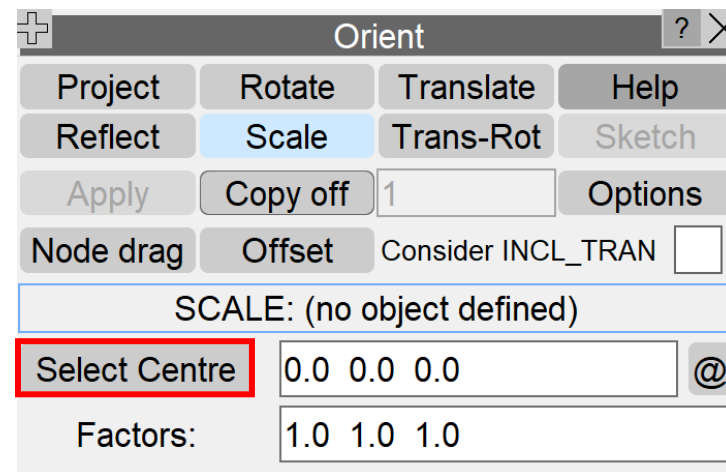
ZX:

342.11258

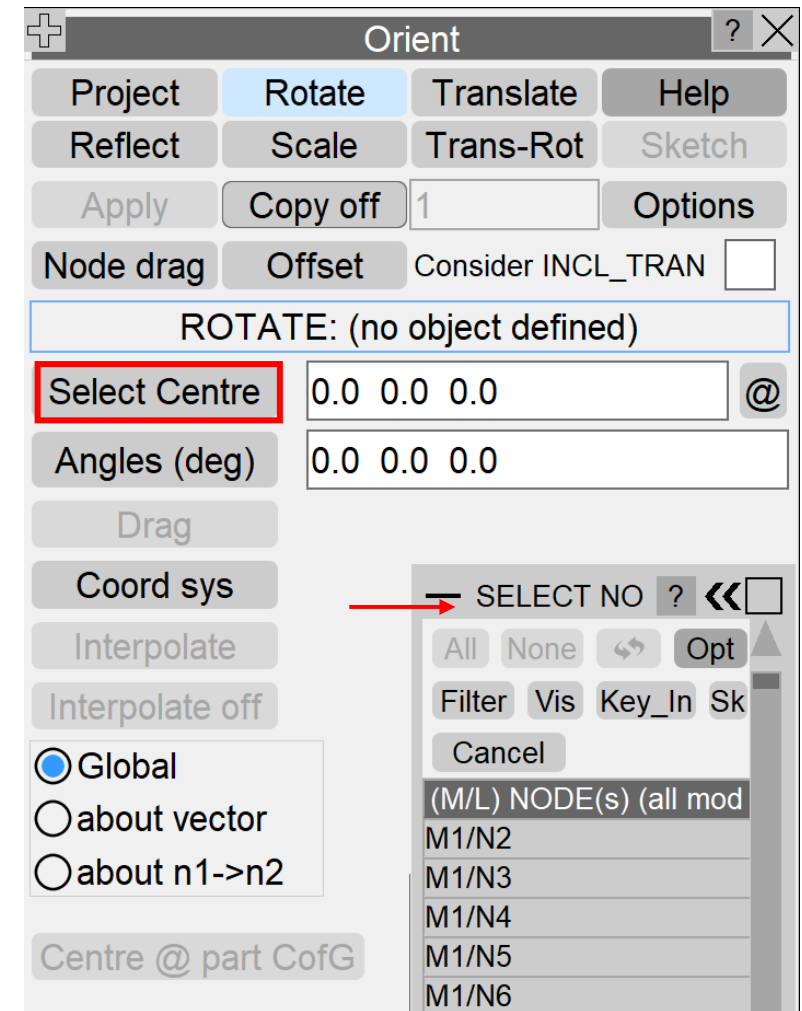
Node Selection in the Orient Panel

Node Selection in Orient Panel

- Changed the processing of the 'Pick Centre' (renamed to 'Select Centre') buttons in Orient>Rotate, Orient>Scale and Orient>Rotate>Drag to display the Node object menu (previously only node picking was activated when these buttons were pressed).



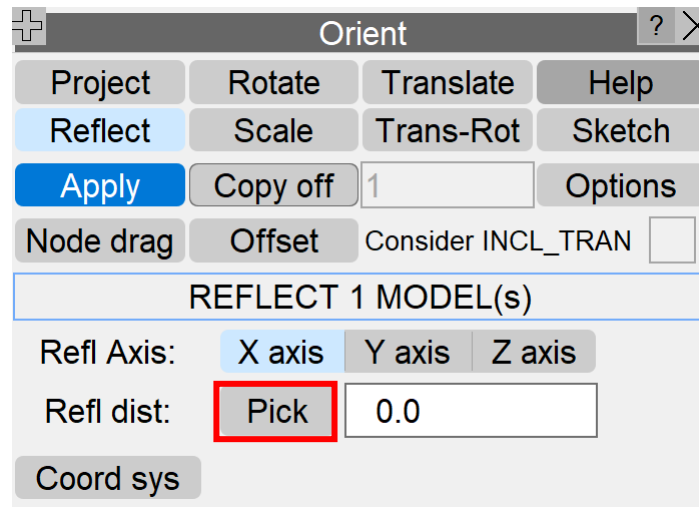
Orient > Scale panel



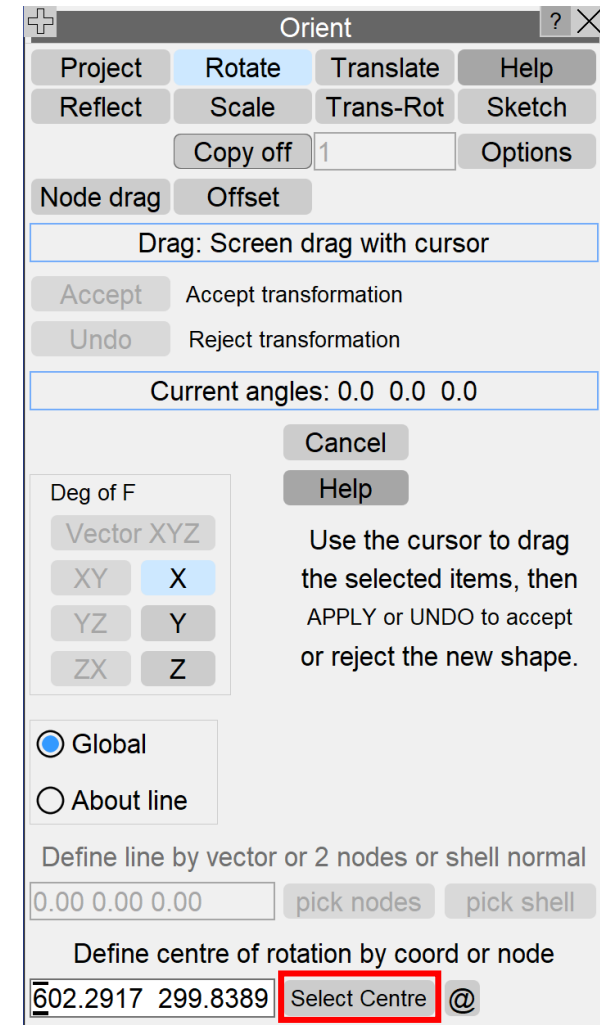
Orient > Rotate panel

Node Selection in Orient Panel

- This allows you to select the centre by node number through the key_in option, while preserving the node picking feature.
- The same changes have also been applied to the 'Pick' button in Orient>Reflect for consistency.



Orient > Reflect panel



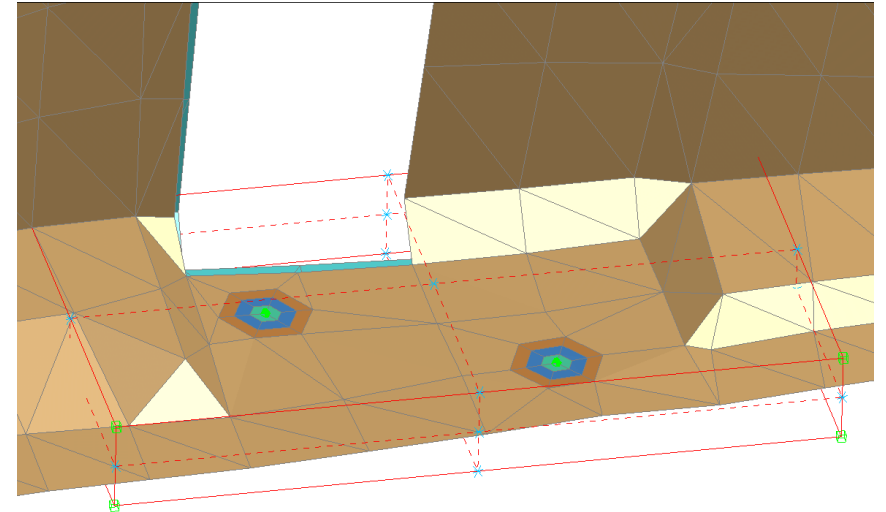
Orient > Rotate > Drag panel

Morphing enhancements

Morphing enhancements

- Improved logic for determining when connections need to be found/updated in morph boxes:
 - Better efficiency as a result;
 - Newly created HAZ welds can now be found by existing morph boxes;
- Automatic detection/updating of morph box connection information (parametric coordinates) can be turned off to allow for manual control:

```
primer*morph_find_new_connections_automatically
```



Morph point options

Visibility

Corner Edge Face Internal

Interpolation

☐ No interpolation
☒ Hidden points only
☐ Full interpolation

Connection options

0 morphed connections require remake

☒ Morph connections

Remake Table

Find new connections: All Boxes Sel Boxes

JavaScript

JavaScript widget additions – Radio buttons

- Radio button widgets allow you to choose a single option from a list.

```
Window.Theme(Window.THEME_CURRENT);

let wi;

// Create window
let w = new Window("JavaScript Radiobutton widget example", 0.85, 1.0, 0.75, 1.0);

// Create radiobutton widget
let r = new Widget(w, Widget.RADIOBUTTON, 1, 51, 1, 41);

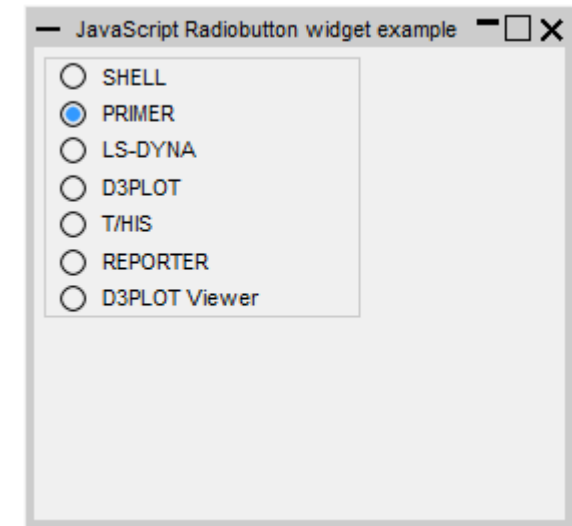
// Add widgetitems to the radio button
wi = new WidgetItem(r, "SHELL");
wi = new WidgetItem(r, "PRIMER");

// Select the PRIMER widgetitem
wi.selected = true;

wi = new WidgetItem(r, "LS-DYNA");
wi = new WidgetItem(r, "D3PLOT");
wi = new WidgetItem(r, "T/HIS");
wi = new WidgetItem(r, "REPORTER");
wi = new WidgetItem(r, "D3PLOT Viewer");

// Show the window and start event loop
w.Show();
```

25,9 All



JavaScript widget additions – Trees

- Tree widgets allow you to display data in a hierarchical structure.

```
// Create window
let w = new Window("JavaScript Tree widget example", 0.85, 1.0, 0.75, 1.0);

// Create tree widget
let t = new Widget(w, Widget.TREE, 1, 71, 1, 61, "Suite");

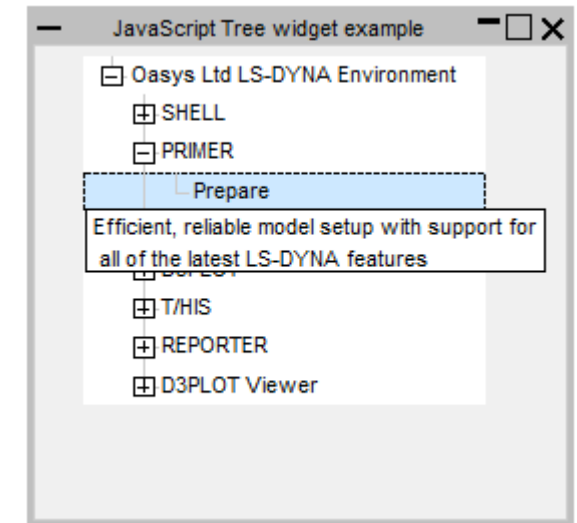
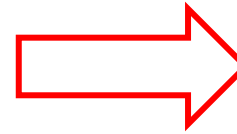
// Add a root node to tree
let env_wi = new WidgetItem(t, "Oasys Ltd LS-DYNA Environment");

// Add a child to the root node
wi = new WidgetItem(t, "SHELL", Widget.CHILD, env_wi);
// add a child to the child node
cwi = new WidgetItem(t, "Portal to the environment", Widget.CHILD, wi);

wi = new WidgetItem(t, "PRIMER", Widget.CHILD, env_wi);
cwi = new WidgetItem(t, "Prepare", Widget.CHILD, wi);
// Add a mouseOver callback for the WidgetItem
wi.onmouseover = wi_onmouseover;
cwi.hover = "Efficient, reliable model setup with support for all of the latest LS-DYNA features";

wi = new WidgetItem(t, "LS-DYNA", Widget.CHILD, env_wi);
cwi = new WidgetItem(t, "Analyse", Widget.CHILD, wi);
```

18,1 11%



Other JS API changes

- The JavaScript engine has been upgraded (Spidermonkey ESR102).
- Added a new `OpenManual()` global function to open a manual at a specified page.
- Options added to View class to select the following second level element contour level settings for Plastic Strain, Form, Area and Volume:
 - Parametric Coordinates;
 - Integration points;
 - Min Strain;
 - Max Strain.
- Colour of NRB, Joint, Airbag, Contact, CrossSection, ExtraNodes, NodeSet, RigidBodies, Spotweld, can now be set from their respective JS API classes.

Other JS API changes

- The following member functions have been added in the classes Solid and Tshell:
 - AspectRatio(), Jacobian() and Warpage().
- Added new JS-API classes for:
 - *FREQUENCY_DOMAIN_FRF;
 - *FREQUENCY_DOMAIN_SSD;
 - *INITIAL_STRESS_BEAM;
 - *INITIAL_STRESS_SOLID;
 - *INITIAL_STRAIN_SHELL;
 - *INITIAL_STRAIN_SOLID.

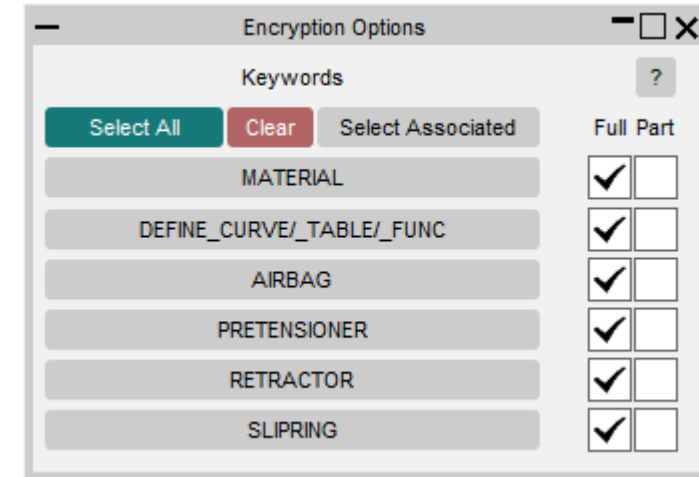
Other JS API changes

- Added a new property 'typeName' to the Material class that provides the material type in the number format (e.g. '001', '034M').
- Added function `Conx.GetShellThickness()` which outputs an array containing:
 - the solid element object;
 - all shell element objects in the specified layer;
 - thicknesses corresponding to each shell.
- Added control for `use_parent_layer` setting for bolts via `Conx.UseParentLayer()`.
- Added functions to select/sketch/unsketch dummy/HBM assemblies.
- Added `GetAssemblyFromID()` to get an assembly object from assembly label.

