

PRIMER 16.0

PRIMER 16.0 - Contents

- [Keyword Input and Output](#)
- [Improvements for Large Models](#)
- [Morphing](#)
- [Meshing Improvements](#)
- [Shell Split](#)
- [Geometry Mesh](#)
- [Swages Updates](#)
- [ICFD Setup](#)
- [Displaying Shell Normals](#)
- [ISO Geometric Analyses](#)
- [*CONSTRAINED](#)
- [Connections](#)
- [Multi Stage Positioning](#)
- [Seatsquash](#)
- [Undo for Renumbering](#)
- [Adding/Replacing and Include File](#)
- [New Orient Translate Option](#)
- [New Options for Orient Dialogue Command](#)
- [Rigidwall Visualisation](#)
- [Rigidify Panel Options](#)
- [Display and Cut-Section Options](#)
- [FMH Markup Tool](#)
- [Pedestrian Markup Tool](#)
- [Images](#)
- [JavaScript API](#)
- [Model Check](#)
- [Element Quality Checks](#)
- [LS-DYNA Output Files](#)
- [PTF/d3plot Write](#)
- [Preferences](#)
- [Licensing 16.0](#)

Keyword Input and Output

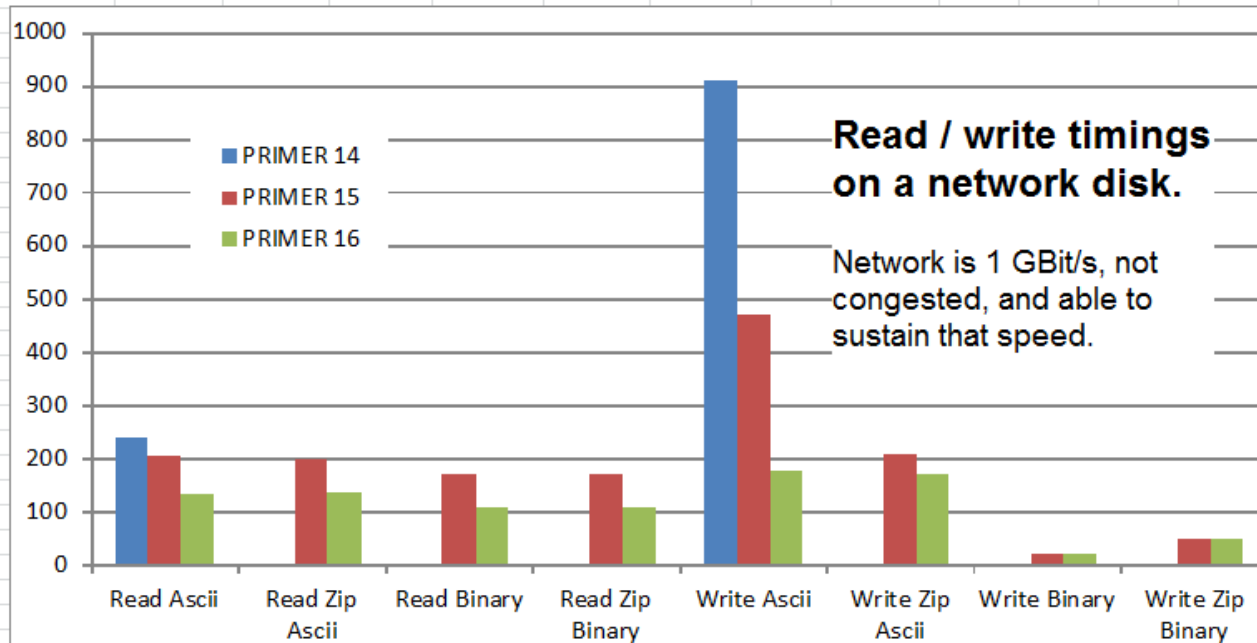
Input and Output Improvements

- Keyword input and output significantly faster for large models due to increased parallelisation of i/o and other improvements. Data are i/o times in seconds.

File / Version	Read Ascii	Read Zip Ascii	Read Binary	Read Zip Binary	Write Ascii	Write Zip Ascii	Write Binary	Write Zip Binary			
PRIMER 14	240				911						
PRIMER 15	205	200	171	170	468	208	23	49			
PRIMER 16	134	136	109	108	175	170	21	48			

File sizes:

- Raw ascii 5.0 GBytes
- Zipped ascii 1.3 GBytes
- Raw binary 1.6 Gbytes
- Zipped binary 0.8 Gbytes



Input and Output Improvements

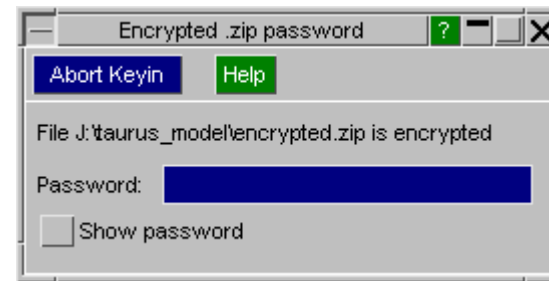
- Zipped (ascii and binary) i/o on a slow or congested network will be significantly better than raw i/o because smaller file sizes mean less network traffic.

Local disk i/o shows proportionally similar speed improvements.

- Can now read encrypted .zip format files.

Will detect encryption and prompt for a password

(Note: PRIMER does not *write* encrypted files.)



- Binary Keyword files now get the extension “.kby” to distinguish them from “.key”.
- When a directory contains multiple versions of an include file, for example:

child.key (Original file)

child.kby (File modified in PRIMER and written out in binary format)

PRIMER searches all variants and lists them, but reads the newest by default.

Other Improvements for Large Models

Other Improvements for Large Models

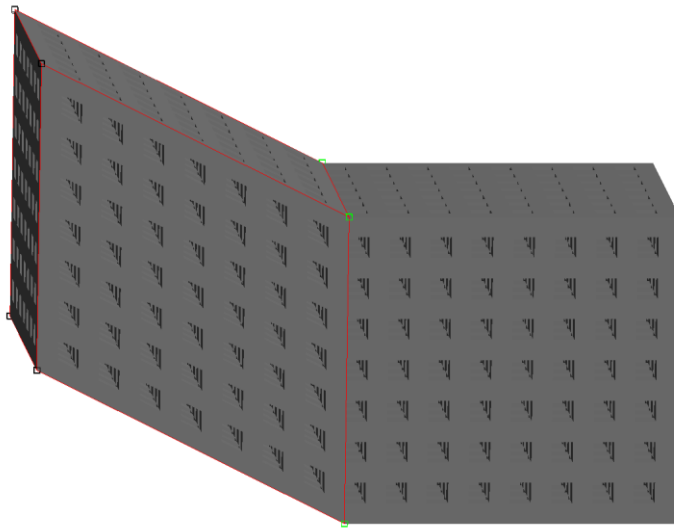
These improvements apply to models of all sizes, but will become more noticeable for models with > 10m elements and nodes:

- Response to “change of graphics” is faster:
 - Application of blanking and unblanking at least 2x faster.
 - Selection of parts, elements and nodes by area at least 2x faster.
- Cut-section display improved.
 - Speed of dragging a cut-section through a model dramatically (> 5x) faster.
- As with the improvements to Model Check and Keyword input most of the increase in speed comes from parallelisation. This trend will continue, and the larger the model the more it will benefit from a greater number of cores on the machine.

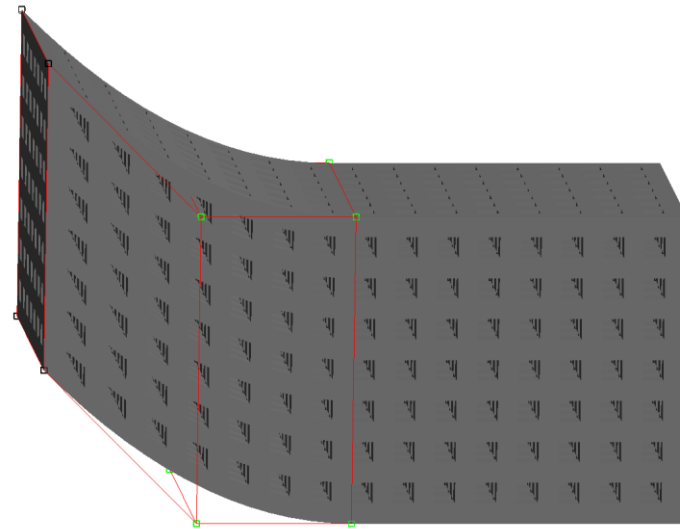
Morphing

High Order Morph Boxes

Morph boxes can now have more than 2 by 2 by 2 morph points. These high order morph boxes give you more control over mesh smoothness while morphing.

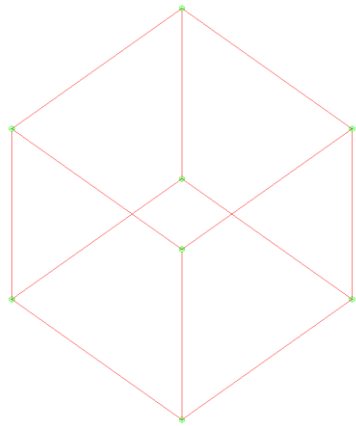


Basic order
box with 2 by
2 by 2 points

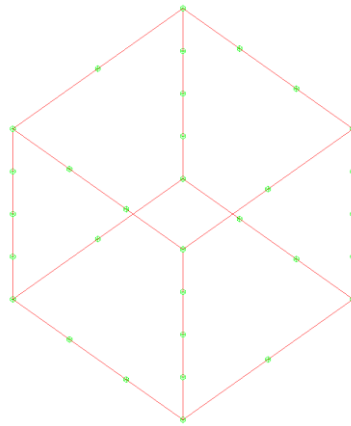


High order
box with 3 by
2 by 2 points

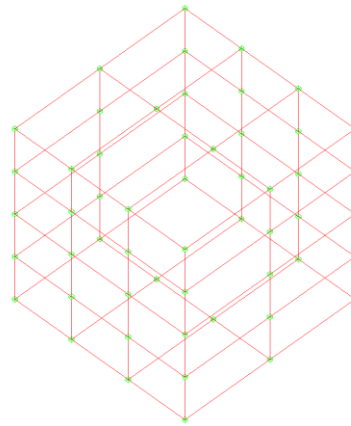
A Box With Different Morph Point Visibility