Encryption Tool enhancements

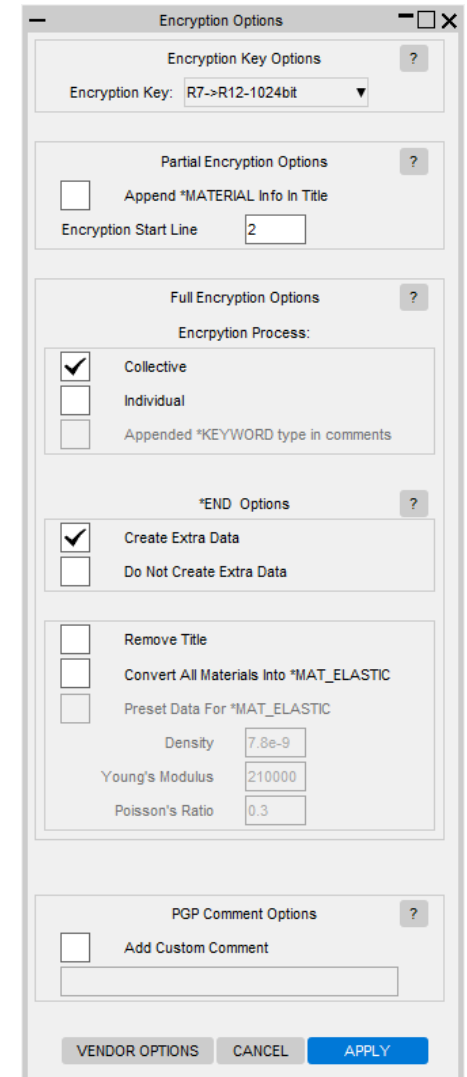
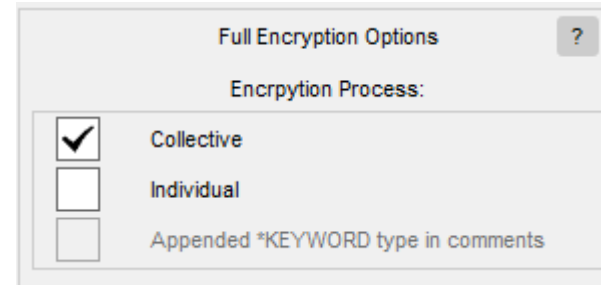
Extended Keyword Support

- Support for full and partial encryption has been extended to the following keywords:
 - *AIRBAG;
 - *ELEMENT_SEATBELT_PRETENSIONER;
 - *ELEMENT_SEATBELT_RETRACTOR;
 - *ELEMENT_SEATBELT_SLIPRING.



Individual vs Collective Encryption

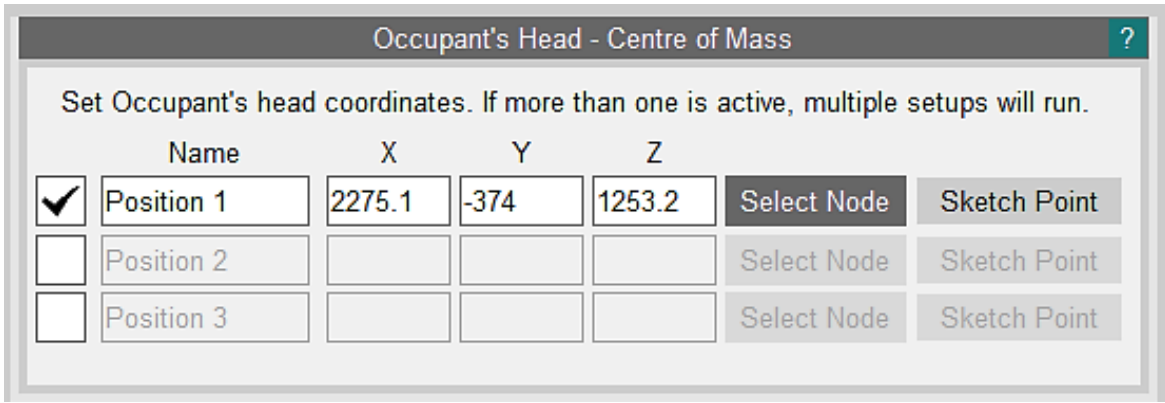
- The encryption tool now has the options of individual vs collective encryption.
- Collective:
 - The collective option groups all the keywords into a single encryption block per include file.
- Individual:
 - The Individual option separates each keyword into its own encryption block.
- Appended *KEYWORD type in comments:
 - Add the material type to the PGP blocks when doing individual keyword encryption. This feature allows you to better identify PGP blocks in the future.



Crash Test Setup enhancements

Crash Test Setup – Occupant's Head panel redesign

- Changes to the Occupant's Head – Center of Mass panel allows you to specify up to three head positions. The head positions can be written to the vehicle xml file and reused later.
- If more than one head position is defined, the script will do the barrier setups for the different head positions simultaneously. This will create a copy of the selected model resulting in multiple models with similar keywords/labels, one for each selected head position.



Occupant's Head - Centre of Mass ?

Set Occupant's head coordinates. If more than one is active, multiple setups will run.

	Name	X	Y	Z		
<input checked="" type="checkbox"/>	Position 1	2275.1	-374	1253.2	Select Node	Sketch Point
<input type="checkbox"/>	Position 2				Select Node	Sketch Point
<input type="checkbox"/>	Position 3				Select Node	Sketch Point

Crash Test Setup – Import Include Transform

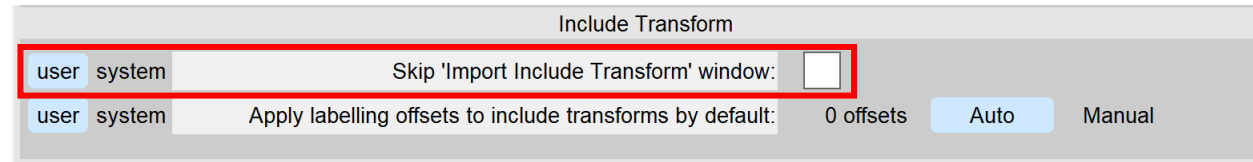
- Added the option to save/import the values in the 'Import Include Transform' window as part of the barrier settings.
- Two toggle buttons 'Auto' and 'Manual' have been added. If 'Auto' is selected, the values will be auto-generated by the script as before. If 'Manual' is selected, the values defined in the barrier settings (barriers.xml) will be used.
- A new preference has been added that can be used to select the default.

The 'Import Include Transform' dialog box features a title bar with standard window controls. Below the title bar, there is a checkbox for 'Apply label numbering offsets:' which is checked, followed by a question mark icon. To the right of this is a red rectangular box containing two toggle buttons: 'Auto' (highlighted in blue) and 'Manual'. Further right is a text field labeled 'TRANID:' containing the value '10001' and another question mark icon. Below these elements is a 2x4 grid of text input fields. The top row is labeled 'IDNOFF', 'IDEOFF', 'IDPOFF', and 'IDMOFF' with values '90000', '50000', '10000', and '10000' respectively. The bottom row is labeled 'IDSOFF', 'IDOFF', 'IDDOFF', and 'IDROFF' with values '10000', '10000', '10000', and '10000' respectively. At the bottom of the dialog are 'Abort' and 'Apply' buttons.

The 'Include Transform' preference window has a title bar. Below it, there are two tabs: 'user' (selected) and 'system'. The main area contains a checkbox for 'Skip 'Import Include Transform' window:' which is unchecked. Below this is a red rectangular box containing the text 'Apply labelling offsets to include transforms by default:' followed by '0 offsets', two toggle buttons 'Auto' (highlighted in blue) and 'Manual', and a question mark icon.

Crash Test Setup – Import Include Transform

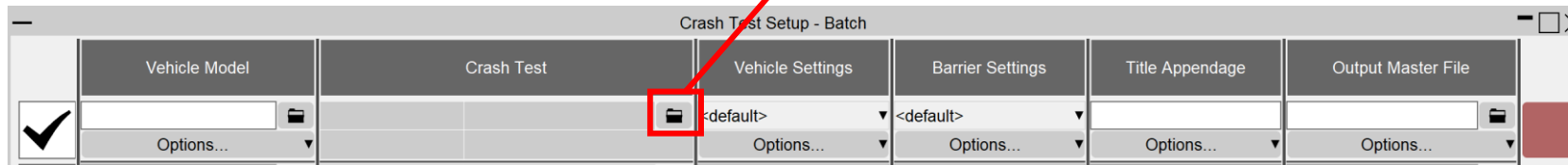
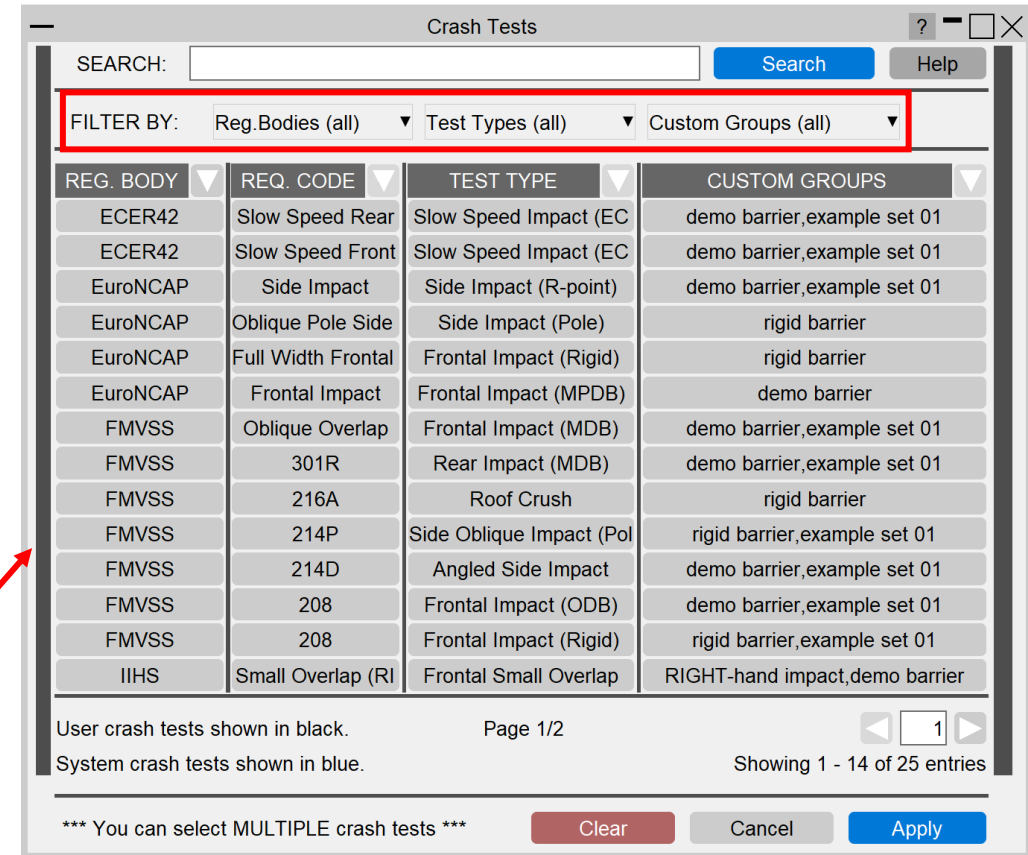
- Added the option to skip the 'Import Include Transform' window through a new preference setting. If this is turned on, then the labeling offsets and the transformation ID will be set based on the mode selected for the 'Apply labelling offsets to include transforms by default' preference.



- Note that if the '0 offsets' or 'Manual' options are selected and 'Skip the 'Import Include Transform' window' is turned on but the TRANID set in the barrier settings is already used in the model then a warning will popup when importing the barrier to give the option to overwrite with the auto-generated value. Another warning will popup if any of the offsets defined in the barrier settings causes conflicts with the vehicle model as the auto-generated values will be used instead.

Batch Crash Test Setup – Filter crash tests in batch setup

- Added the option to filter the list of load cases in the test selection window of the batch script.
- This will allow for easier selection of load cases within the batch script.



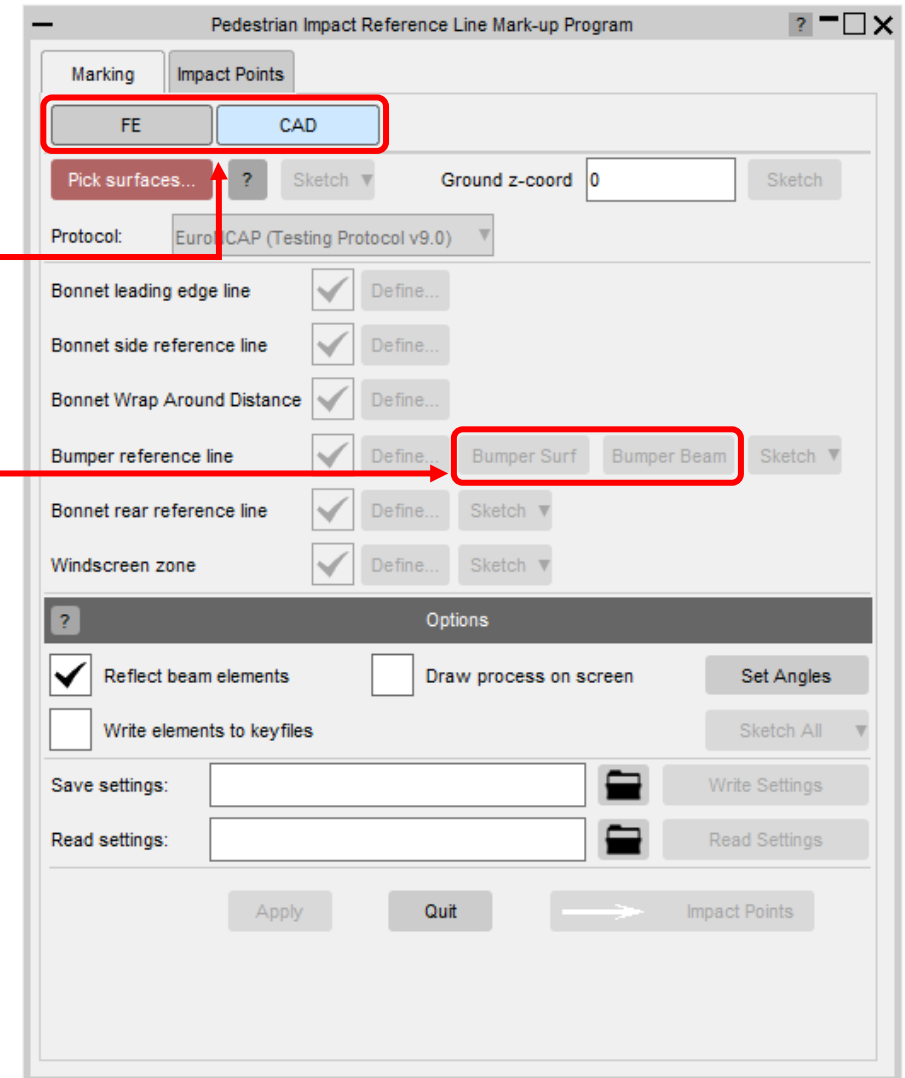
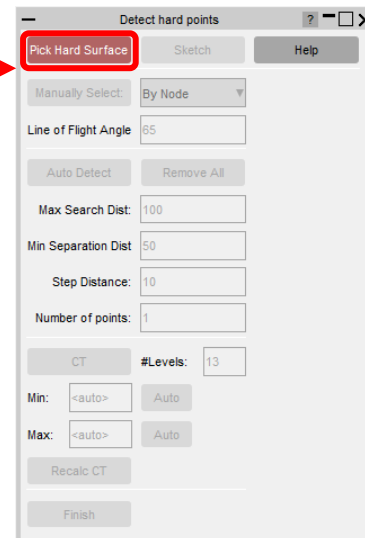
Crash Test Setup – Other Changes

- In the barrier settings of the IIHS Small Overlap Right or Left load case, an option to use the default impact side was added. This will allow you to easily select the default barrier impact side and is active by default.
- The Batch Crash Test Setup script now reuses vehicle models if they are the same for multiple setups by copying the model. This will speed up the script as it skips unnecessary rereads of vehicle keyword files.
- The Batch Crash Test Setup script can now do a Frontal Rigid Wall load case. This load case is available through the main script by saving a crash test using the 'Use Rigidwall' option in the EuroNCAP Full Width Frontal Impact case.

Pedestrian Markup enhancements

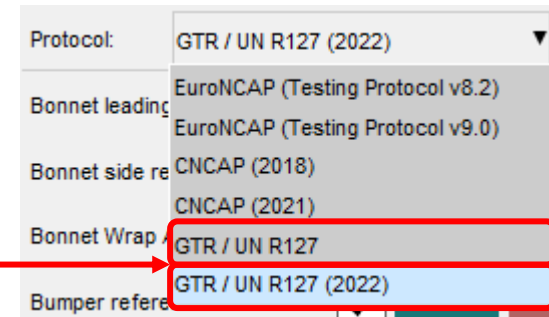
Pedestrian Markup – CAD model input

- Support added to read in a CAD model as input for the markup tool:
- Toggling the top button to “CAD” will allow you to use a CAD model;
- You can pick “Surfaces” instead of “Parts” everywhere in the script.



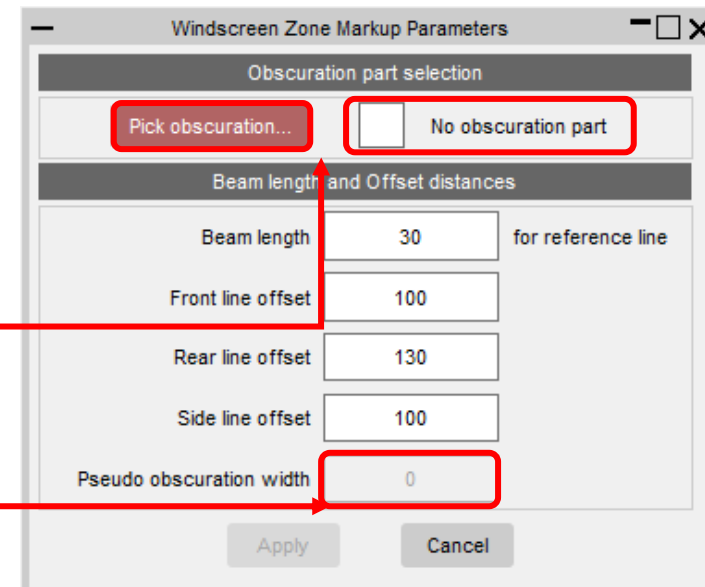
Pedestrian Markup – Updated UN R127 Support

- Support added for UN R127 03 series amendments (issued in 2022):
 - A new protocol option has been added, *GTR / UN R127 (2022)*, for these amendments. The previous protocol is now named *GTR / UN R127 (pre 2022)*;
 - Calculation of new windscreen and cowl testing zones have been added;
 - Windscreen zone can be determined from an obscuration part or from the windscreen free edge;
 - An optional pseudo obscuration width can be applied when no obscuration part is provided.



A screenshot of a software interface showing a dropdown menu for protocol selection. The menu is open, displaying several options. A red box highlights the bottom two options: "GTR / UN R127" and "GTR / UN R127 (2022)". A red arrow points from the text "The previous protocol is now named" in the list to the "GTR / UN R127" option.

Protocol:	GTR / UN R127 (2022)
Bonnet leading	EuroNCAP (Testing Protocol v8.2)
Bonnet side re	EuroNCAP (Testing Protocol v9.0)
Bonnet Wrap	CNCAP (2018)
Bumper refer	CNCAP (2021)
	GTR / UN R127
	GTR / UN R127 (2022)

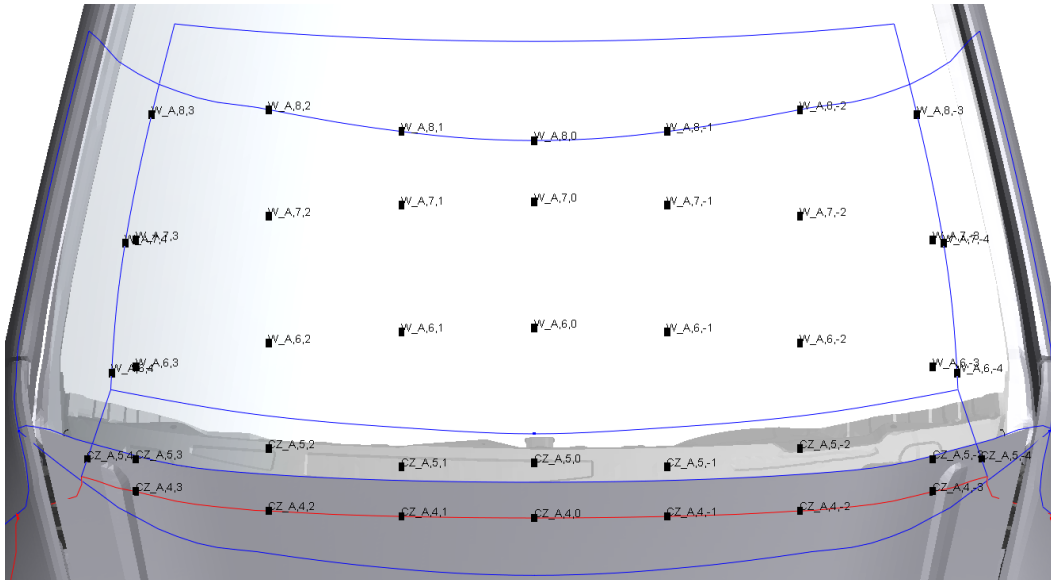


A screenshot of the "Windscreen Zone Markup Parameters" dialog box. The dialog has two main sections. The first section, "Obscuration part selection", contains a "Pick obscuration..." button and a "No obscuration part" checkbox. The second section, "Beam length and Offset distances", contains input fields for "Beam length", "Front line offset", "Rear line offset", "Side line offset", and "Pseudo obscuration width". A red box highlights the "Pseudo obscuration width" field, which has the value "0". A red arrow points from the text "An optional pseudo obscuration width can be applied" in the list to this field.

Obscuration part selection		
Pick obscuration...	<input type="checkbox"/>	No obscuration part
Beam length and Offset distances		
Beam length	30	for reference line
Front line offset	100	
Rear line offset	130	
Side line offset	100	
Pseudo obscuration width	0	

Pedestrian Markup – Updated UN R127 Support

- Support added for new UN R127 03 series amendments:
 - Updated impact points GUI to allow the specification of impactor orientation angles for each impact zone;
 - Windscreen and Cowl testing area labels begin with *WZ_(A or C)* and *CZ_(A or C)* respectively.



Marking

Impact Points

Type: Head

Explain

Sketch

☒ Label points

Method: Define Distance

Distance between points: 100

Label: Standard

?

Create Manually

Hard Points

Robustness Points

Write Boundaries

Testing areas:

	Bonnet top	Windscreen	Cowl	
Points:	Adult: <input checked="" type="checkbox"/>	Adult: <input checked="" type="checkbox"/>	Adult: <input checked="" type="checkbox"/>	?
Angles:	0	0	0	
	Child: <input checked="" type="checkbox"/>	Child: <input type="checkbox"/>	Child: <input type="checkbox"/>	
	0			

Impactor Parameters:

Adult

Child

?

Run Parameters:

Adult

Child

Output points to file:

?

Use point name for zone: ☒

Bonnet top Adult:

Bonnet top Child:

Windscreen Adult:

Windscreen Child:

Cowl Adult:

Cowl Child:

Write

Build

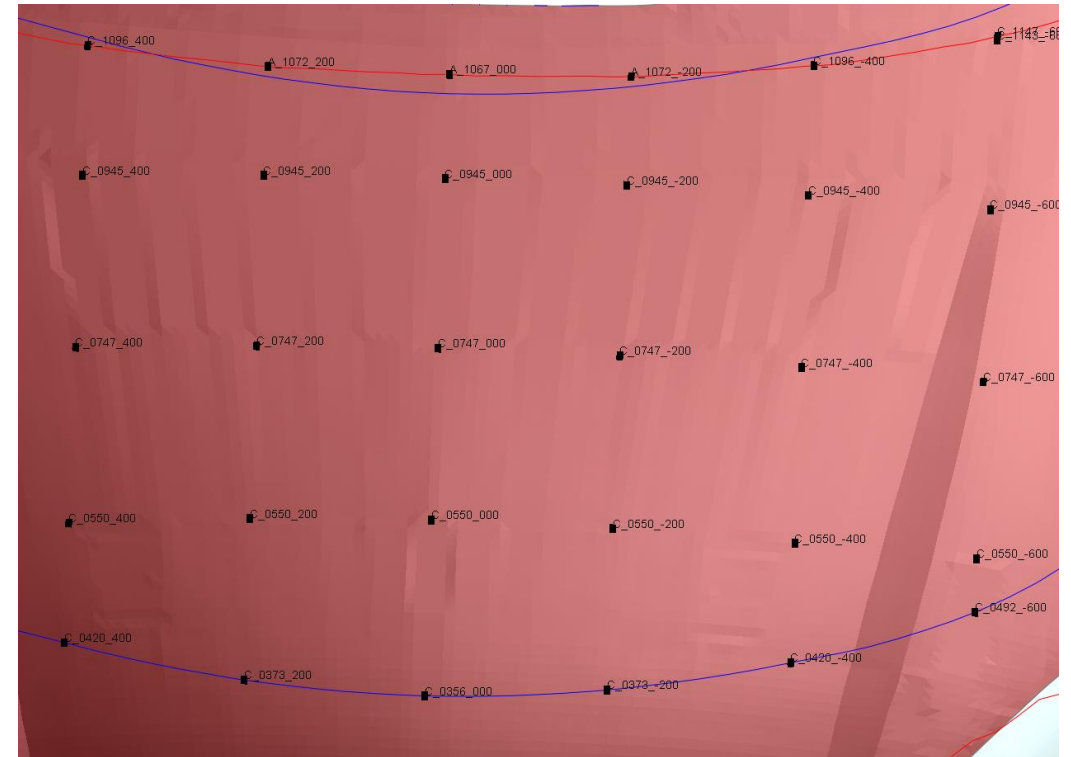
Quit

←

Marking

Pedestrian Markup – Coordinate Labelling

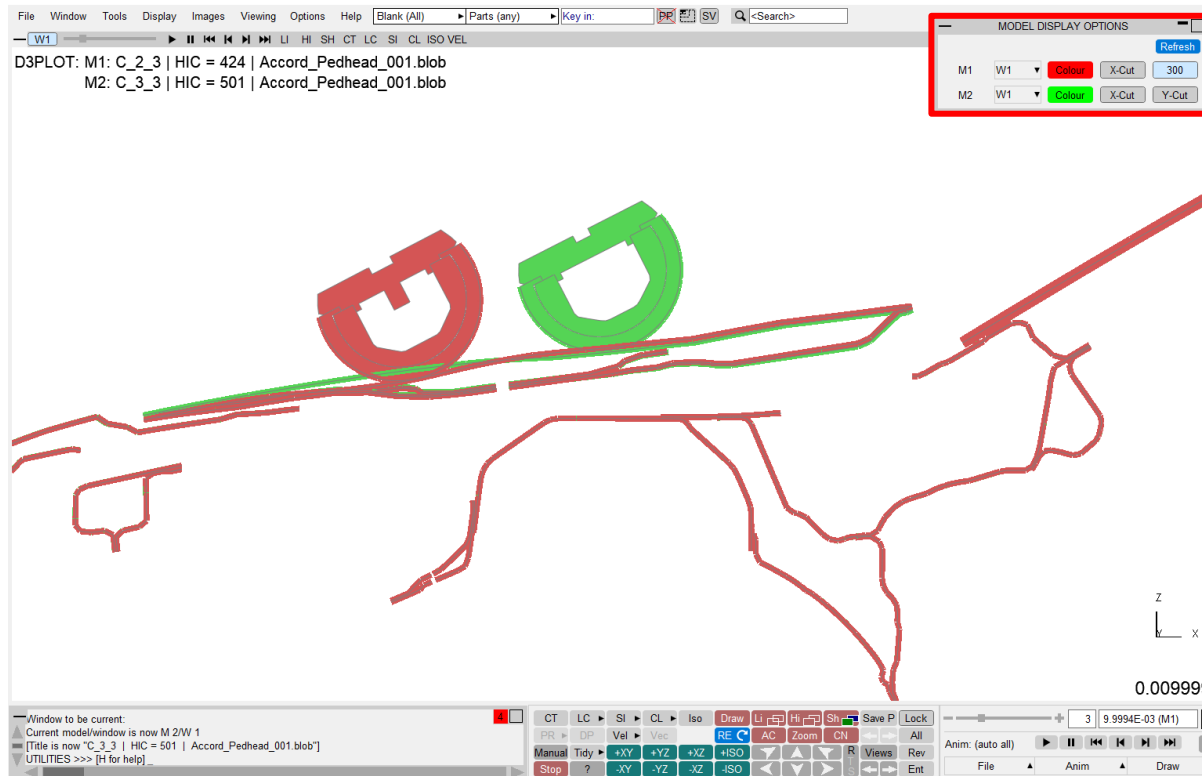
- A new labelling scheme has been added for Head impact points:
 - Labels represent coordinates of impact points;
 - P_XXXX_YYY format; where P is the zone name, $XXXX$ is the point's x-coordinate, and YYY is the point's y-coordinate;
 - Duplicate labels will have the point's z-coordinate appended: $P_XXXX_YYY_ZZZZ$.



HIC Area Calculator

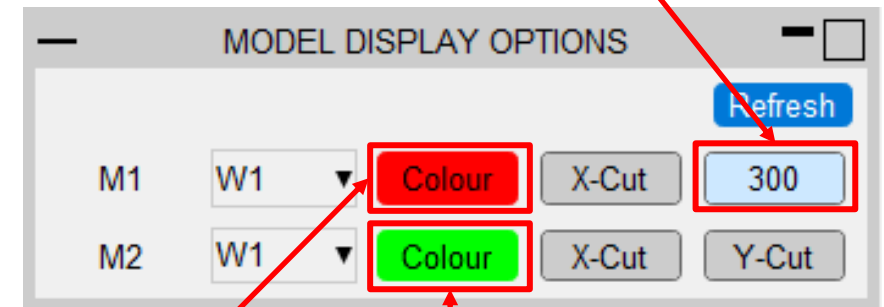
D3PLOT Display Options

- For models opened in D3PLOT using the HIC Area Calculator, there are now options to control the model display:



- Option to colour models different colours.
- Option to switch on/off cut-sections at the head impact point's x or y coordinates.

Cut-section at y coordinate (300) for M1 in W1.



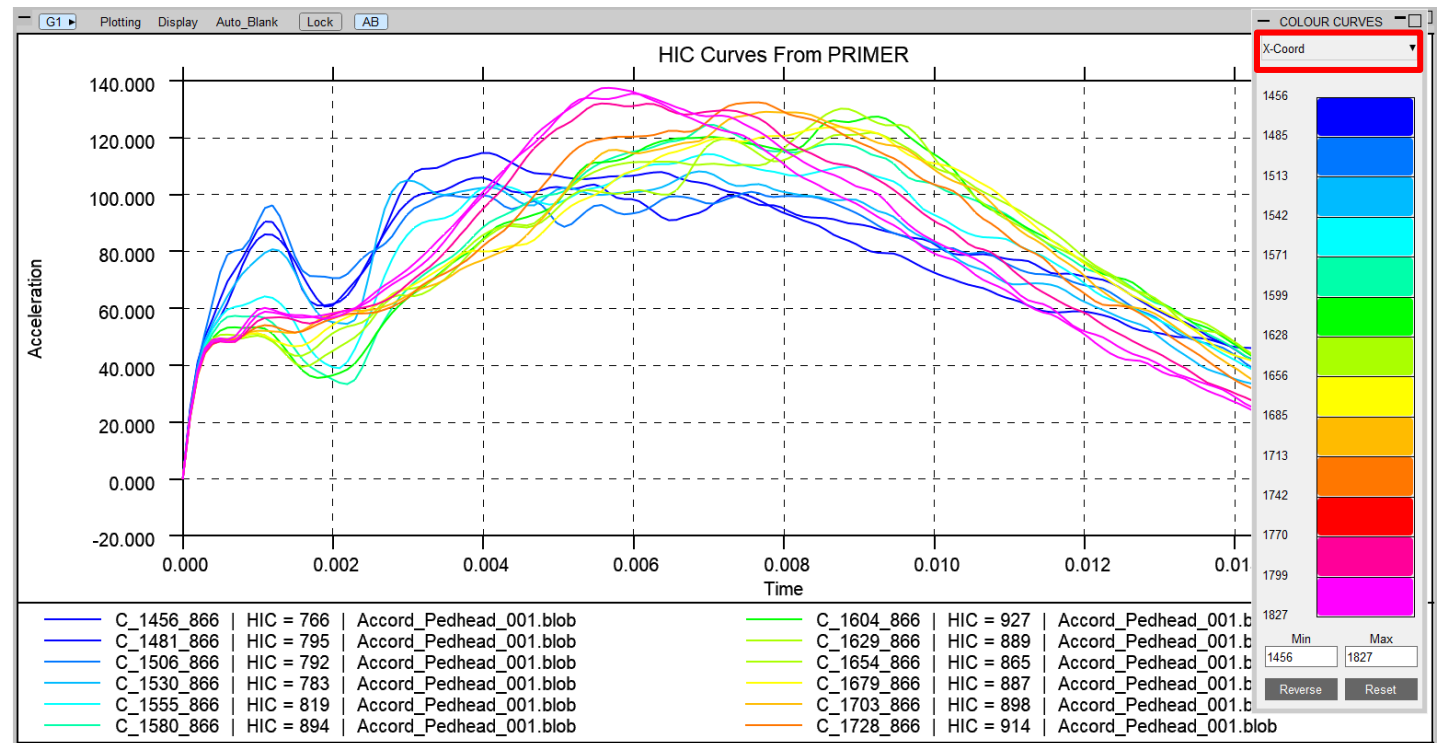
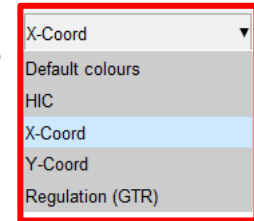
Colour M1 red

Colour M2 green

T-HIS Curve Colouring Options

- There are now options which allow you to colour HIC curves in T-HIS according to:
 - HIC;
 - X-Coordinate;
 - Y-Coordinate;
 - Regulation (regulation and bands used taken from selection in PRIMER);
 - User defined data (if defined in input file).

Right-click to access other options



T-HIS Curve Colouring Options – User Data

- Any number of user defined data entries can be provided in the input text file and used to colour curves in T-HIS. The format of the input file with two additional data entries would be:

X coordinate, Y coordinate, Z coordinate, HIC value, [Name], [Path], [User Data 1], [User Data 2]

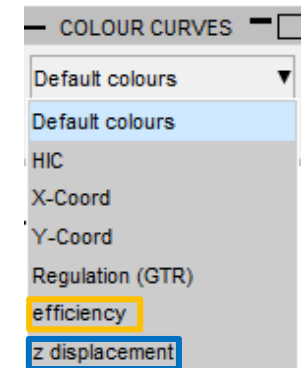
- This information can optionally be used with a new comment line which can be included at the top of the input text file: `$user_data_headings`

- This comment allows you to define headings for the user defined data.