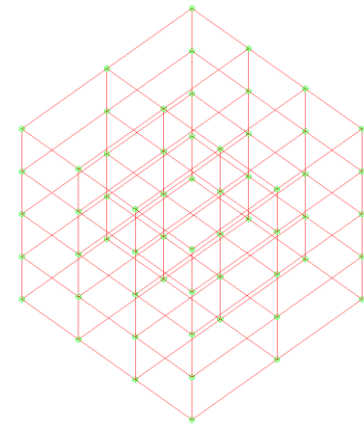
Corner points only



Corner and edge points visible

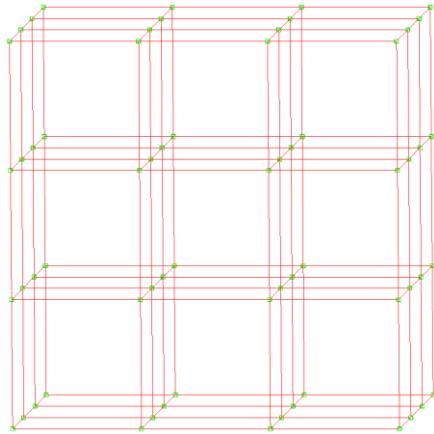


Corner, edge and face points visible

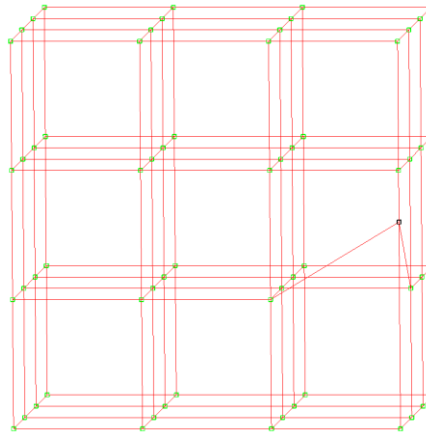


All morph points visible

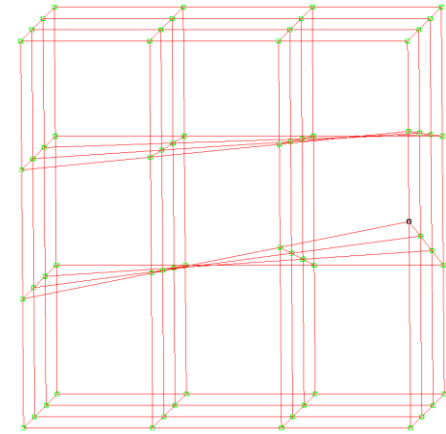
Morph Point Interpolation



Original position

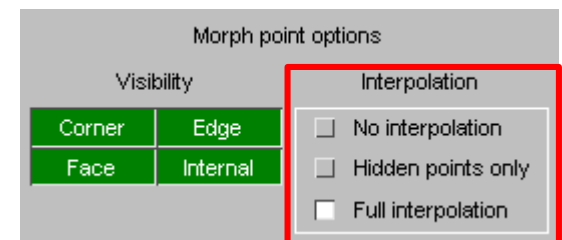


No interpolation



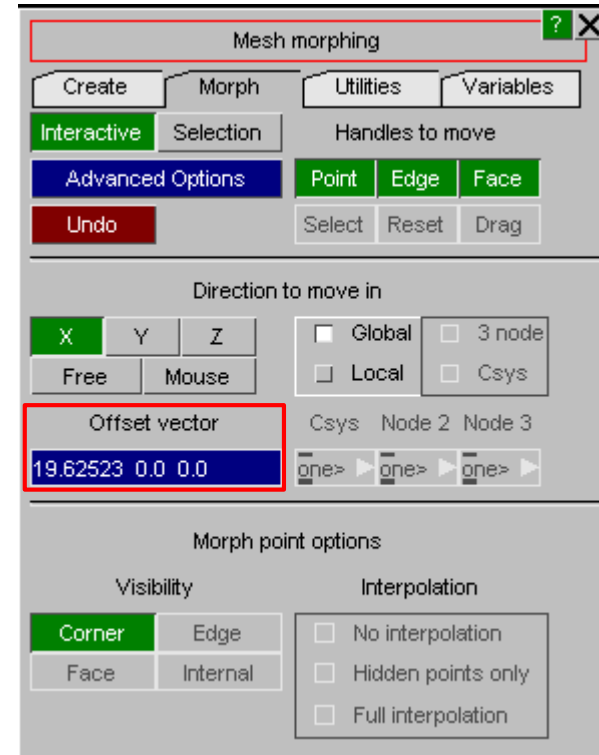
Full interpolation

Morph point movements can be interpolated. Without interpolation only the explicitly selected morph points move, whereas with interpolation others move as well by a fraction of the vector. This is illustrated in the pictures where only one edge point is explicitly selected. The option “Hidden points only” only interpolates those whose visibility is turned off.



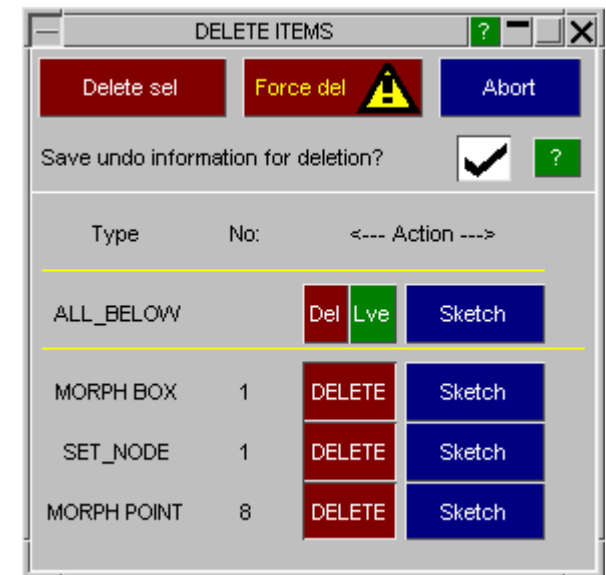
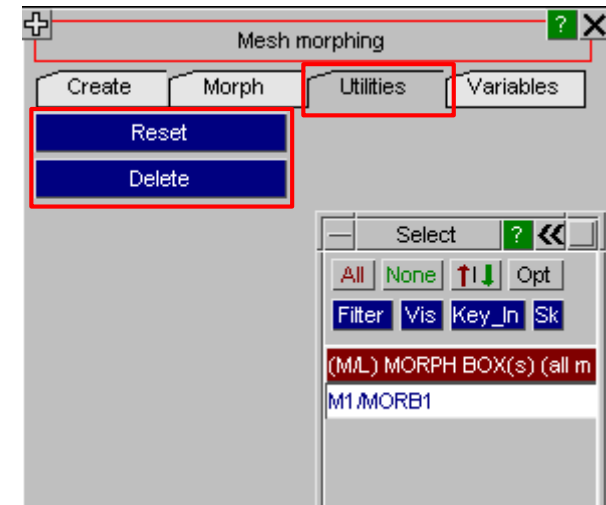
Offset Vectors

- During interactive dragging the offset vector of the selected morph points is now dynamically shown on the panel.
- You can also type in a vector, which will immediately move the points by this vector, which will be either in global or local coordinates specified by the radio buttons.
- This makes it easier to ensure symmetry during morphing.
- This replaces the offset text box in PRIMER 15, which was only available in Selection mode with drag direction X, Y or Z and where only one number was typed in.



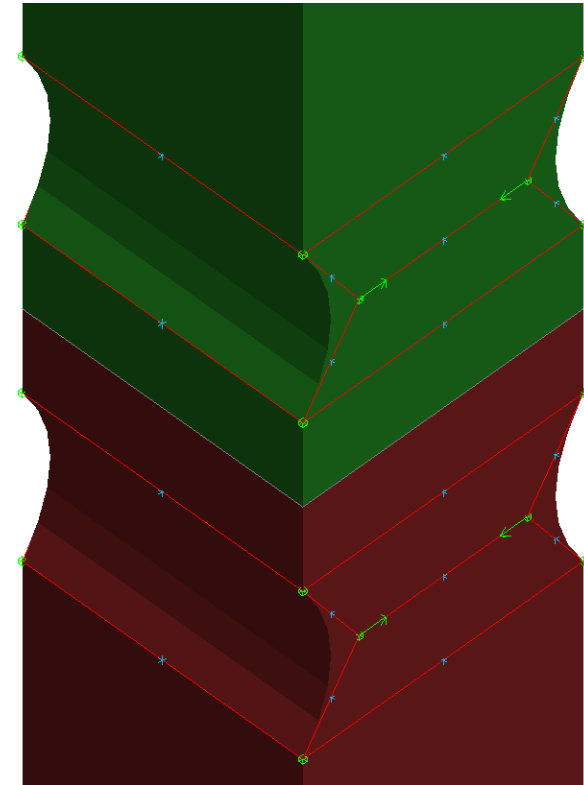
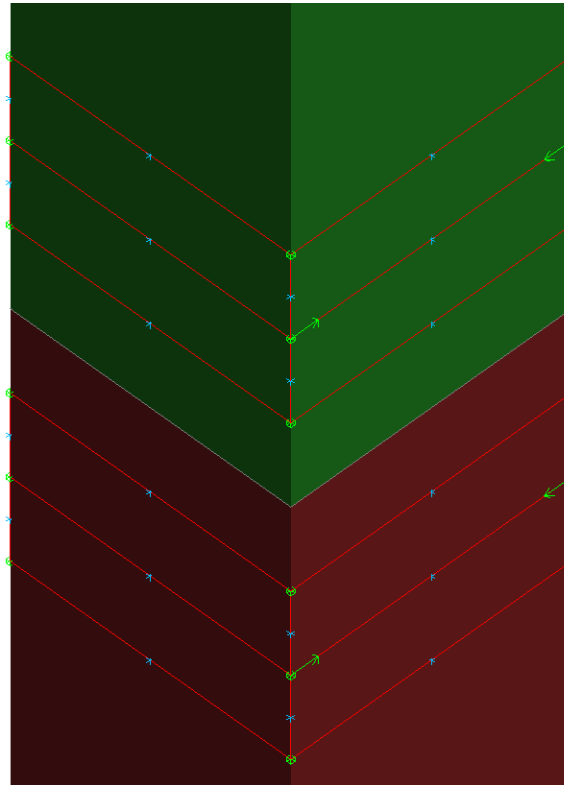
Deleting Morph Boxes

- The Reset tab in PRIMER 15 has now been replaced with a Utilities tab, which also allows deletion of morph boxes.



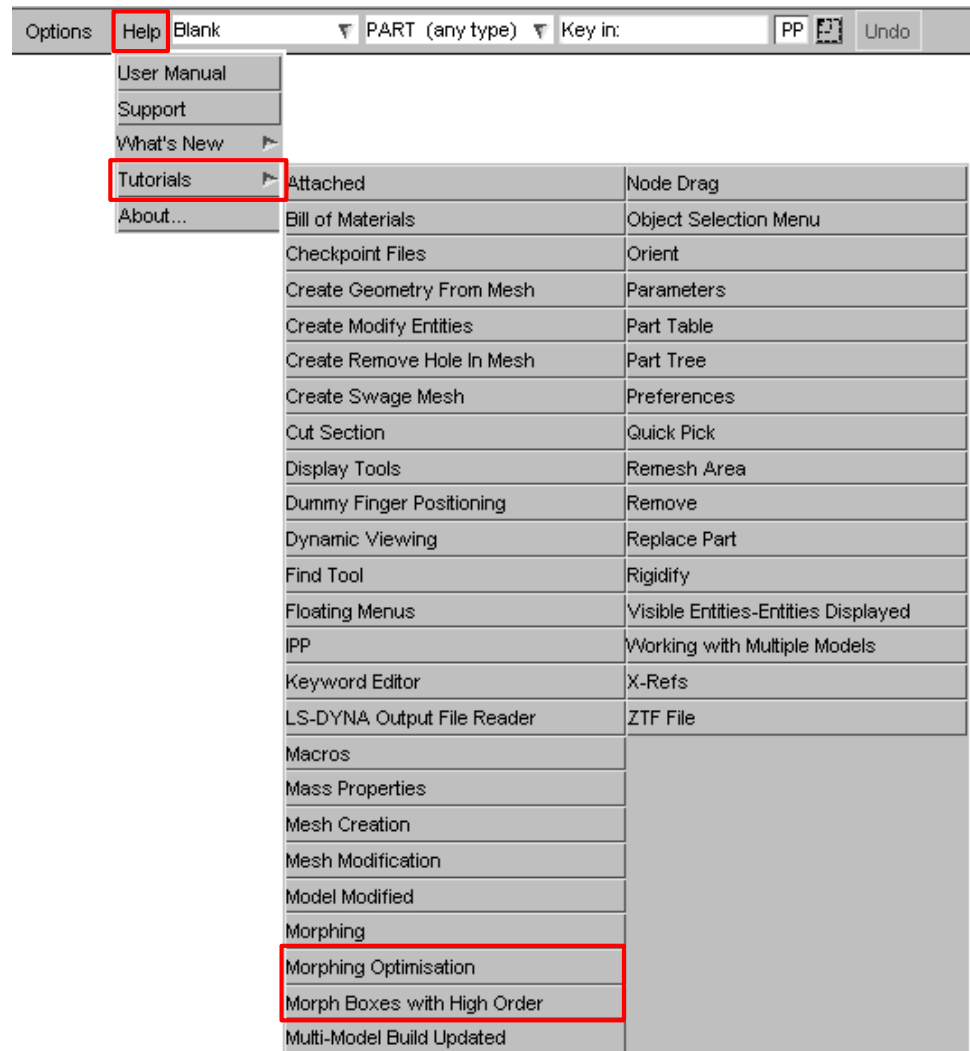
Morph Flows

- A morph flow is a list of morph points assigning a vector to each point.
- Each morph flow describes how morph points move when a variable is applied from the command line.
- Morph flows allow you to optimise objectives in LS-OPT by morphing in PRIMER.



Morphing Tutorials

- Tutorials are available on Help -> Tutorials.
- The **Morph Boxes with High Order** tutorial explains in more detail how smoothness can be controlled and what options are available.
- Details of the setup and the use for LS-OPT can be found in the **Morphing Optimisation** tutorial.

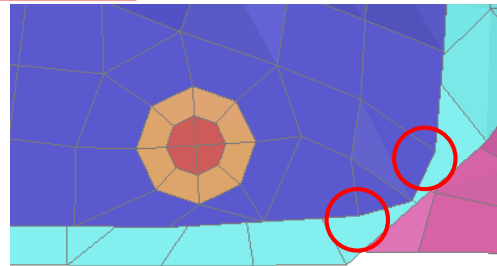
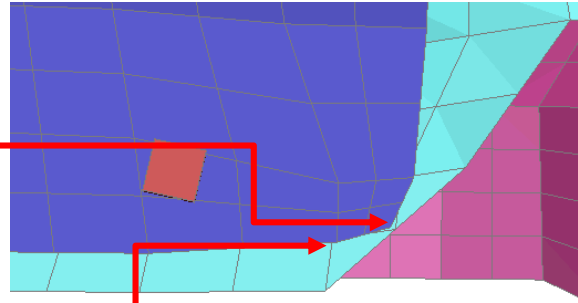


Meshing Improvements

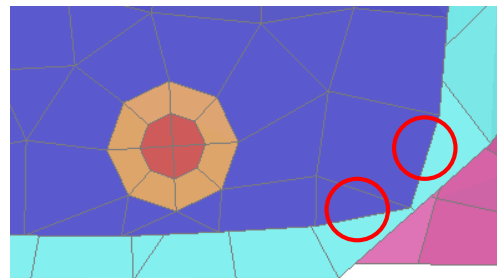
Meshing Improvements

Corners can be preserved during remeshing around spotwelds:

- Corners formed by an angle on one shell only are always preserved
- Corners formed by an angle at the junction of two (or more) shells are preserved if that angle is greater than a user specified value. This value can be saved as preference.



Corners preserved at (default) 10° setting

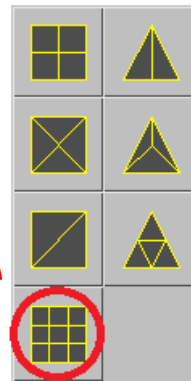
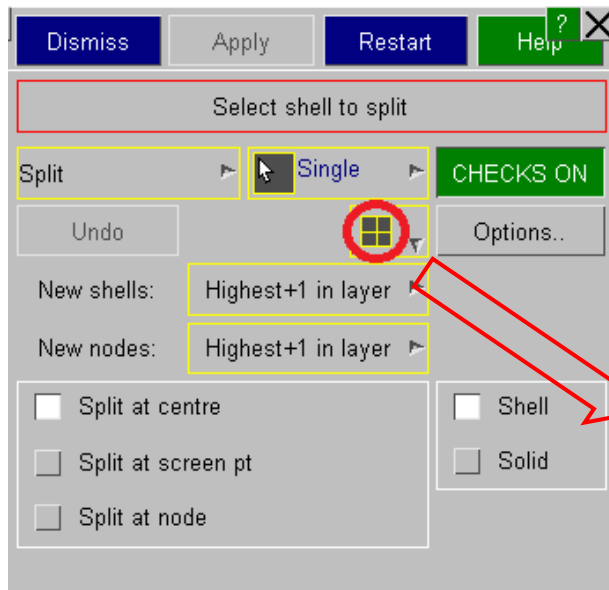


Corners not preserved at 20° setting

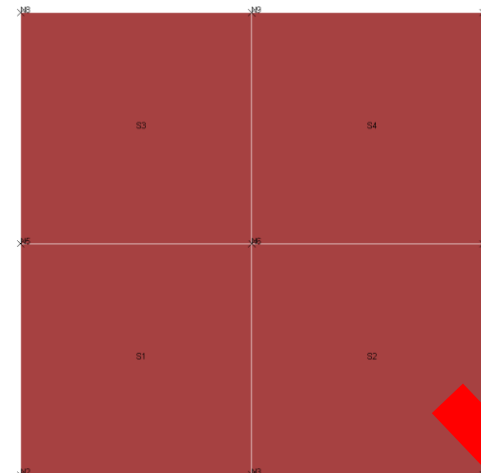
Shell Split Using 3x3 Split

Shell Split Using 3x3 Split

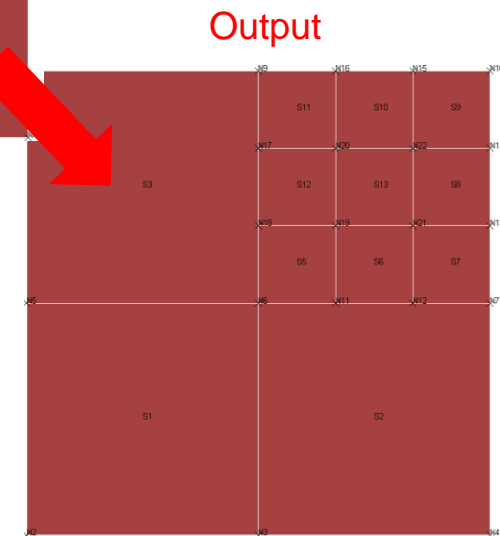
- A “3x3” option has been added to Shell split. Shell can be split using ‘Single’ option as well as ‘Multiple’ option.