- For example:

```
$user_data_headings, efficiency, z displacement
data 1020 -800 858.1 901.2 C_6_8 C:/ANALYSIS/C_6_8/C_6_8.key 0.134 40.9
data 1070 -800 873.7 817.1 C_7_8 C:/ANALYSIS/C_7_8/C_7_8.key 0.209 45.7
data 1490 -800 986.2 312.3 A_9_8 C:/ANALYSIS/A_9_8/A_9_8.key 0.389 63.2
data 1396 -700 965.2 263.2 A_9_7 C:/ANALYSIS/A_9_8/A_9_8.key 0.148 57.3
```

Able to colour HIC curves based on user defined data in T-HIS

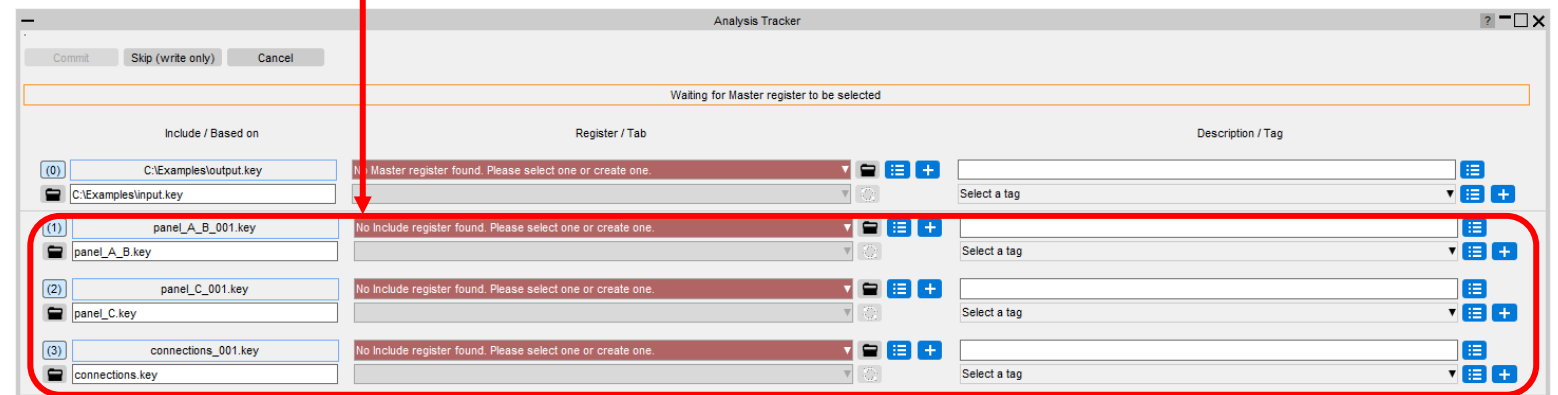
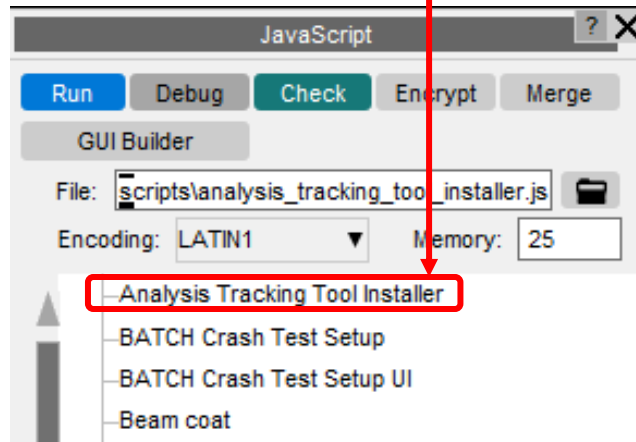
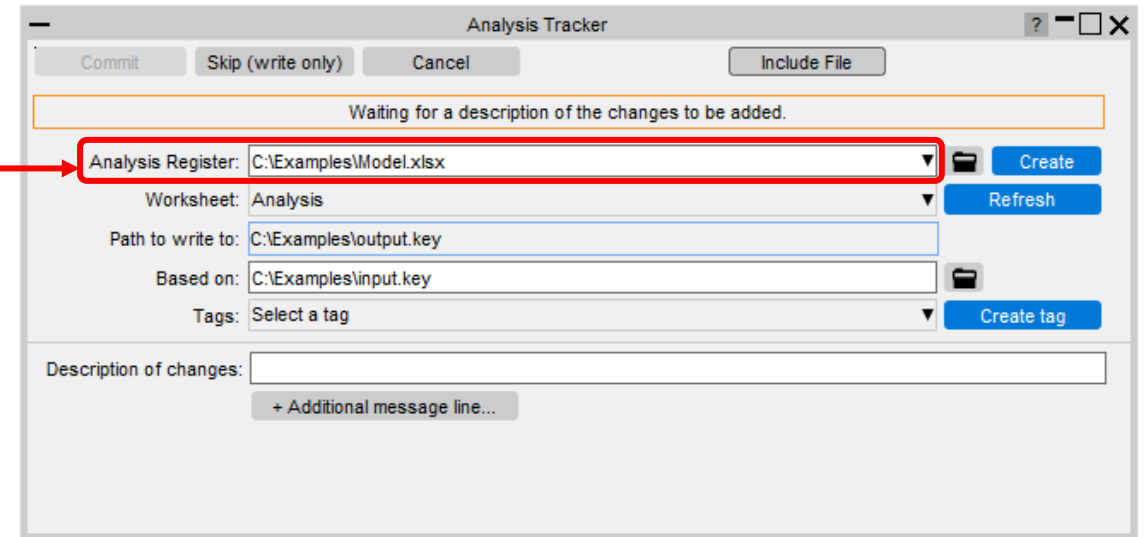


- The headings must be comma delimited.

Analysis Tracking Tool

Analysis Tracking Tool

- The Analysis Tracking Tool is designed to automatically populate an analysis register spreadsheet. Making it easy for you to keep an analysis register up to date with changes in models or include files.
- It is available for installation with other JavaScript Tools, see tutorial for details.



Metal-Forming Tool

Metal-Forming Tool

- The orientation of shells is now taken into consideration when mapping stress and strain data from source to target.
- The angle between the normal vectors of matched shells is computed, and if it is above 90° (meaning the normal vectors of shells are facing opposite/nearly opposite directions), then the sign of the parametric position (field 't') for each integration point in the *INITIAL_STRESS_SHELL and *INITIAL_STRAIN_SHELL cards is inverted before the data is mapped to target. This allows the above data to be mapped in terms of the global system.

Workflows

Workflows

Previous versions of the Oasys Suite contained powerful tools and capabilities that could be used to interrogate and debug your analysis results. However...

1. The tools were not always customised for specific loadcases or tasks
2. You may have needed to manually perform a number of steps to process your results, which could be time-consuming and prone to error
3. The JavaScript API could be used to create tools to automate your post-processing workflow, but this requires time, resource and knowledge, which is not always available

To address this issue, a new **Workflows** feature has been added that provides tools customised for specific loadcases and tasks, and built upon the existing capabilities in the Oasys Suite, to make it easier to interrogate and post-process results.

Workflows

Workflows allow you to tag your model in PRIMER with meta data, e.g.

Crash test type

Occupants

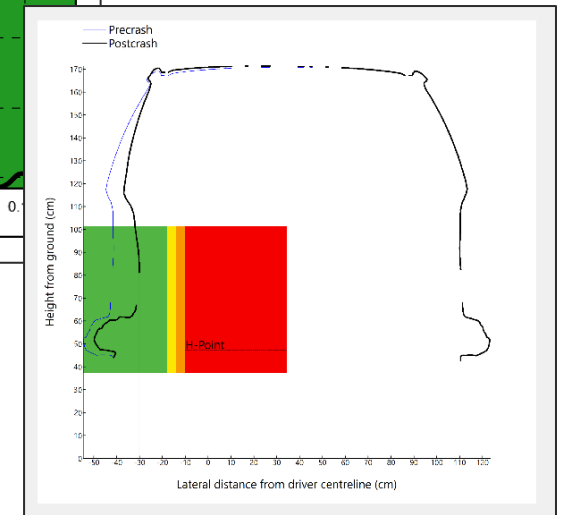
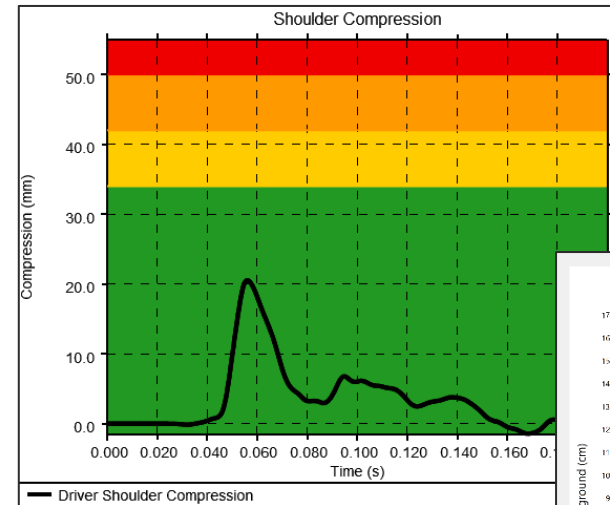
Structures

Unit system

etc.



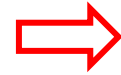
The post-processing software reads this data and provides tools that are relevant to your analysis



Workflows

To use a Workflow:

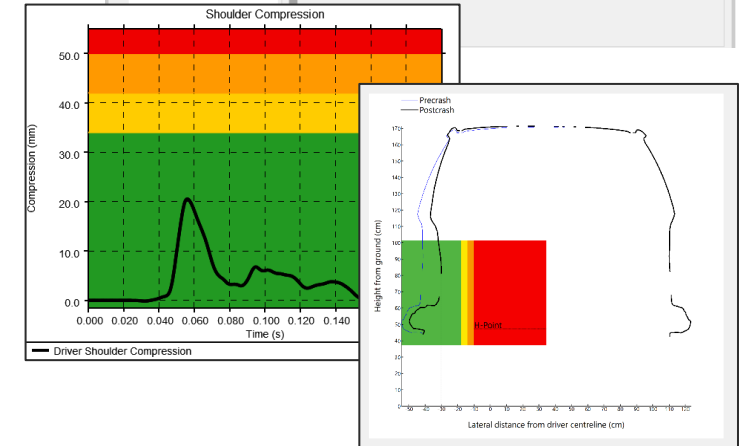
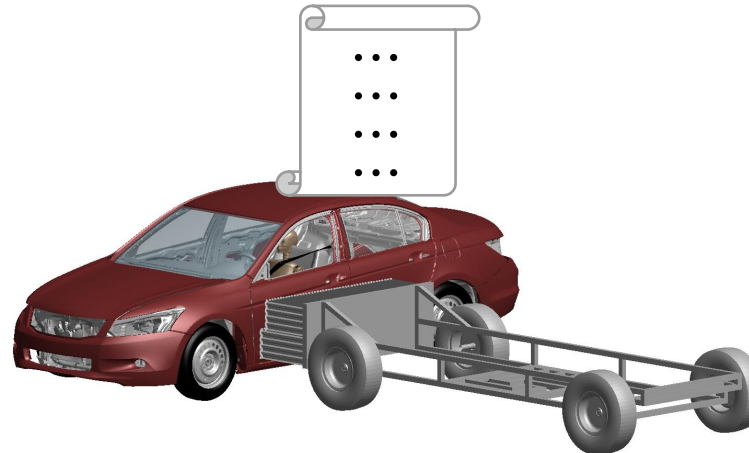
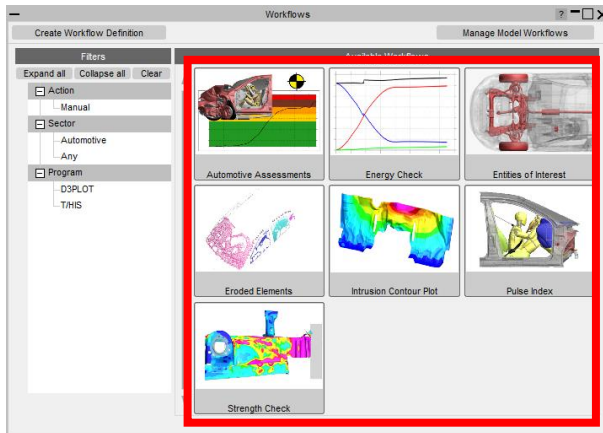
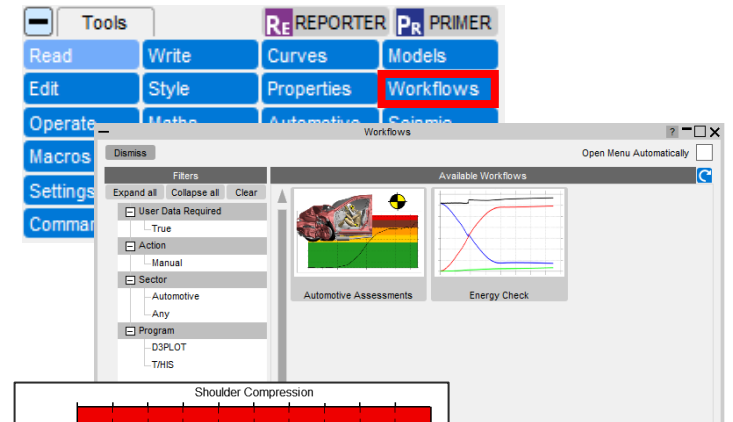
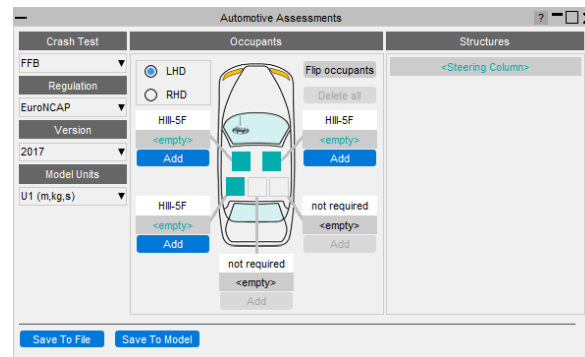
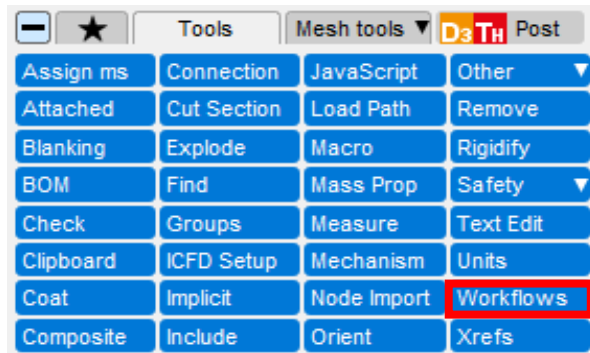
1. Select in PRIMER



2. Tag the model



3. Use in D3PLOT or T/HIS



Workflows

This release contains a collection of Workflows (described in more detail in further slides):

We plan to update these tools in future releases, as well as adding new tools.

If you have any ideas for additional features you would like to see in the existing tools or any ideas for new tools, please [contact us](#).

Workflow	Description
<u>Automotive Assessments</u>	Process crash and occupant safety simulations according to legal regulations and consumer safety NCAP protocols
<u>Energy Check</u>	Plot total, kinetic, internal and hourglass energy
<u>Entities of Interest</u>	Show and highlight groups of parts and export them to GLB files to view in D3PLOT Viewer
<u>Eroded Elements</u>	Visualise deleted elements at different states
<u>Intrusion Contour Plot</u>	An intrusion contour plot of selected parts (e.g. vehicle cockpit intrusion)
<u>Pulse Index</u>	Performs a vehicle pulse index calculation
<u>Strength Check</u>	Visualise yielding parts

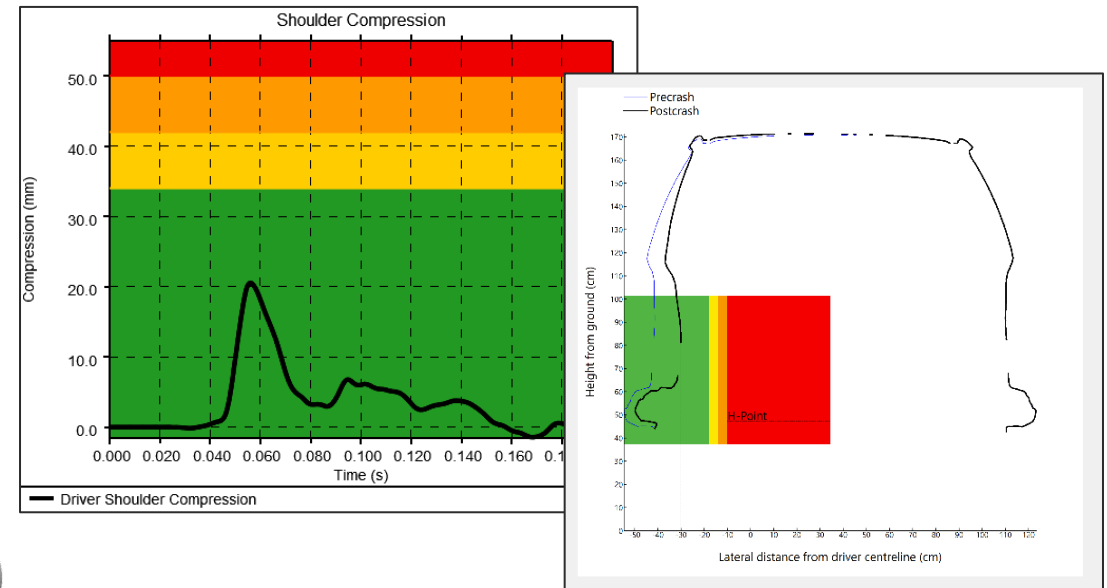
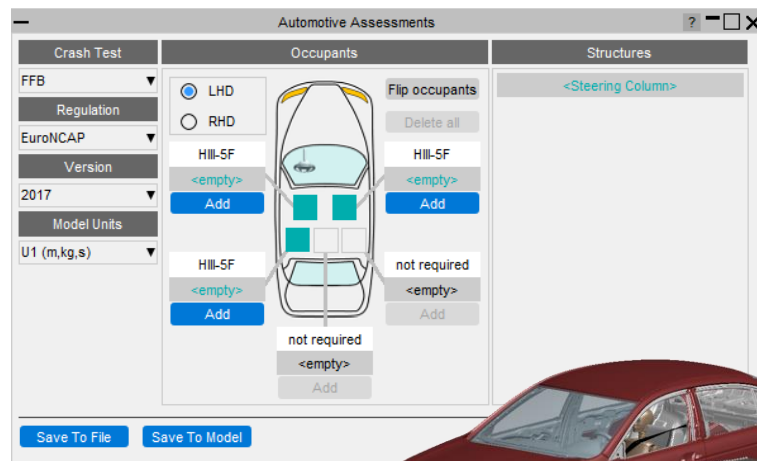
Workflows: Automotive Assessments

The **Automotive Assessments** workflow tool is used to post-process analyses according to various crash test regulations.