Predefined split pattern popup



Input

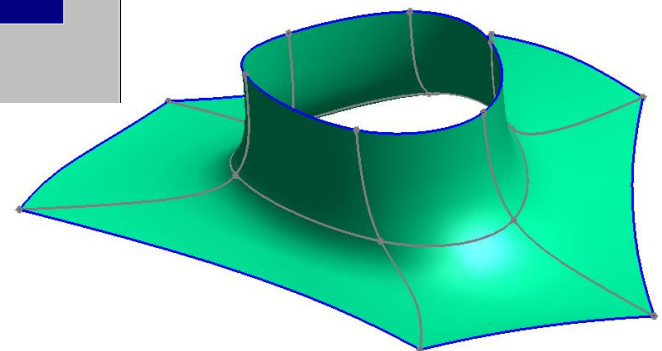
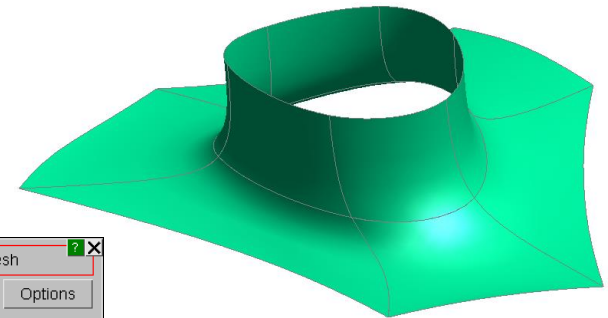
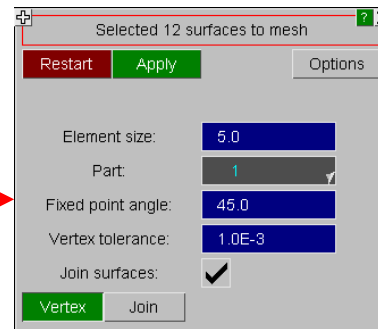
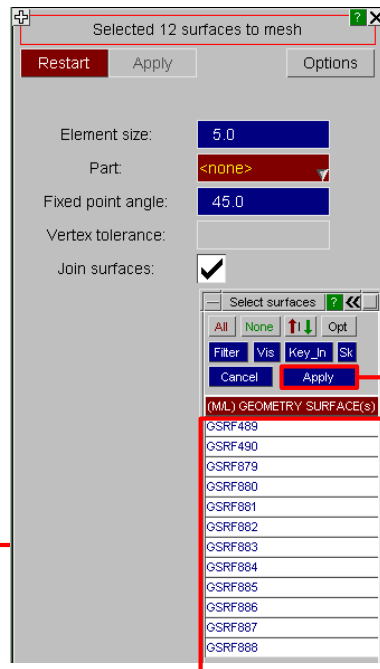
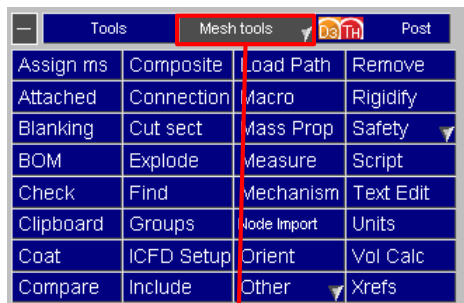


Output

Geometry Mesh

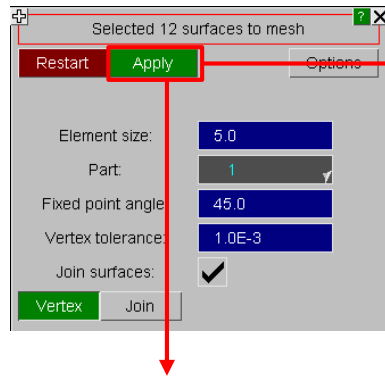
Geometry Mesh

- PRIMER v16.0 has ability to mesh the selected surface into a single meshed part. For example the image below shows 12 surfaces that we want to mesh. Each of the 12 surfaces will be meshed together rather than creating nodes along the shared surface edges.
- Select the surface(s) that you want to mesh in the object menu and press Apply in the object menu. PRIMER will then show the surfaces that will be meshed together with Grey line boundaries and fixed points with Grey circles.

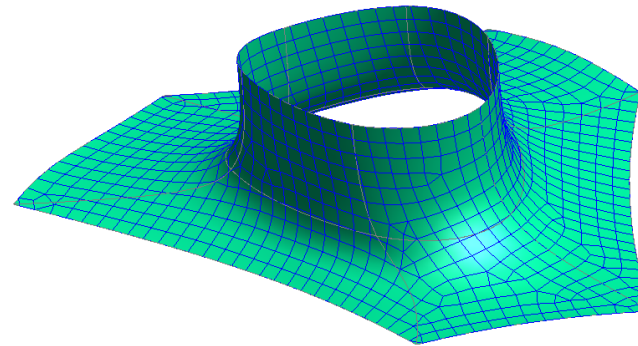


Geometry Mesh

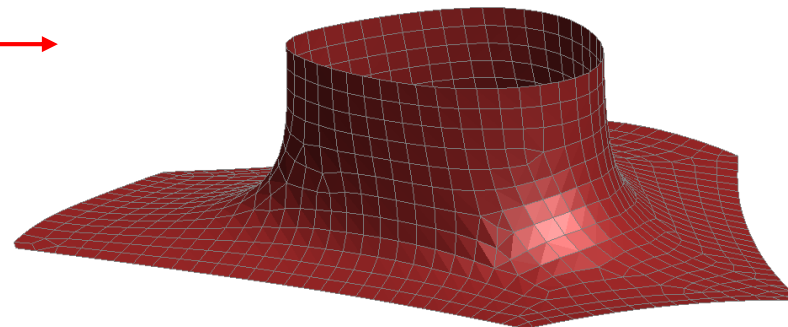
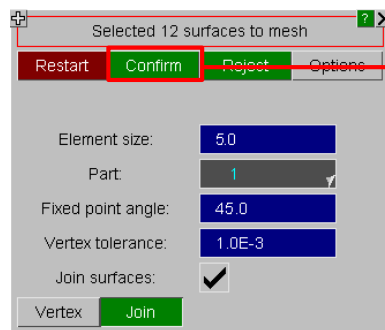
- Once the surfaces have been selected and a part ID is given the Apply button will become active.
- If you press it then PRIMER will show a preview of the mesh that will be generated.



Preview of mesh



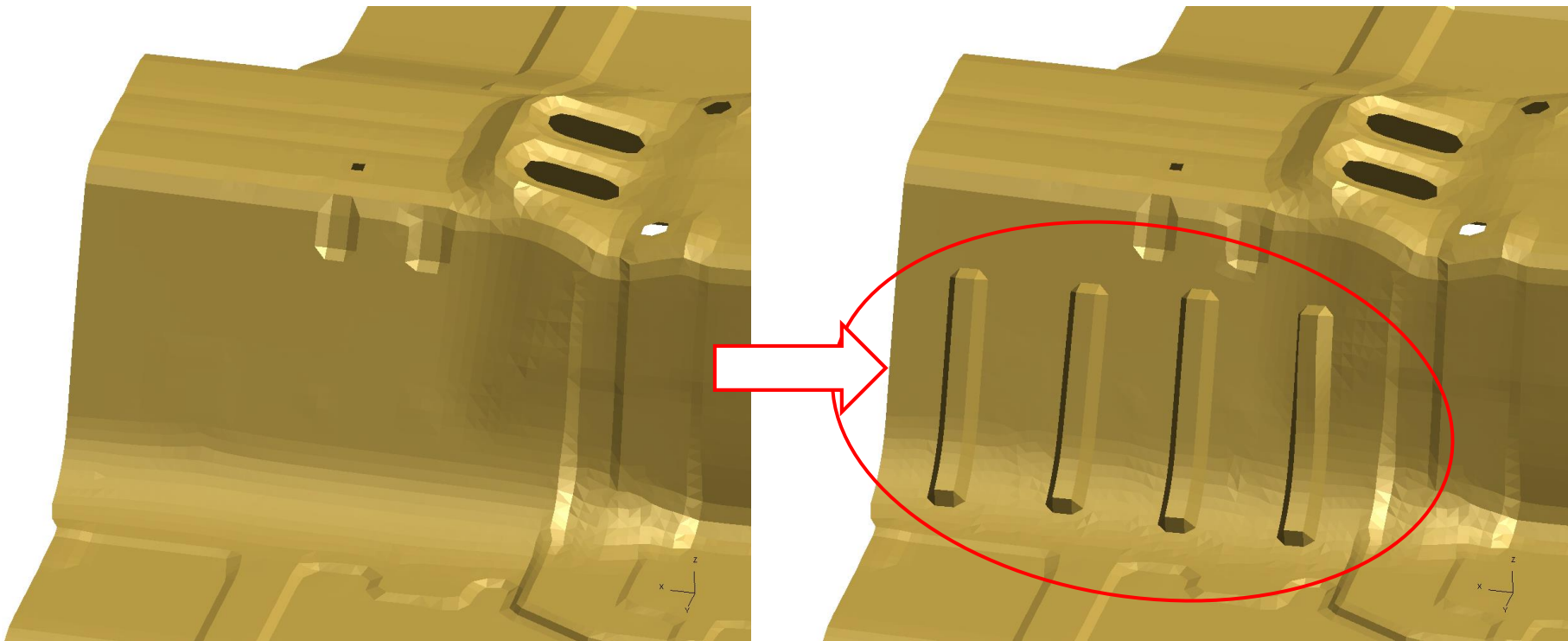
Meshed surface



Swages Updates

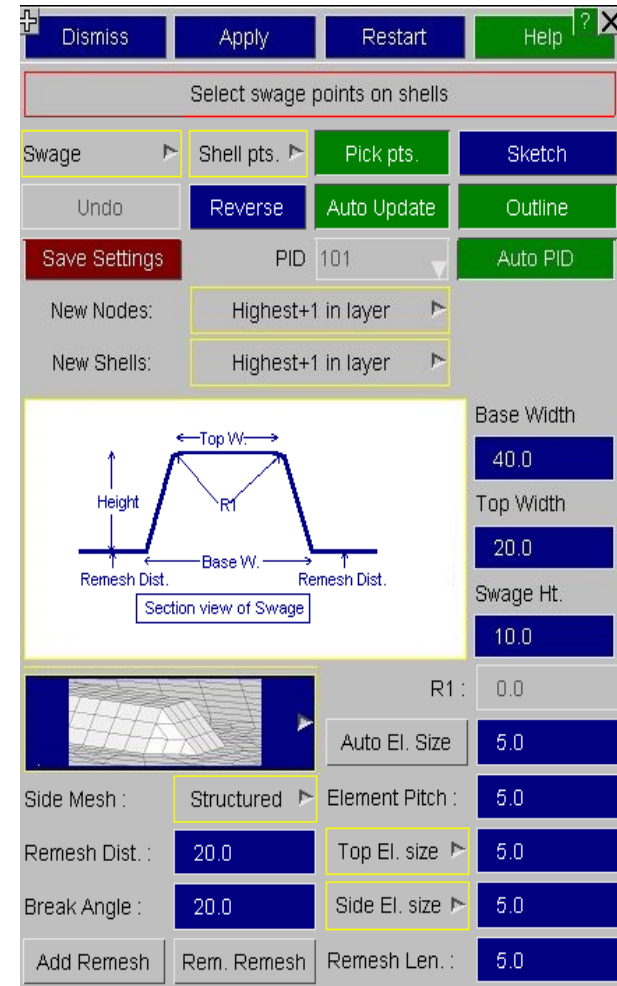
Mesh Tools -> Swages

- Mesh Tools -> “Swages” is used to create swages/beads in shell meshes.



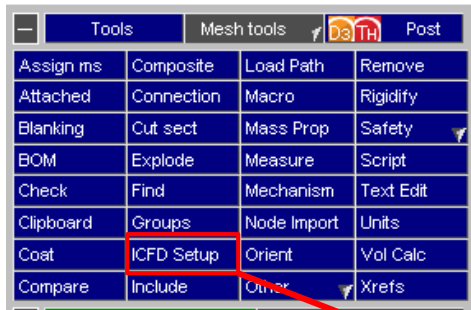
Mesh Tools -> Swages (cont.)

- To define the swage path, previously only points on shells could be used. Now the user can also use nodal points to define a path.
- A new 'Outline' button just shows the outline of the proposed swage sketch.
- The TOP surface of the swage can be created in a circular shape with a new input parameter : R1. This parameter can take values up to half the size of the TOP surface width, or half the size of the Swage Height, whichever is lower.



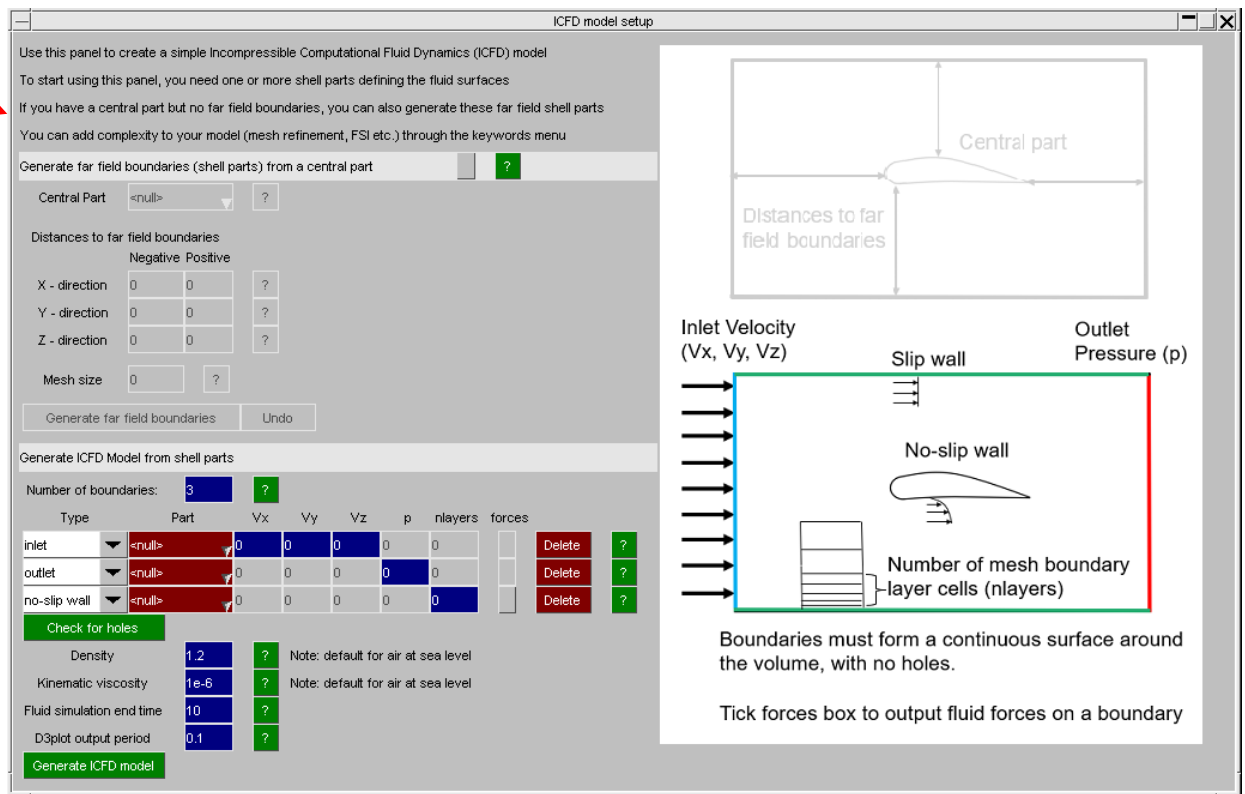
ICFD Setup

ICFD Model Setup Tool

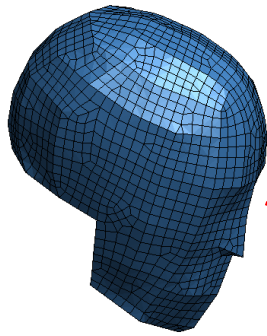


A new tool has been created to generate an ICFD model from shell parts.

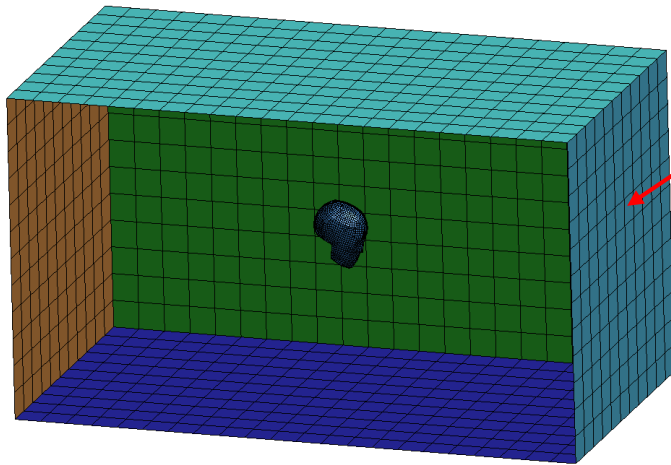
It includes an option to create far field fluid boundaries around a central specified part.



ICFD Model Setup Tool



Option to automatically create far field boundaries around central part.



Use this panel to create a simple Incompressible Computational Fluid Dynamics (ICFD) model
To start using this panel, you need one or more shell parts defining the fluid surfaces
If you have a central part but no far field boundaries, you can also generate these far field shell parts
You can add complexity to your model (mesh refinement, FSI etc.) through the keywords menu

Generate far field boundaries (shell parts) from a central part ☒ ?

Central Part 193 ?

Distances to far field boundaries

	Negative	Positive	
X - direction	1000	1000	?
Y - direction	500	500	?
Z - direction	500	500	?

Mesh size 100 ?

Generate far field boundaries Undo

Generate ICFD Model from shell parts

Number of boundaries: 3 ?

Type	Part	Vx	Vy	Vz	p	nlayers	forces	
Inlet	<null>	0	0	0	0	0		Delete ?
outlet	<null>	0	0	0	0	0		Delete ?
no-slip wall	<null>	0	0	0	0	0		Delete ?

Check for holes

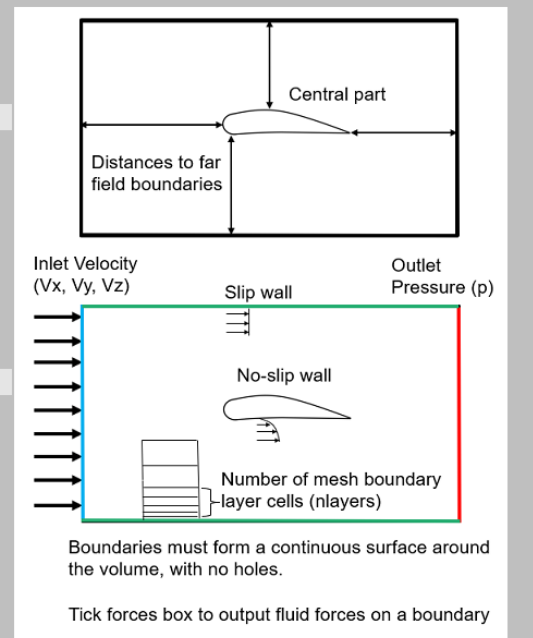
Density 1.2e-9 ? Note: default for air at sea level

Kinematic viscosity 1 ? Note: default for air at sea level

Fluid simulation end time 10 ?

D3plot output period 0.1 ?

Generate ICFD model



ICFD Model Setup Tool

ICFD model setup

Use this panel to create a simple Incompressible Computational Fluid Dynamics (ICFD) model
To start using this panel, you need one or more shell parts defining the fluid surfaces
If you have a central part but no far field boundaries, you can also generate these far field shell parts
You can add complexity to your model (mesh refinement, FSI etc.) through the keywords menu

Generate far field boundaries (shell parts) from a central part ☒ ?

Central Part 193 ?

Distances to far field boundaries

	Negative	Positive
X - direction	1000	1000
Y - direction	500	500
Z - direction	500	500

Mesh size 100 ?

Generate far field boundaries Undo

Generate ICFD Model from shell parts

Number of boundaries: 7 ?

Type	Part	Vx	Vy	Vz	p	nlayers	forces
inlet	195	5000	0	0	0	0	
outlet	194	0	0	0	0	0	
no-slip wall	196	0	0	0	0	5	
no-slip wall	197	0	0	0	0	5	
no-slip wall	198	0	0	0	0	5	
no-slip wall	199	0	0	0	0	5	
no-slip wall	193	0	0	0	0	5	<input checked="" type="checkbox"/>

Check for holes

Density 1.2e-9 ? Note: default for air at sea level

Kinematic viscosity 1.8e-8 ? Note: default for air at sea level

Fluid simulation end time 10 ?

D3plot output period 0.1 ?

Generate ICFD model

Diagram illustrating the ICFD model setup. It shows a central part (airfoil) and distances to far field boundaries. The diagram also shows the inlet velocity (Vx, Vy, Vz) and outlet pressure (p) conditions. The boundaries are labeled as Inlet Velocity (Vx, Vy, Vz), Slip wall, No-slip wall, and Number of mesh boundary layer cells (nlayers).