1. In **PRIMER**, you select the crash test type, regulation, and the occupants and structures you wish to assess



2. **D3PLOT** and **T/HIS** can then plot results for the different assessments required by the regulation



Workflows: Automotive Assessments

When the Automotive Assessments tool is launched in PRIMER you are presented with a menu that lets you specify the crash test type and the occupants and structures you wish to assess:

The screenshot shows the 'Automotive Assessments' window with three main panels: 'Crash Test', 'Occupants', and 'Structures'. A red box highlights the 'Crash Test' panel, which includes dropdowns for 'Regulation' (set to EuroNCAP), 'Version' (set to 2017), and 'Model Units' (set to None). A callout points to this panel with the text: 'Set the crash test type, regulation, version and model units'. Another red box highlights the 'Occupants' panel, which features radio buttons for 'LHD' (selected) and 'RHD', a car diagram, and lists of occupant types (HIII-SF, <empty>, not required) with 'Add' buttons. A callout points to this panel with the text: 'Define the occupants in the model'. A third red box highlights the 'Structures' panel, which shows a list of structures (<Steering Column>) with an 'Add' button. A callout points to this panel with the text: 'Define the structures in the model'. At the bottom of the window are 'Save To File' and 'Save To Model' buttons.

Set the crash test type, regulation, version and model units

Define the structures in the model

Define the occupants in the model

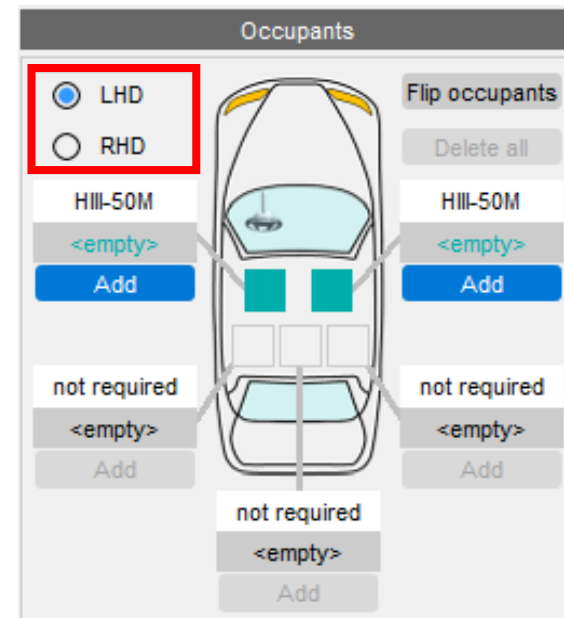
Workflows: Automotive Assessments

The Occupants section is used to select which occupants are in the vehicle, their positions in the vehicle, and the IDs of the entities from which data can be read.

This section updates automatically to show the required occupants in each position for the selected crash test type and regulation.

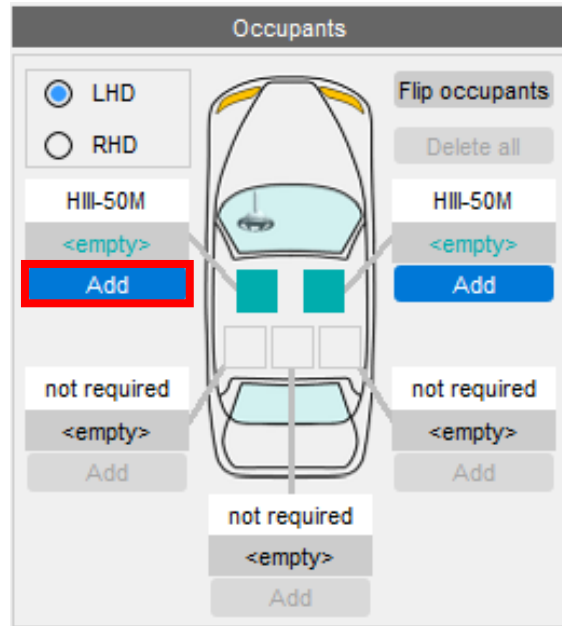
The first thing to do is select whether the vehicle is left hand or right hand drive (LHD or RHD).

The vehicle image will update to show the steering wheel on the correct side and the occupant positions will also update.



Workflows: Automotive Assessments

1. For each occupant, click **Add**



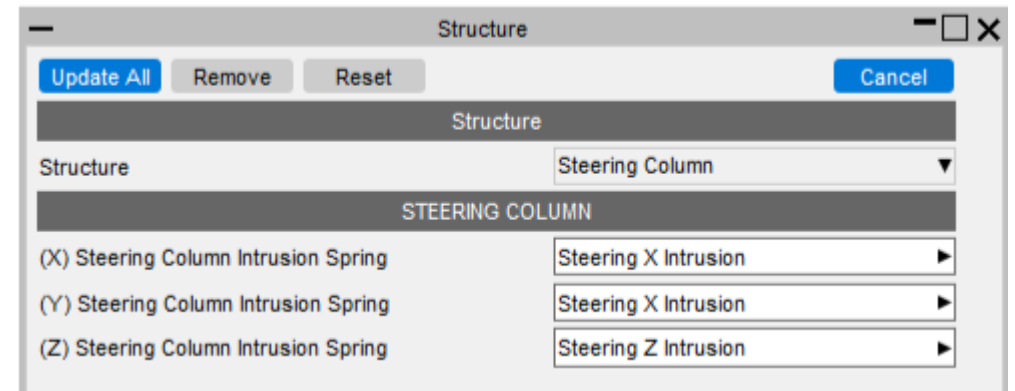
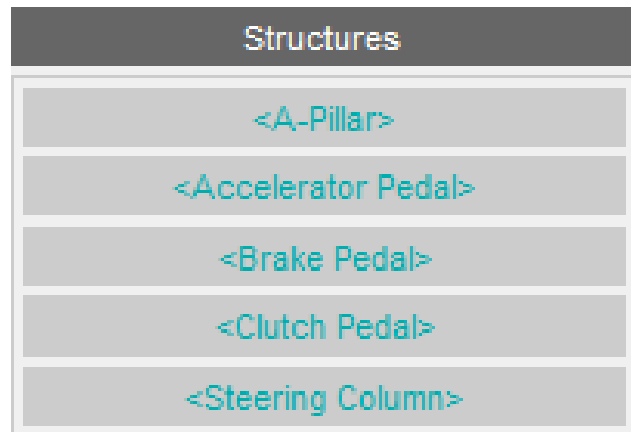
2. This will open a window where you can select the occupant type. Input entity IDs are filled automatically from a list of popular suppliers.

A screenshot of the 'Occupant' configuration window, showing detailed settings for an occupant. The window has a title bar with 'Occupant' and standard window controls. It includes an 'Add' button and a 'Cancel' button. Below these are 'Occupant Filters' for 'Supplier' (all), 'Product' (HIII), 'Physiology' (50M), and 'Occupant Name' (LSTC HIII 50M Detailed 190217 Beta). The 'Position' is set to 'Driver'. There are 'Entity IDs' and 'Offset for IDs' (0) fields. A 'Toggle Entity IDs' section has 'Use ID numbers' selected. The main body of the window is divided into sections for different body parts: HEAD, NECK, CHEST, FEMUR, KNEE, TIBIA, and FOOT. Each section contains a list of sensors and their corresponding entity IDs. For example, under HEAD, there are three 'Head Node' entries (X, Y, Z accelerometer) all set to 'HeadAccel_GLOBAL_AXES'. Under CHEST, there are four entries: 'Chest Transducer (Spring)' set to 'ChestDeflRotation_INJURY', and three 'Chest Node' entries (X, Y, Z Accn) all set to 'PelvisAccel_GLOBAL_AXES'. Under FEMUR, there are two 'Femur Loadcell' entries (X-Section) set to '50500001' and '50600001'. Under KNEE, there are two 'Knee Transducer' entries (Joint) set to '50700002' and '50800002'. Under TIBIA, there are four 'Tibia Loadcell' entries (X-Section) set to '50700003', '50800003', '50700002', and '50800002'. Under FOOT, there are two 'Foot Node' entries set to '10175835' and '10180679'.

Note that if your model does not have an occupant (or you don't want to carry out an assessment on it) you can leave it empty. T/HIS and D3PLOT will only attempt to process results for occupants that have been added.

Workflows: Automotive Assessments

- The Structures section is used to select the IDs of the entities from which data can be read from structures in the vehicle.
- This section updates automatically to show the required structures for the selected crash test type and regulation.



Workflows: Automotive Assessments

When launched in T/HIS, a menu appears, allowing you to select which occupant injuries and which structural checks you wish to perform:

Select which occupant injury assessments to plot.

The list of available occupants will depend on what was selected in PRIMER

Options to control how graphs are created and displayed

The screenshot shows the 'Automotive Workflow POST' window. It has a title bar with a question mark, minimize, maximize, and close buttons. The main area is divided into several sections. At the top, there's a 'Crash Test: ODB' section. Below it, there are four main selection areas: 'Regulation' (EuroNCAP, Rating Version: 2017, Unit Systems: M1 - U2 (mm,T,s)), 'Occupants' (Driver, Front-passenger), 'Body Parts' (HEAD, NECK, CHEST, FEMUR, KNEE), and 'Occupant Assessment Types' (CHEST_COMPRESSION, CHEST_VISCOUS_CRITERION, LEFT_FEMUR_COMPRESSION_EXCEEDENCE, RIGHT_FEMUR_COMPRESSION_EXCEEDENCE). Below these are 'Structures' (A-Pillar, Accelerator Pedal, Steering Column) and 'Structure Assessment Types' (PEDAL_FORE_AFT_INTRUSION, PEDAL_VERTICAL_INTRUSION). At the bottom, there's an 'Options' section with four radio buttons: 'Graphs on same page' (selected), 'Overwrite existing graphs' (selected), 'Graphs on separate pages', and 'Append to existing graphs'. A 'Plot' button is below the options. At the very bottom is an 'Output' section which is currently empty.

Regulation	Occupants	Body Parts	Occupant Assessment Types
EuroNCAP	Driver	HEAD	CHEST_COMPRESSION
Rating Version	Front-passenger	NECK	CHEST_VISCOUS_CRITERION
2017		CHEST	LEFT_FEMUR_COMPRESSION_EXCEEDENCE
Unit Systems		FEMUR	RIGHT_FEMUR_COMPRESSION_EXCEEDENCE
M1 - U2 (mm,T,s)		KNEE	

Structures	Structure Assessment Types
A-Pillar	PEDAL_FORE_AFT_INTRUSION
Accelerator Pedal	PEDAL_VERTICAL_INTRUSION
Steering Column	

Options

☒ Graphs on same page ☒ Overwrite existing graphs
☐ Graphs on separate pages ☐ Append to existing graphs

Plot

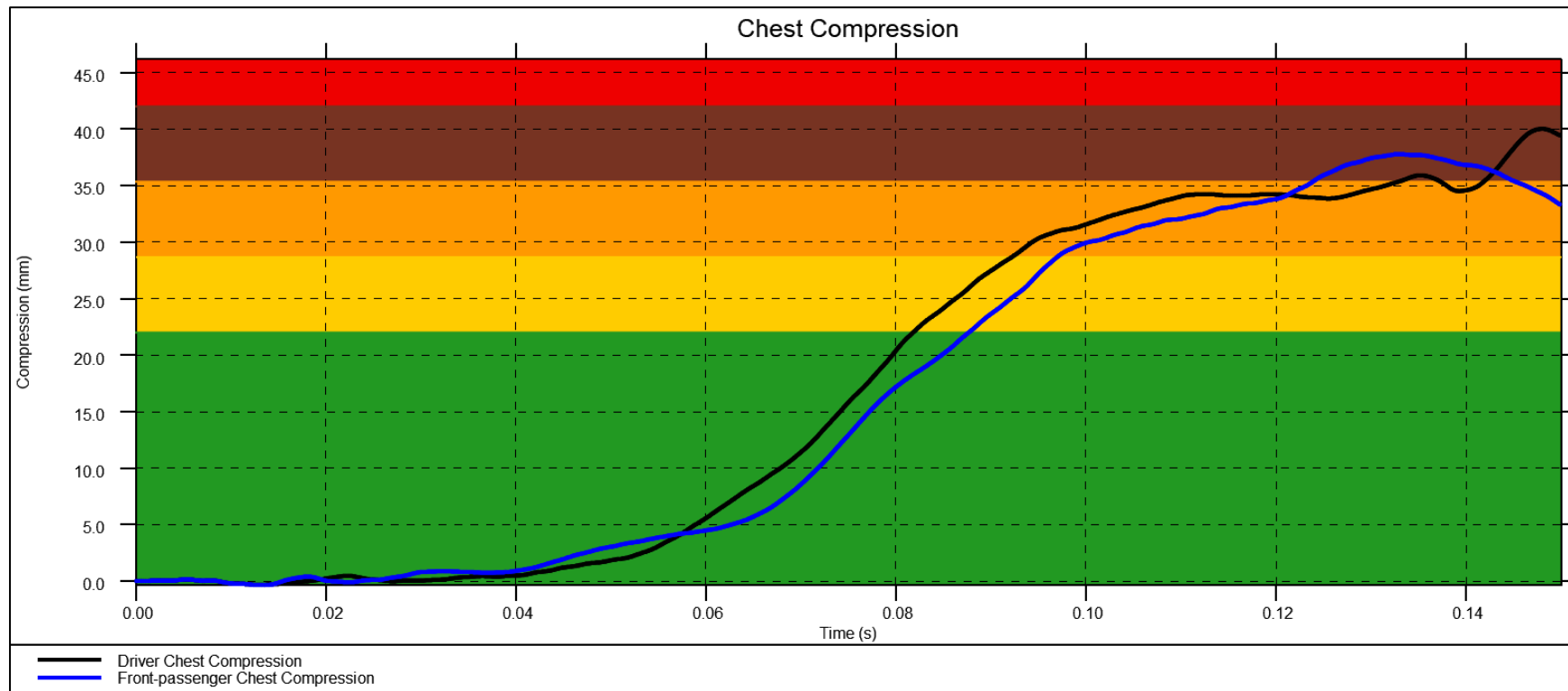
Output

Select structural assessments.

The list of available structures will depend on what was selected in PRIMER

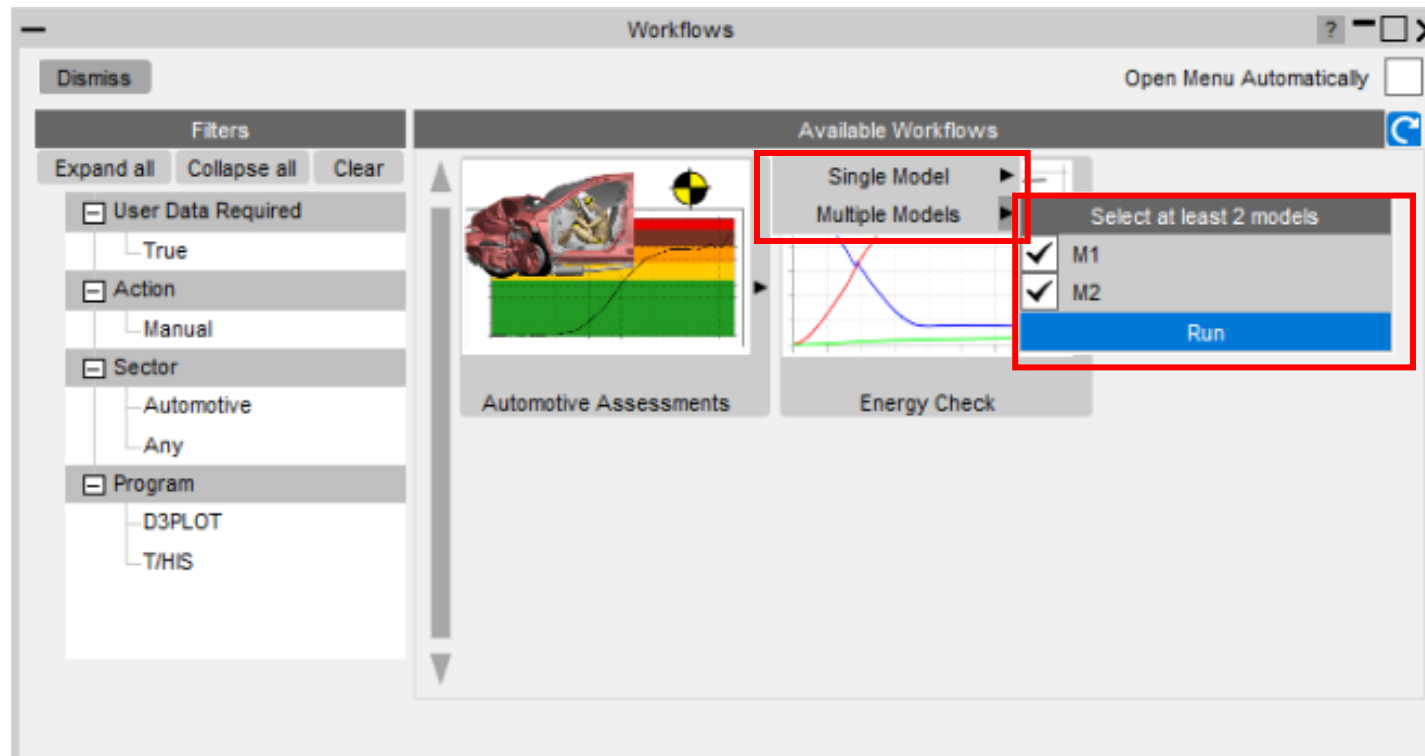
Workflows: Automotive Assessments

T/HIS will extract the data required, process it according to the rules set out in the regulation and plot the results on a graph with datums showing allowable limits:



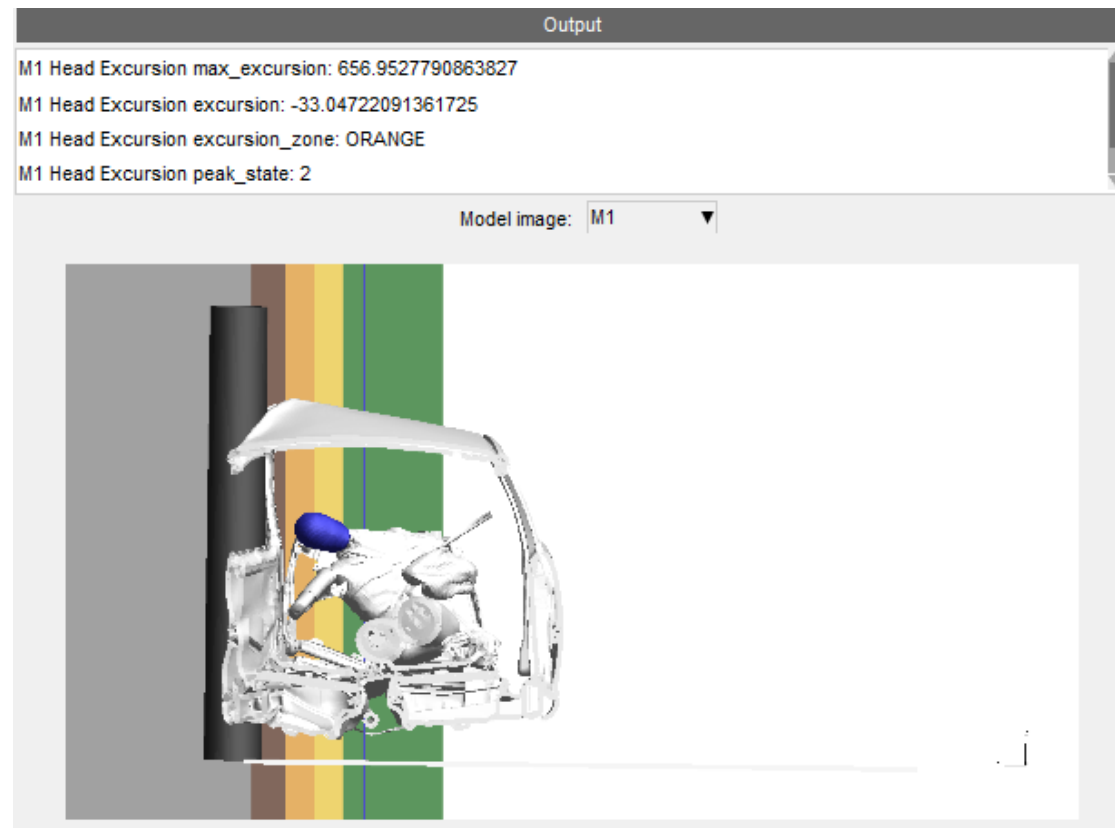
Workflows: Automotive Assessments

The tool allows you to plot results from multiple models. This is useful when you want to compare results between different runs:



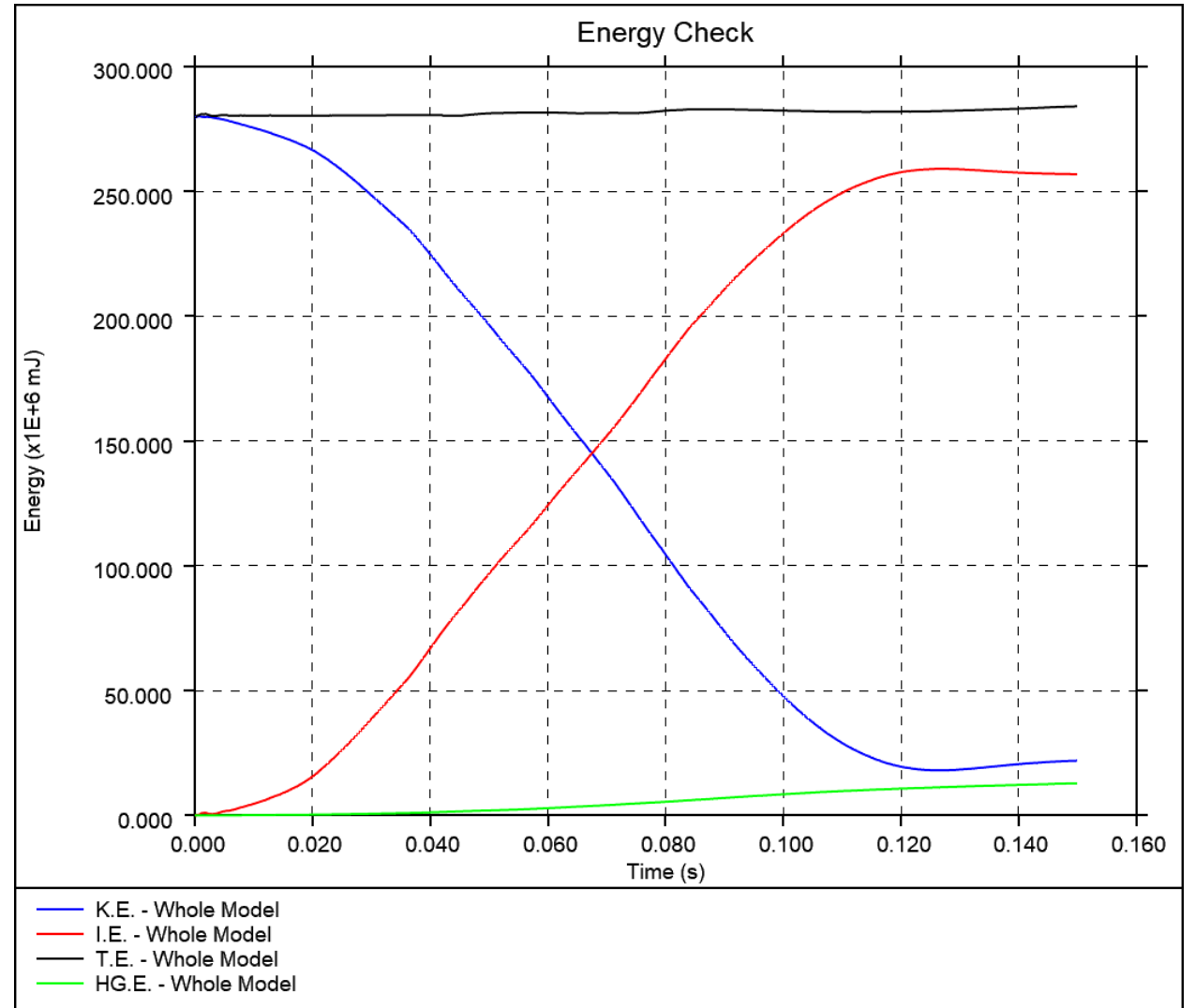
Workflows: Automotive Assessments

- When launched in D3PLOT, the Automotive Assessments Workflow will extract the data required for the assessment and process it according to the rules set out in the regulation.
- Example output from Euro NCAP Far Side assessment:



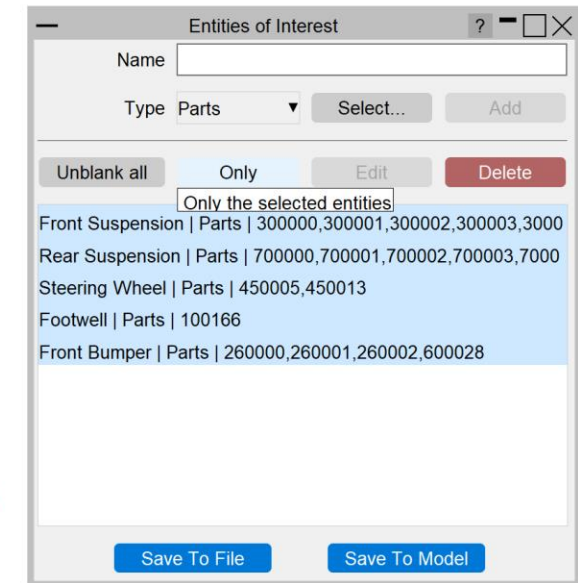
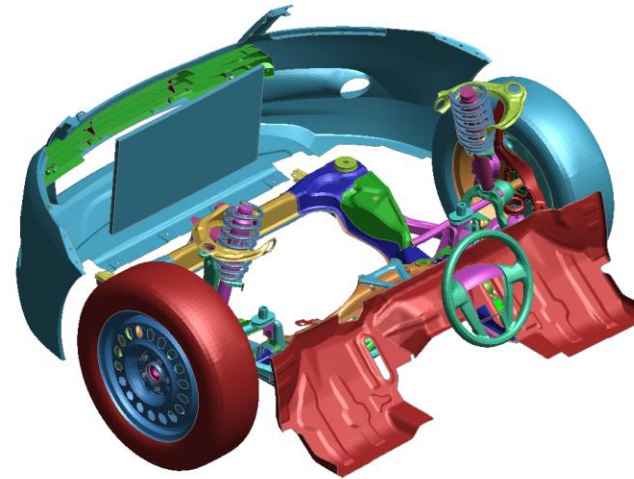
Workflows: Energy Check

The **Energy Check** Workflow is a quick tool to help plot global energies for your model. In PRIMER, select a unit system (to define the energy units plotted). Then in T/HIS, select the Energy Check Workflow to plot a graph of total energy, kinetic energy, internal energy and hourglass energy for your model.

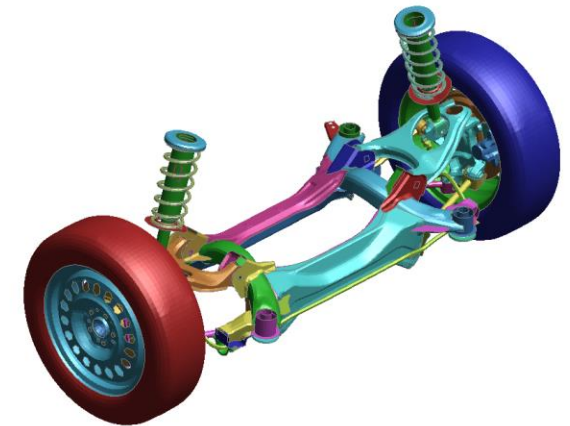


Workflows: Entities of Interest

- The Entities of Interest tool allows you to visualise specific groups of entities quickly in D3PLOT and action them.
- First, create a list of Entities of Interest in PRIMER – the list can comprise Parts and Part Sets.



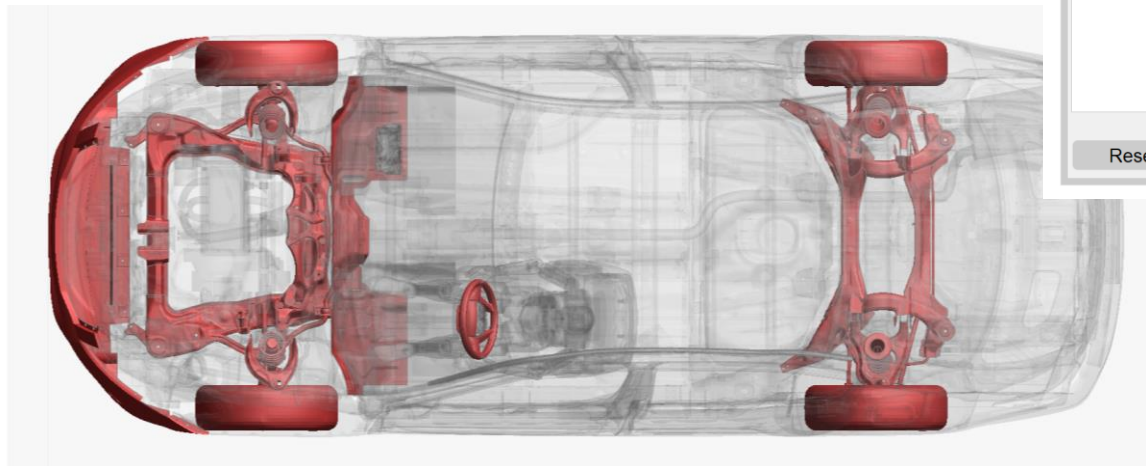
Only








Workflows: Entities of Interest

In D3PLOT, you can then execute various actions, including highlighting the selected entities, and exporting them as a set of GLB files for viewing in D3PLOT Viewer.

Highlight



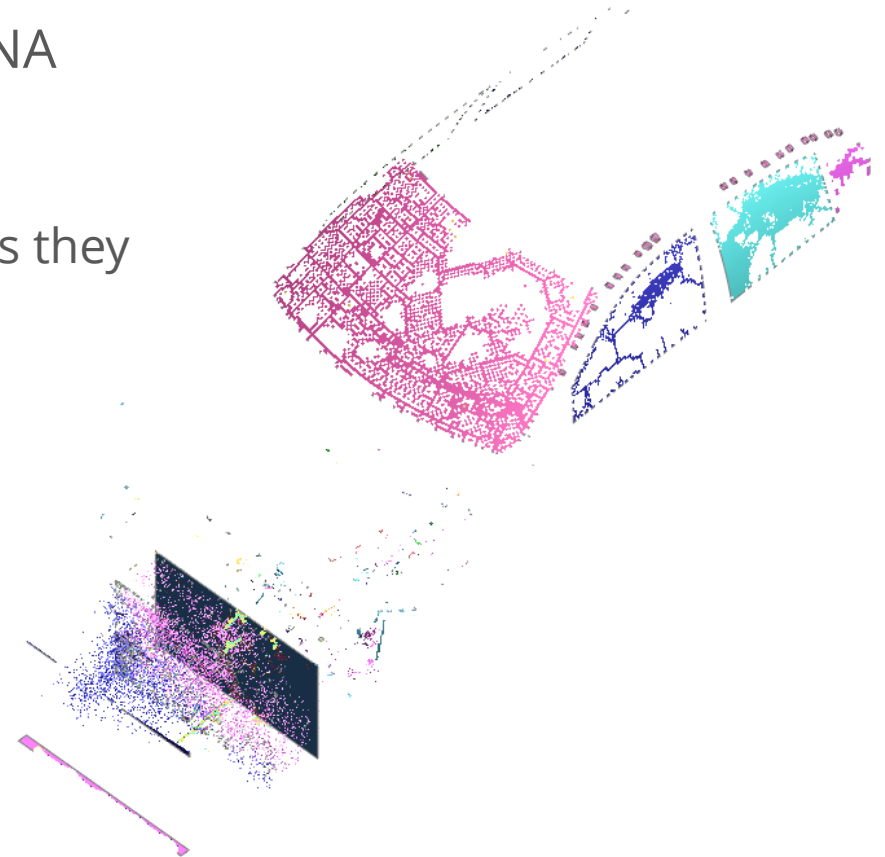
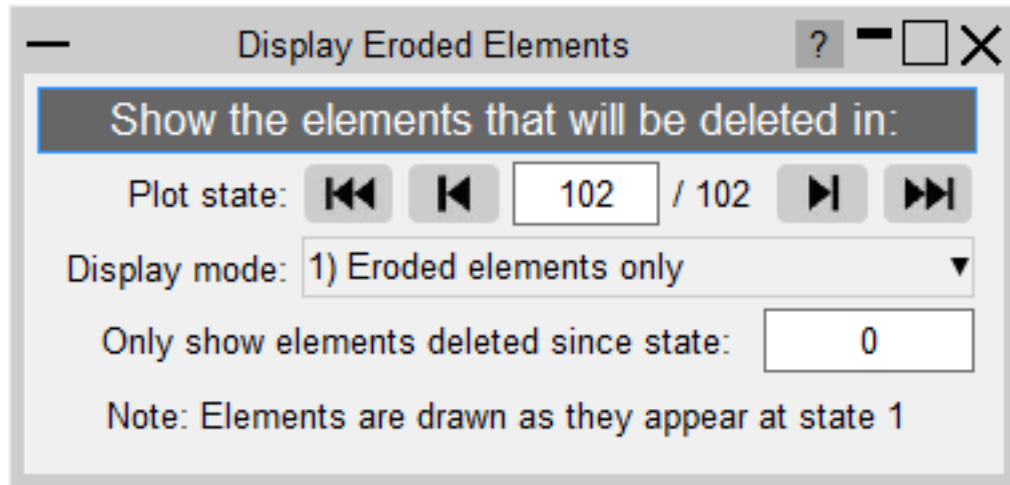
GLB Export

Name	Type
 Footwell.glb	3D Object
 Front Bumper.glb	3D Object
 Front Suspension.glb	3D Object
 Rear Suspension.glb	3D Object
 Steering Wheel.glb	3D Object

Workflows: Eroded Elements

The **Eroded Elements** Workflow provides a quick method of visualising eroded (deleted) elements in your LS-DYNA simulation.