Boundaries must form a continuous surface around the volume, with no holes.

Tick forces box to output fluid forces on a boundary

Boundary Inputs

Keywords automatically created

ICFD Model generated

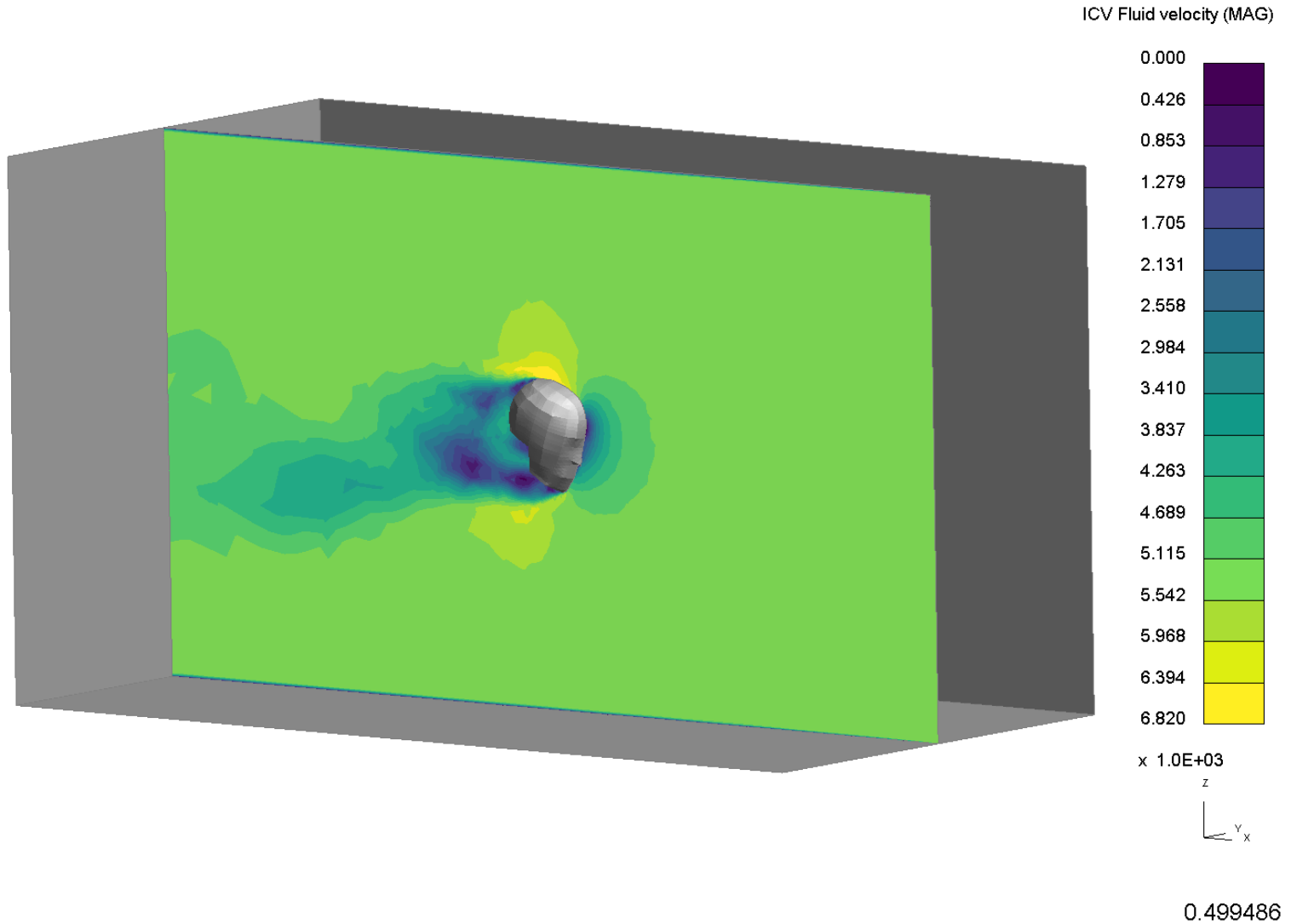
New ICFD Model created in M2. The following keywords have been created:

- *ICFD_PART
- *MESH_SURFACE_ELEMENT
- *MESH_NODE
- *ICFD_SECTION
- *ICFD_MAT
- *MESH_VOLUME
- *ICFD_PART_VOL
- *ICFD_CONTROL_TIME
- *DATABASE_BINARY_D3PLOT
- *ICFD_BOUNDARY_PRESCRIBED_VEL
- *ICFD_BOUNDARY_PRESCRIBED_VEL
- *ICFD_BOUNDARY_PRESCRIBED_PRE
- *ICFD_BOUNDARY_NONSLIP
- *MESH_BL
- *ICFD_BOUNDARY_NONSLIP
- *MESH_BL
- *ICFD_BOUNDARY_NONSLIP
- *MESH_BL
- *ICFD_BOUNDARY_NONSLIP
- *MESH_BL
- *ICFD_BOUNDARY_NONSLIP
- *MESH_BL
- *MESH_SYM

OK

ICFD Model Setup Tool

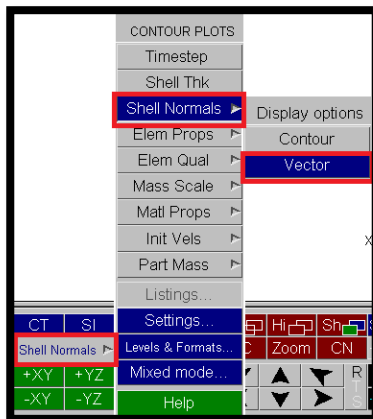
D3PLOT: ICFD Model



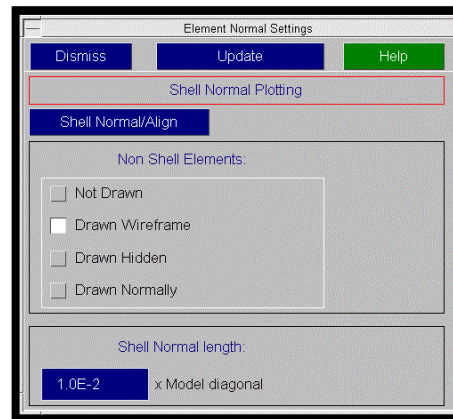
Displaying Shell Normals

Display Shell Normals as Arrows

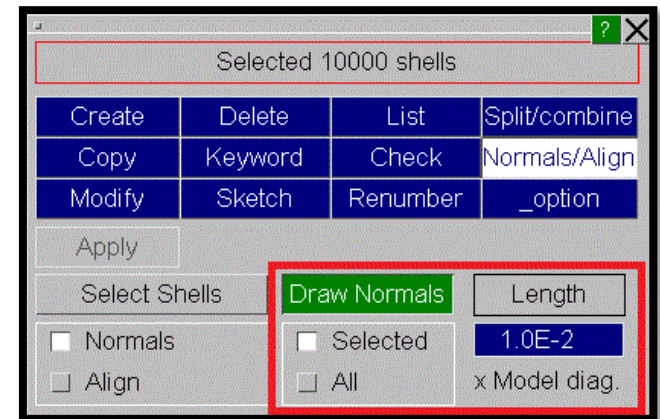
- A “Vector” sub option has been added in **View->Contour plots->Shell Normals** to display the direction of shell normals as arrows (rather than shell contouring).
- The length of the arrows can be changed in View->contour plots->Settings panel.
- The shell normal arrows can also be accessed from the Shell Elements panel: **Elements->Shell-> Normals/Align**
- The “Draw Normals” toggle button displays the shell normal direction as Arrows.
- The shell normals can be drawn for the “Selected” shells (via ‘Select Shells’) or for all the “All” shells in the model.
- The length of the shell normal arrows can be changed in the same panel.



View->Contour plots popup

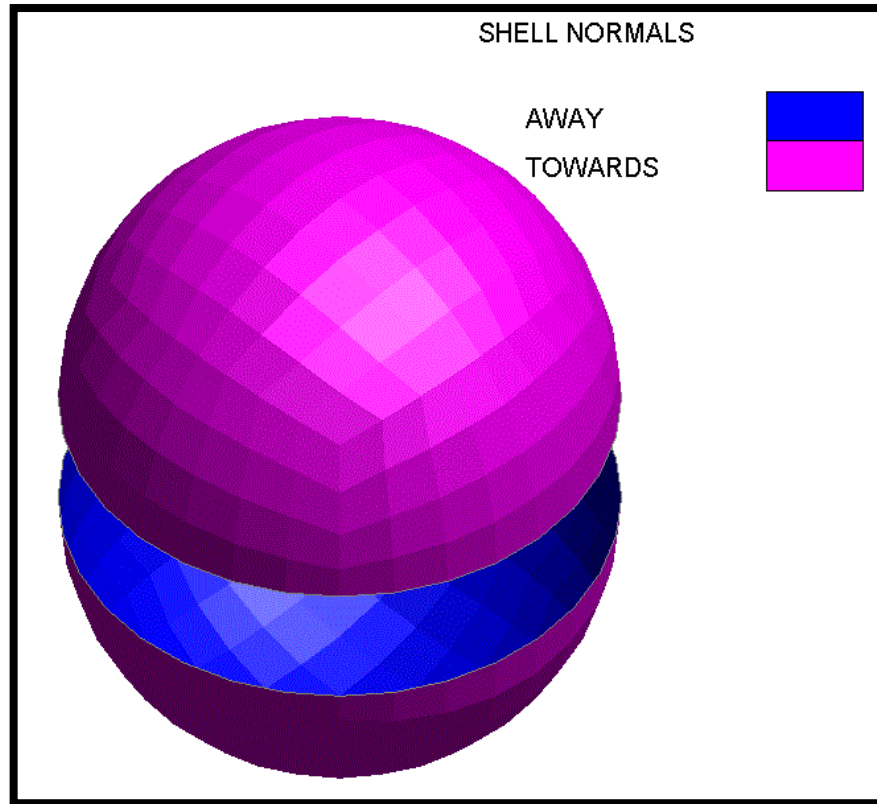


Shell Normals settings panel

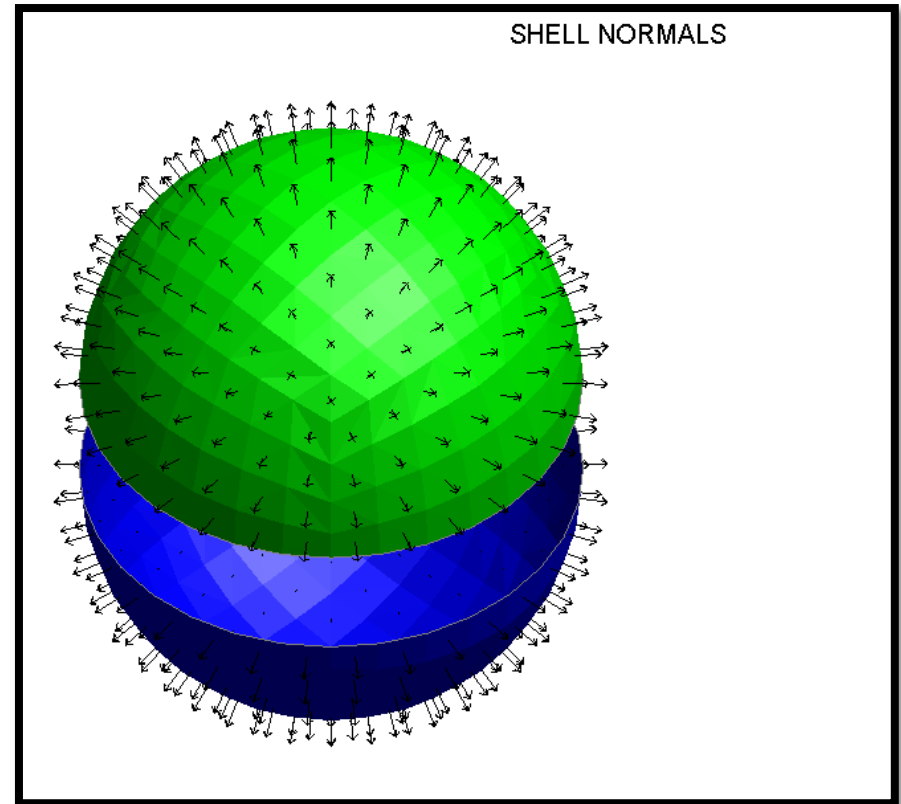


Elements-> Shell-> Normals/Align panel

Display Shell Normals as Arrows



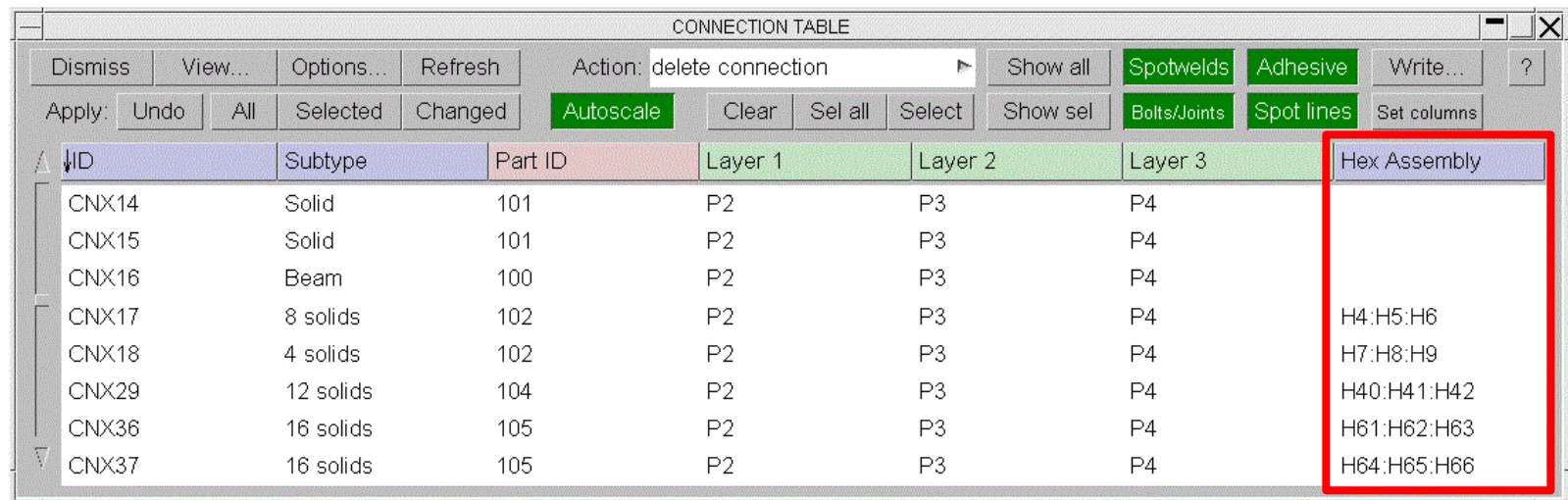
Shell Normals->Contour sub option



Shell Normals->Vector sub option

Hex Assembly Column in Connection Table

- A column “Hex Assembly” has been added to the connection table.
- The column shows the labels of Hex assemblies (*DEFINE_HEX_SPOTWELD_ASSEMBLY) of solid elements associated with connection entities in the model.



CONNECTION TABLE

Dismiss View... Options... Refresh Action: delete connection Show all Spotwelds Adhesive Write... ?

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

ID	Subtype	Part ID	Layer 1	Layer 2	Layer 3	Hex Assembly
CNX14	Solid	101	P2	P3	P4	
CNX15	Solid	101	P2	P3	P4	
CNX16	Beam	100	P2	P3	P4	
CNX17	8 solids	102	P2	P3	P4	H4:H5:H6
CNX18	4 solids	102	P2	P3	P4	H7:H8:H9
CNX29	12 solids	104	P2	P3	P4	H40:H41:H42
CNX36	16 solids	105	P2	P3	P4	H61:H62:H63
CNX37	16 solids	105	P2	P3	P4	H64:H65:H66

Connection table along with Hex Assembly column

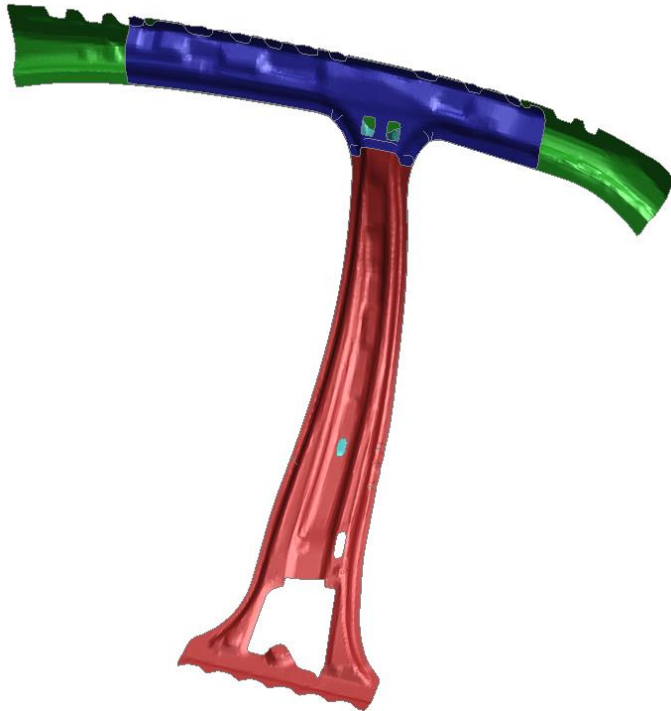
Iso Geometric Analyses

Iso Geometric Analyses

Iso Geometric Analyses can be run in LS-DYNA using the keywords

*ELEMENT_SHELL_NURBS_PATCH and *DEFINE_NURBS_CURVE to define the surfaces.

In v16 PRIMER can now display those surfaces.