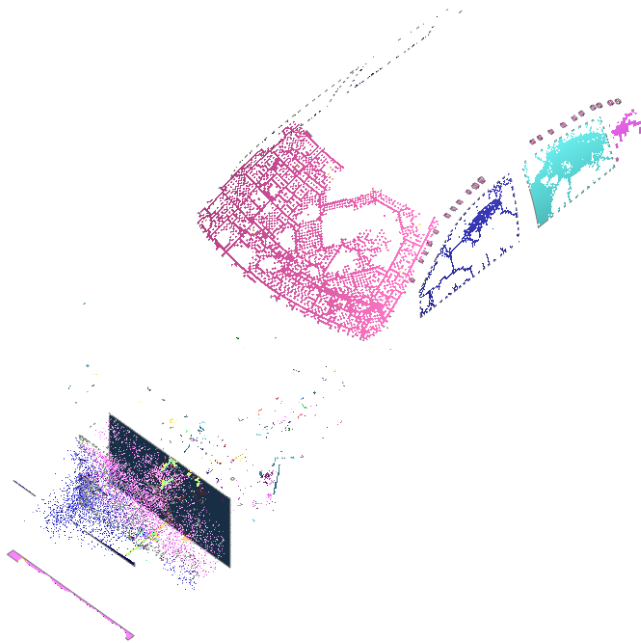
- Note that while using this tool, elements are drawn as they appear at state 1.



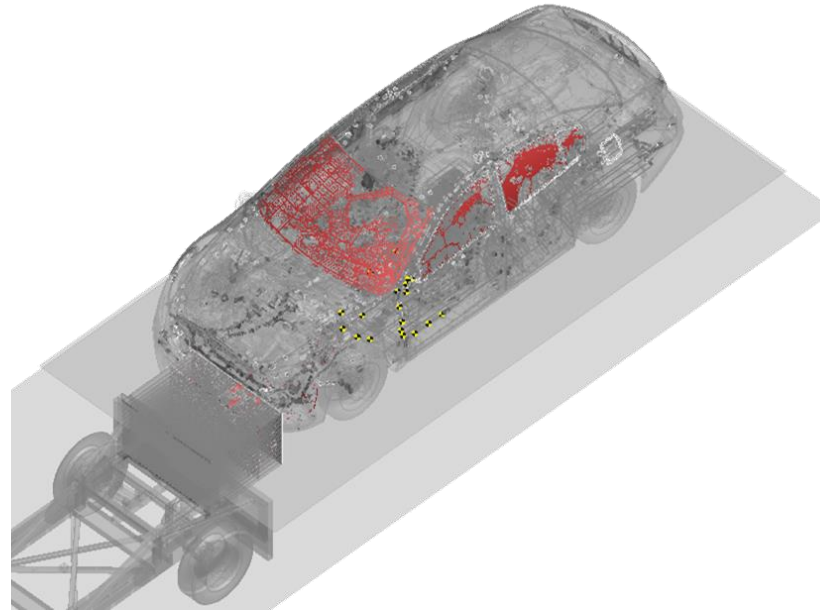
Workflows: Eroded Elements

There are three display modes:

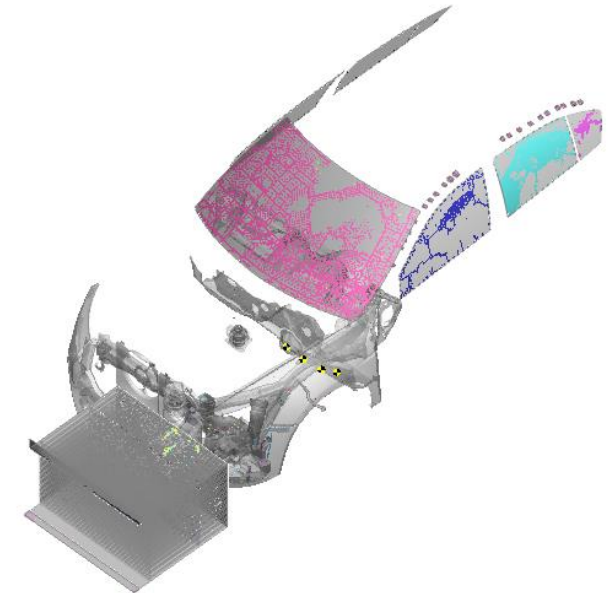
1. Eroded elements only (default)



2. Eroded elements in red

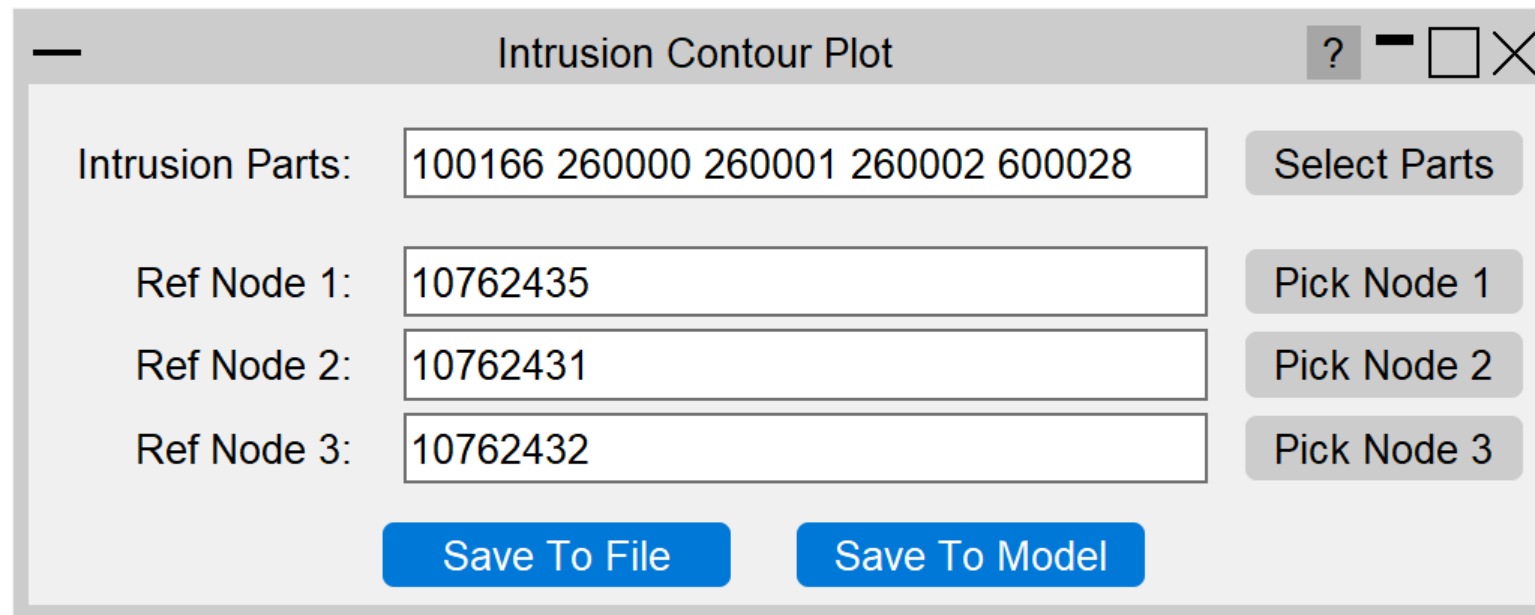


3. Parts containing eroded elements



Workflows: Intrusion Contour Plot

- The Intrusion Contour Plot tool creates a contour plot of intrusion displacements for selected parts, relative to specified reference coordinates.
- Configure the plot in PRIMER:



Intrusion Contour Plot

Intrusion Parts: 100166 260000 260001 260002 600028 Select Parts

Ref Node 1: 10762435 Pick Node 1

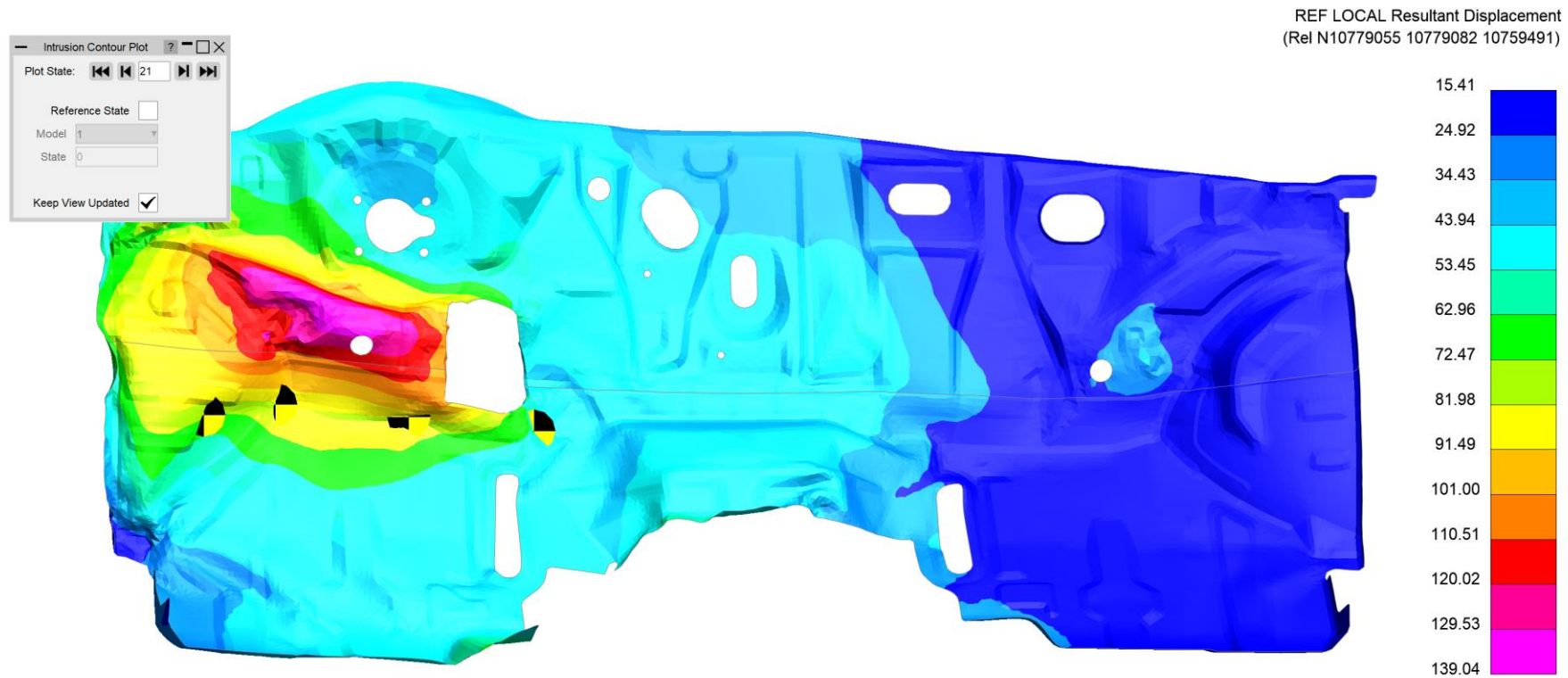
Ref Node 2: 10762431 Pick Node 2

Ref Node 3: 10762432 Pick Node 3

Save To File Save To Model

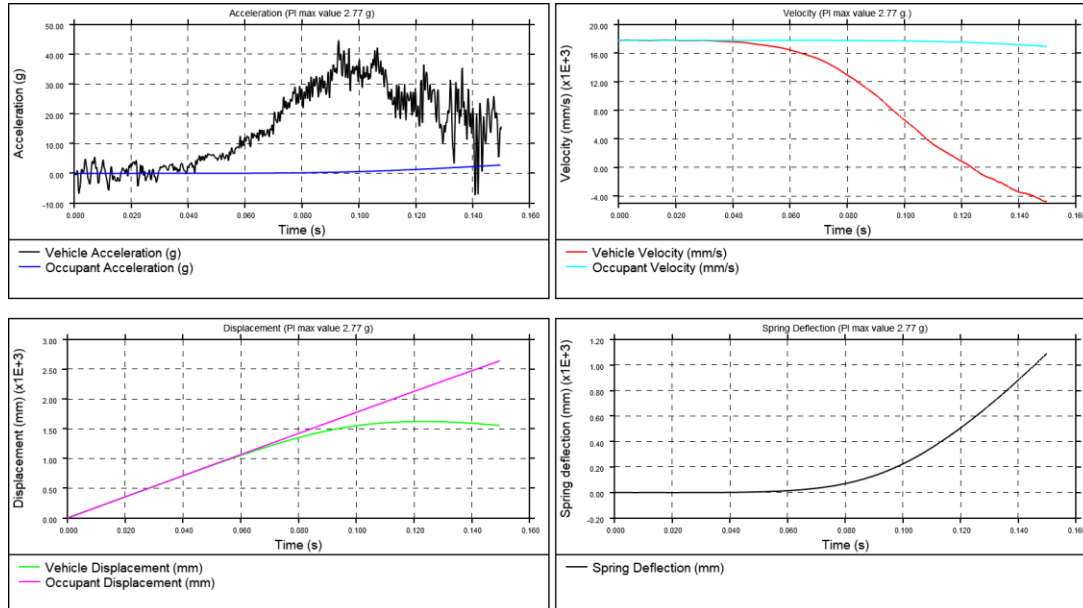
Workflows: Intrusion Contour Plot

In D3PLOT, you can control the plot state, plot the intrusion values relative to a reference state or different model, and control the view. Here is an example plot:



Workflows: Pulse Index

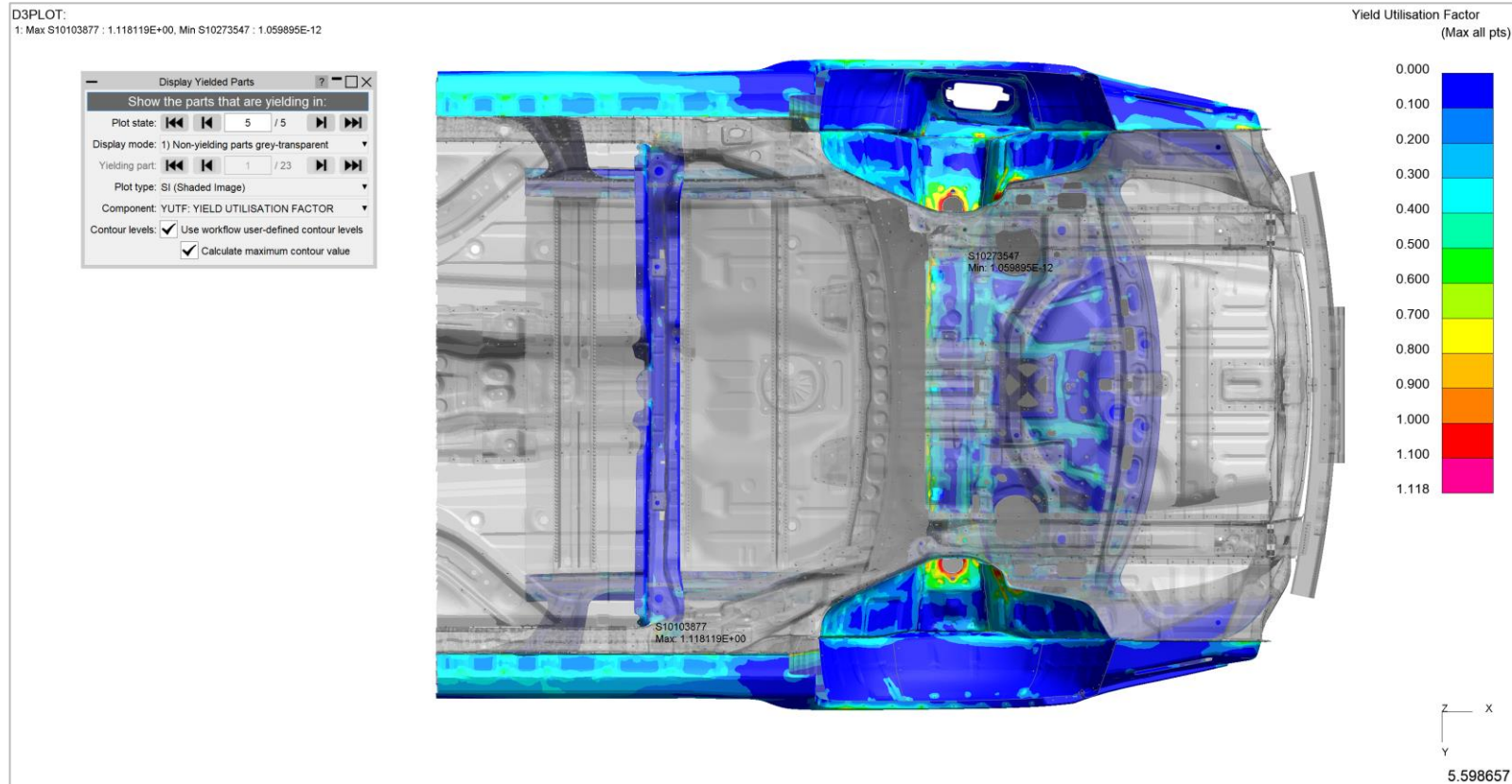
The screenshot shows the 'Pulse Index' software window. It has a 'Model' section with a 'Units' dropdown set to 'U2 (mm,T,s)'. The 'Parameters' section includes: 'Mass of occupant (tonne): 0.08', 'Initial velocity (mm/s): 17777.8', 'Restraint stiffness (N/mm):' with radio buttons for 'Constant' (selected) and 'Variable' (disabled), a text box with '2', and a disabled 'Select curve' button. 'Slack (mm): 0' and 'Measurement node: X-Axis' with a dropdown menu showing 'NID 50358124'. At the bottom are 'Save to file' and 'Save to model' buttons.



- During the early stages of vehicle development, it can be useful to understand occupant acceleration without needing to include a complex and computationally expensive occupant model.
- The **Pulse Index** Workflow allows you to estimate the acceleration that would be experienced by a vehicle occupant in a crash test scenario.
- The tool assumes a virtual single-degree-of-freedom mass-spring system in which the occupant is represented by a **mass** and the seatbelt is represented by a **spring**.
- This virtual mass-spring system is effectively attached to a selected **node** moving with a set **initial velocity**.
- The tool optionally takes a **slack** input to account for seatbelt engagement.
- Occupant parameters are computed and visualised in T/HIS.

Workflows: Strength Check

The Strength Check tool allows you to visualise *yielding parts in D3PLOT:

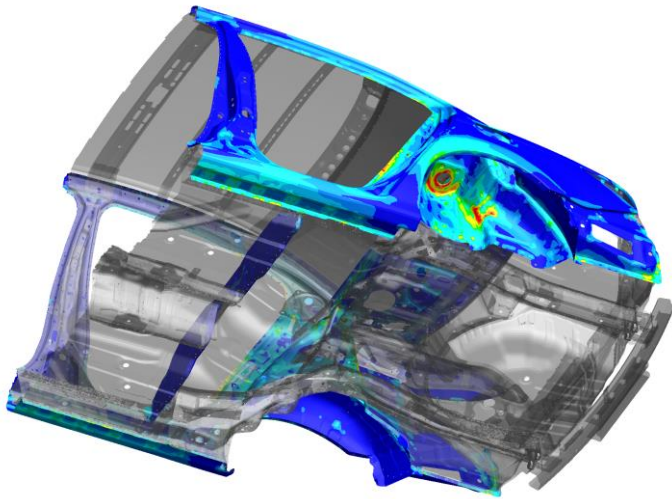


***Note:** Yielding parts are parts containing at least one yielding element, measured as an element with a [Yield Utilisation Factor](#) greater than one (or [Yield Utilisation Percentage](#) greater than 100%).

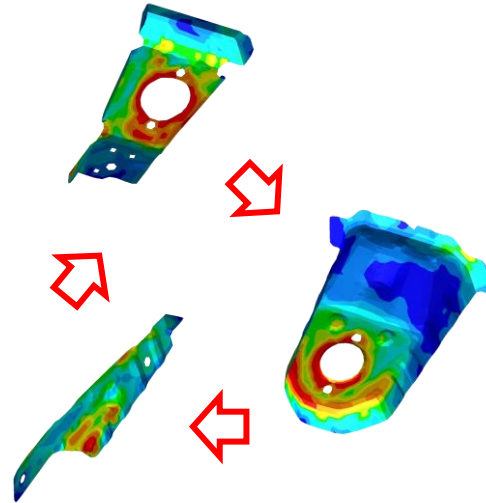
Workflows: Strength Check

There are three display modes:

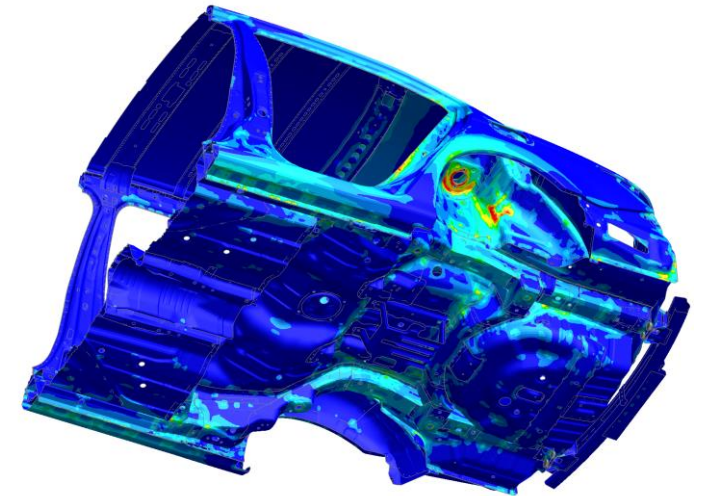
**1. Non-yielding parts
transparent-grey**



**2. Cycle through
yielding parts**



**3. Plot of entire
model**



Workflows: Strength Check

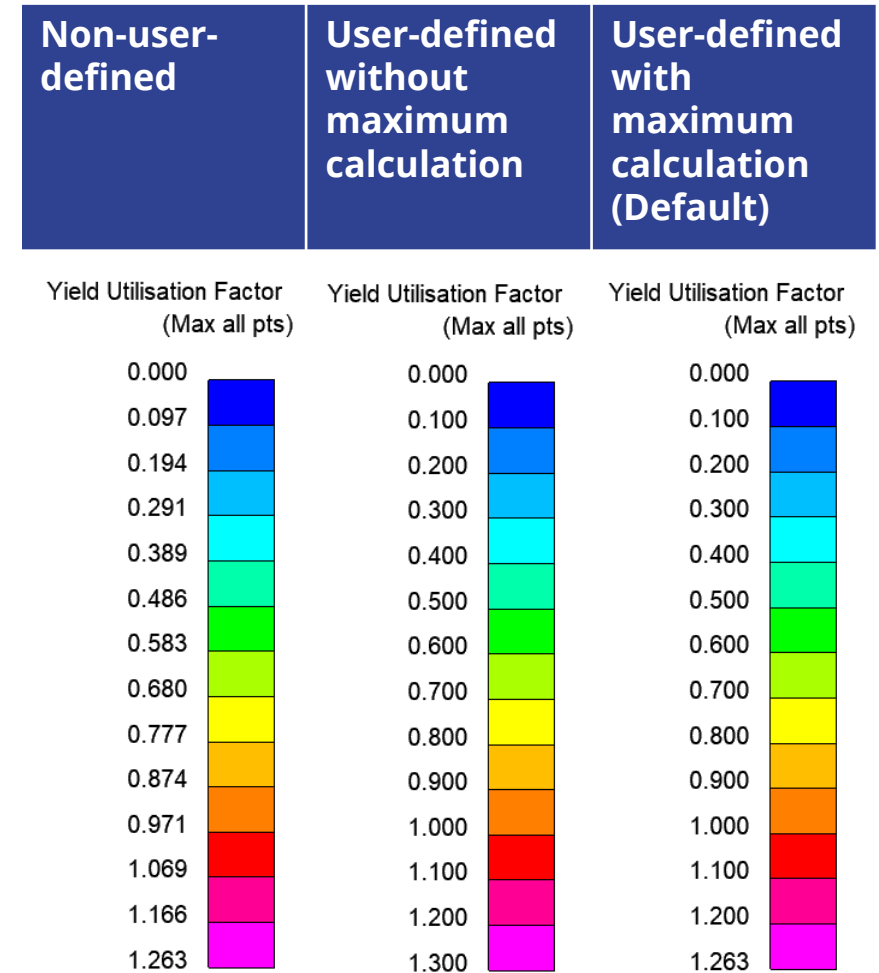
You can also control the contour bar levels:

- **Use workflow user-defined contour levels**

To better visualise yielding parts, this option is enabled by default. The contour bar has values from 0.0 to 1.0 (blue to dark orange) in increments of 0.1, and then three further contour levels in red and magenta, so that all yielding material is clearly indicated in red/magenta. If this option is unchecked, automatic contour levels will be used.

- **Calculate maximum contour value**

With the above option enabled then by default, the actual maximum data value will be calculated for the maximum contour level. For example, if the maximum yield utilisation factor is 1.263 then the contour levels above 1.0 will be 1.1, 1.2 and 1.263. This makes the maximum yield utilisation clearer. You can uncheck this option to speed up the plot (skips the maximum value calculation). In this case, the contour levels above 1.0 will be fixed to 1.1, 1.2 and 1.3.



Workflows – Creating Custom Workflows

In addition to the Workflows provided, you can also add your own custom workflows.

Workflows are written in JavaScript with three steps required to create them:

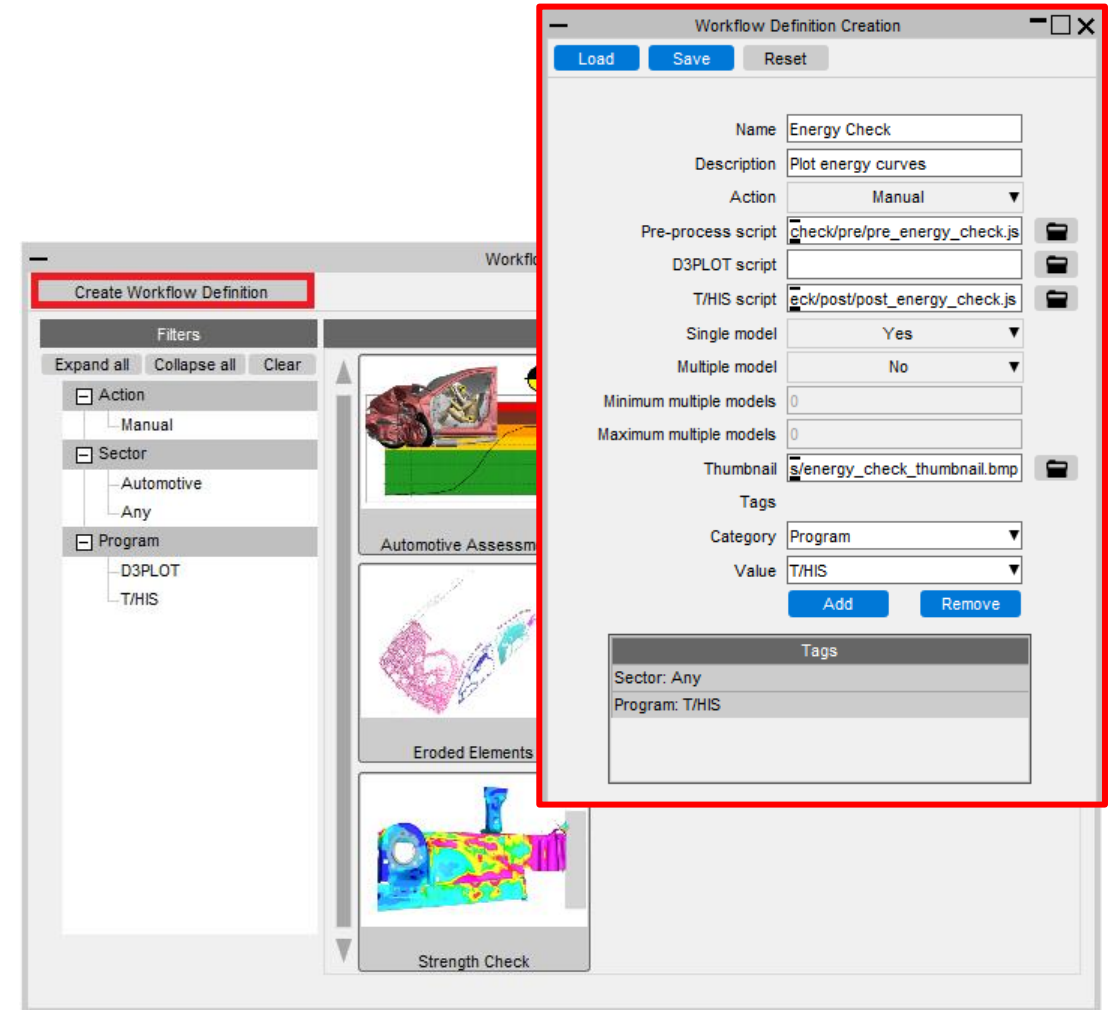
1. Create a **Workflow Definition** file to describe what the workflow does and point to the scripts to run in PRIMER, D3PLOT and/or T/HIS
2. Create a script to run in PRIMER that will collect data from user inputs and write it to a file or save it in the keyword file as post *END data
3. Create scripts to run in D3PLOT and/or T/HIS that will read the data and use it to carry out any appropriate actions

If you have an idea for a tool and would like some help creating it, have a look at the following slides, or please [contact us](#).

Workflows – Creating Custom Workflows

1. Create a Workflow Definition file...

- The Workflow Definition file is a JSON file containing information about the tool and which scripts it should run.
- It can be created by hand or by using the 'Create Workflow Definition' function in the PRIMER Workflow menu.

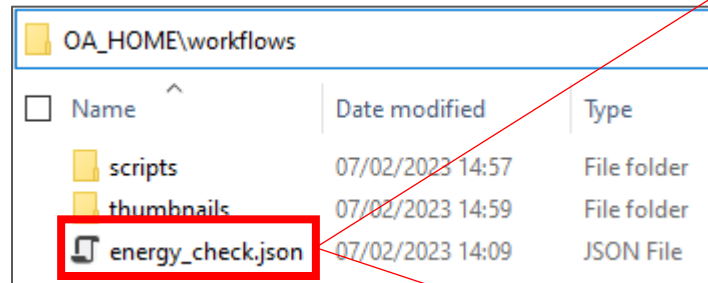


Workflows – Creating Custom Workflows

1. Create a Workflow Definition file...

The Workflow Definition file needs to be saved in one of the following directories:

- The directory set by the ***oasys*workflow_definition_directory*** preference
- ***OA_HOME/workflows***
- ***OA_INSTALL/workflows***
- ***OA_ADMIN/workflows***



```
{
  "name": "Energy Check",
  "description": "Plot energy curves",
  "user_data_required": true,
  "pre_process_script": "scripts/energy_check/pre/pre_energy_check.js",
  "t_his_script": "scripts/energy_check/post/post_energy_check.js",
  "action": "Manual",
  "single_model": true,
  "multiple_models": false,
  "thumbnail": "thumbnails/energy_check_thumbnail.bmp",
  "tags": {
    "Sector": "Any"
  }
}
```

Workflows – Creating Custom Workflows

2. Create a script to run in PRIMER...

The script to run in PRIMER needs to collect data from user inputs and then save it to a JSON file or as post *END data in the keyword file.

The **Workflow** JavaScript API class has helper methods to make it easy to save the data:

```
Workflow.WriteToModel()
```

```
Workflow.WriteToFile()
```

Please see the JavaScript API manual for further information.

Workflows – Creating Custom Workflows

3. Create scripts to run in D3PLOT and/or T/HIS...

The scripts to run in D3PLOT and/or T/HIS need to read the data saved with the model and then use it to provide the desired functionality.

The **Workflow** JavaScript API class has helper methods to make it easy to read the data for the models selected:

```
Workflow.NumberOfSelectedModels()
```

```
Workflow.ModelIdFromIndex()
```

```
Workflow.UserDataFromIndex()
```

```
Workflow.UnitSystemFromIndex()
```

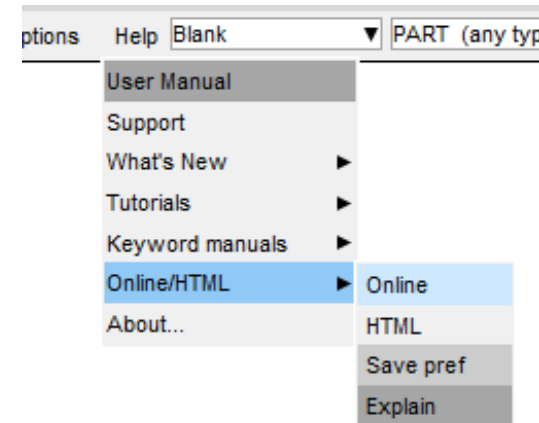
Please see the JavaScript API manual for further information.

If you have an idea for a tool and would like some help creating it, please [contact us](#).

Help & Documentation

Help & Documentation

- The manuals are now online by default:
 - Powerful search tool;
 - Google translation available;
 - Continuous updates and fixes.
- A local HTML version of the manuals can still be used:
 - Useful if there there is no Internet access;
 - Local or online can be saved via a preference:
oasys*online_manuals: TRUE/FALSE.
- The new JS API manual combines documentation for PRIMER, D3PLOT, T/HIS and REPORTER:
 - i.e. D3PLOT, T/HIS and REPORTER JS-API documentation is no longer in separate appendices;
 - Local HTML and PDF versions are available;
 - The new JS API manual is not yet online.



Miscellaneous

*LOAD_BODY includes

- You can now move the separate variables within the *LOAD_BODY keyword panel to separate include files.
- The button in the top-right corner controls the include file of all the variables; the browse buttons next to the variable names moves the individual variable.
- The text box in the top right displays the include status of the active variables.

PSID
0

LCID	SF	LCIDDR	CID
0	0.0	0	0

LCID	SF	LCIDDR	CID
0	0.0	0	0

LCID	SF	LCIDDR	CID
0	0.0	0	0

LCID	SF	LCIDDR	XC	YC	ZC	CID
0	0.0	0	0.0	0.0	0.0	0

LCID	SF	LCIDDR	XC	YC	ZC	CID
0	0.0	0	0.0	0.0	0.0	0

LCID	SF	LCIDDR	XC	YC	ZC	CID
0	0.0	0	0.0	0.0	0.0	0

LCID	SF	LCIDDR	XC	YC	ZC	CID
0	0.0	0	0.0	0.0	0.0	0

V1	V2	V3
0.0	0.0	0.0

Miscellaneous

- When reading in dynain files, the treatment of severe errors is temporarily set to 'Skip & Continue'. Controlled by a new preference, 'skip_severe_errors_dynain'.
- Quick Pick for *INITIAL_AXIAL_FORCE_BEAM and *INITIAL_STRESS_SECTION added.
- DEFINE_CURVE warning and usage comments no longer written when xref comments turned off, reducing file size.
- Added ability to write out in free format for all keywords (LS-DYNA output options -> Options2). Nothing will be written between the commas if the data in the field is zero, which reduces file size.
- Vector magnitude is displayed in *DEFINE_VECTOR create panel.
- When adding points in DEFINE_CURVE and adding entries in DEFINE_TABLE, the graph is now auto scaled automatically (Can be disabled with the toggle button 'Autoscale').
- Ambiguous pick menu can now be dismissed using the 'X' button at the top right corner of the pick menu or using a Esc key on the keyboard.

Contact Information



www.arup.com/dyna

For more information please contact us:

UK

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

China

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

India

T: +91 40 69019723 / 98
india.support@arup.com

USA West

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

or your local Oasys distributor