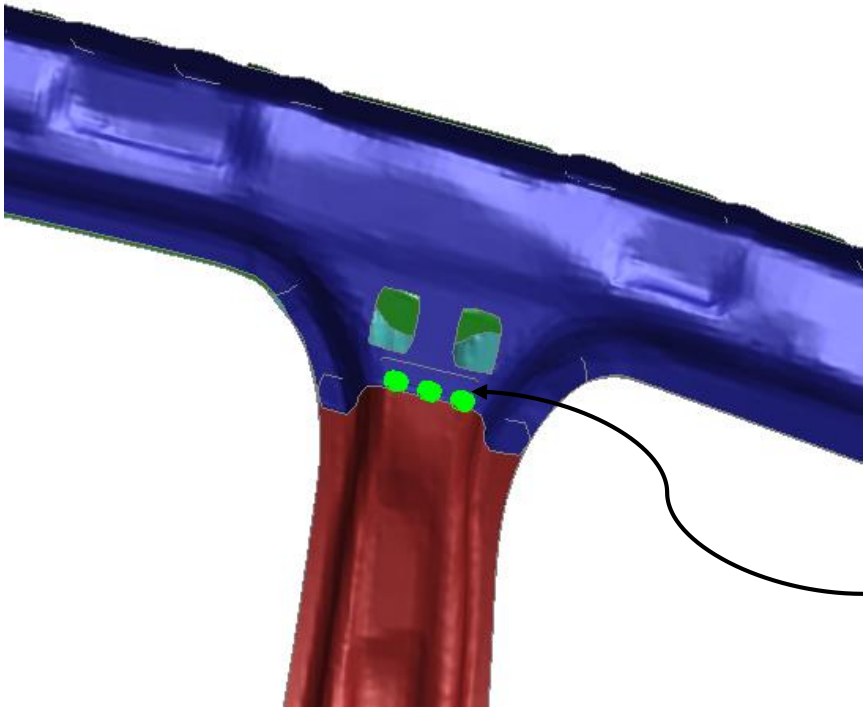
*ELEMENT_SHELL_NURBS_PATCH_TRIMMED								
\$#	npeid	pid	npr	pr	nps	ps	perir	peris
	1	1	39	3	40	3	1	2
\$#	wf1	form	int	nistr	niss	inass		
	0	0	0	3	3	0		
\$#	rk1	rk2	rk3	rk4	rk5	rk6	rk7	rk8
0.000000	0.000000	0.000000	0.000000	0.055556	0.086255	0.111111	0.123660	
0.135806	0.166667	0.182984	0.196627	0.222222	0.250185	0.267219	0.277778	
0.294823	0.304425	0.317454	0.333333	0.360275	0.388889	0.416701	0.444444	
0.476282	0.500000	0.513410	0.528246	0.544263	0.555556	0.611111	0.622222	
0.666667	0.686017	0.722222	0.777778	0.833333	0.888889	0.944444	1.000000	
1.000000	1.000000	1.000000	0.000000	0.000000	0.000000	0.000000	0.000000	
*DEFINE_NURBS_CURVE								
\$#	sk1	sk2	1	99	3	0	1	0
0.000000	0.000000	0.0	0.000000	0.000000	0.000000	0.000000	0.039873	0.049960
0.166667	0.180731	0.1	0.100059	0.120103	0.160312	0.180391	0.200372	0.210351
0.277778	0.283358	0.2	0.225270	0.230212	0.240102	0.250013	0.254973	0.259933
0.388889	0.444444	0.4	0.269731	0.274598	0.279413	0.298809	0.308538	0.313372
0.630935	0.666667	0.7	0.322808	0.327466	0.336747	0.345990	0.355226	0.359806
1.000000	1.000000	1.0	0.381978	0.390392	0.406645	0.422375	0.437803	0.453405
			0.529257	0.541556	0.554140	0.568236	0.575869	0.579691
			0.587291	0.591136	0.591136	0.591136	0.595085	0.597066
			0.606728	0.614307	0.629653	0.645624	0.661572	0.677034
			0.754367	0.785472	0.816134	0.845939	0.860336	0.874423
			0.910622	0.918635	0.926312	0.933059	0.938921	0.944256
			0.970490	0.978306	0.982138	0.985939	0.989728	0.991625
			0.994375	0.995181	0.996791	1.000000	1.000000	1.000000
			0.713665	0.999886	0.000000	1.000000		
			0.693618	0.999420	0.000000	1.000000		
			0.668504	0.998561	0.000000	1.000000		
			0.638315	0.998446	0.000000	1.000000		
			0.618123	0.997535	0.000000	1.000000		
			0.592968	0.996121	0.000000	1.000000		
			0.562790	0.994548	0.000000	1.000000		
			0.522526	0.991710	0.000000	1.000000		
			0.482261	0.988379	0.000000	1.000000		
			0.442062	0.984649	0.000000	1.000000		
			0.417029	0.982088	0.000000	1.000000		

Iso Geometric Analyses

The connection tool has been updated so it can make connections between

- *ELEMENT_SHELL_NURBS_PATCH elements and between

- *ELEMENT_SHELL_NURBS_PATCH elements and normal *ELEMENT_SHELL elements

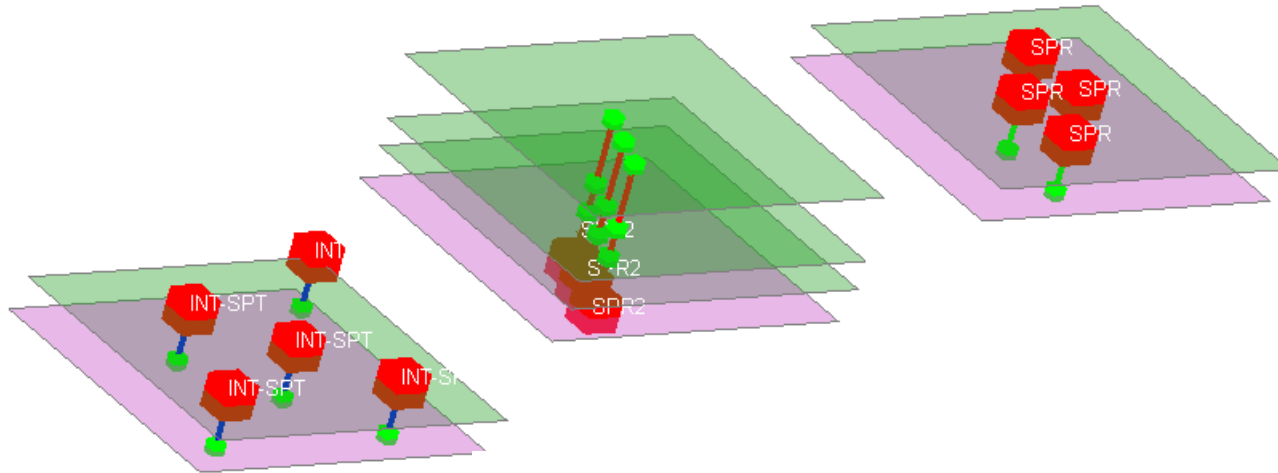


Connections between
*ELEMENT_SHELL_NURBS_PATCH elements

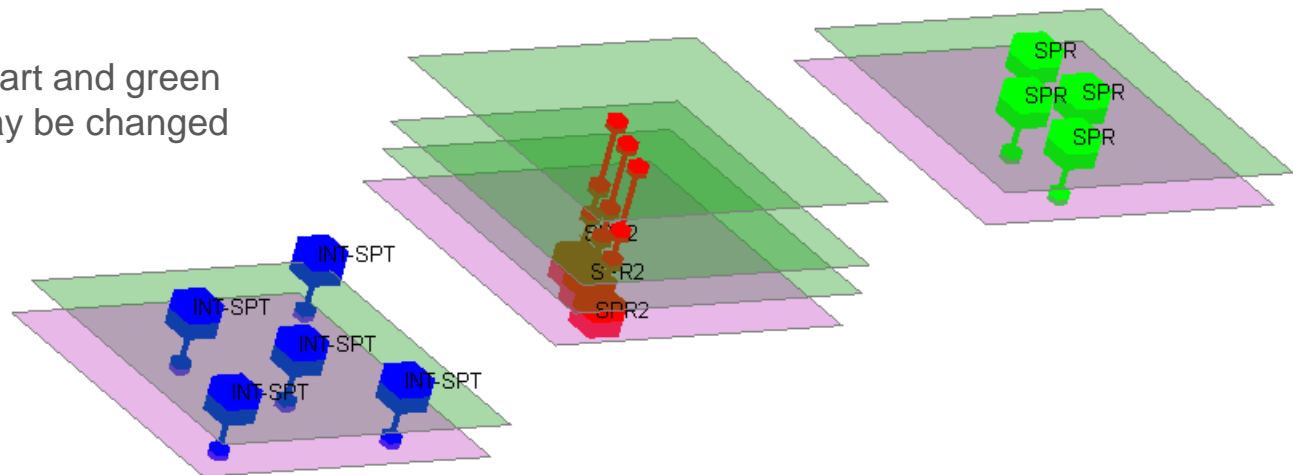
***CONSTRAINED**

*CONSTRAINED_INTERPOLATION_SPOTWELD C_SPR C_SPR2

These are now drawn using projection to the specified sheets



Default is red head on master part and green tail. On Display > colour this may be changed to colour by node set



Constrained SPR/SPR2/SPR3

☐ Red head, Green tail

☐ By node set

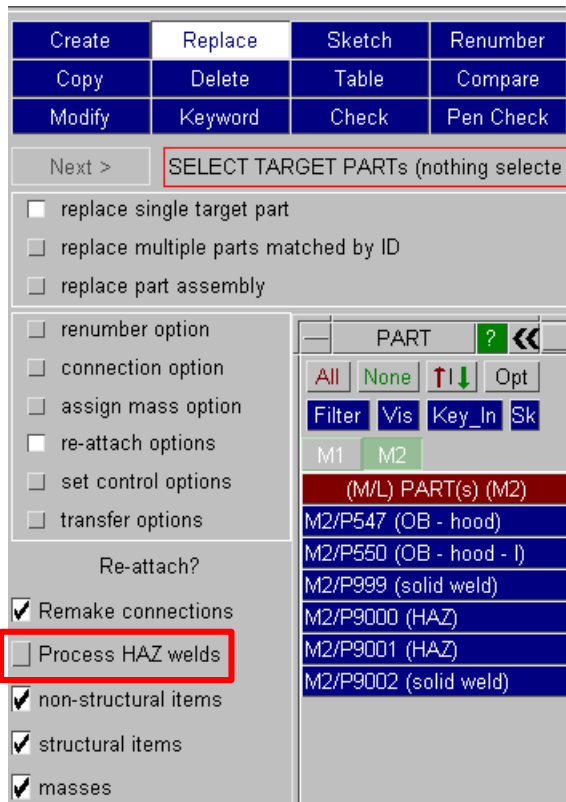
Connections

Part Replace and Processing of HAZ Weld Connections

Part replace gives control over processing of HAZ weld connections.

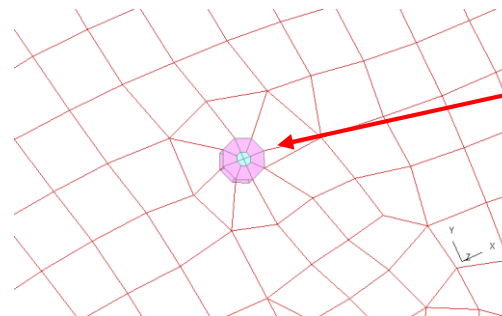
ON – unmake before deletion and remake on completion if ‘Remake connections’ is on (note - this option will remesh all parts connected by the weld, not just the replaced target part).

OFF – do not unmake HAZ welds and do not remake (irrespective of ‘Remake connections’).



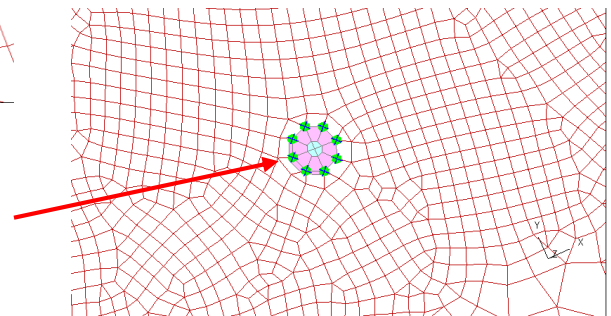
If HAZ rings are different to target part ,source part nodes configured to merge onto HAZ rings.

OFF option will enable part-replace without remesh on other layers.



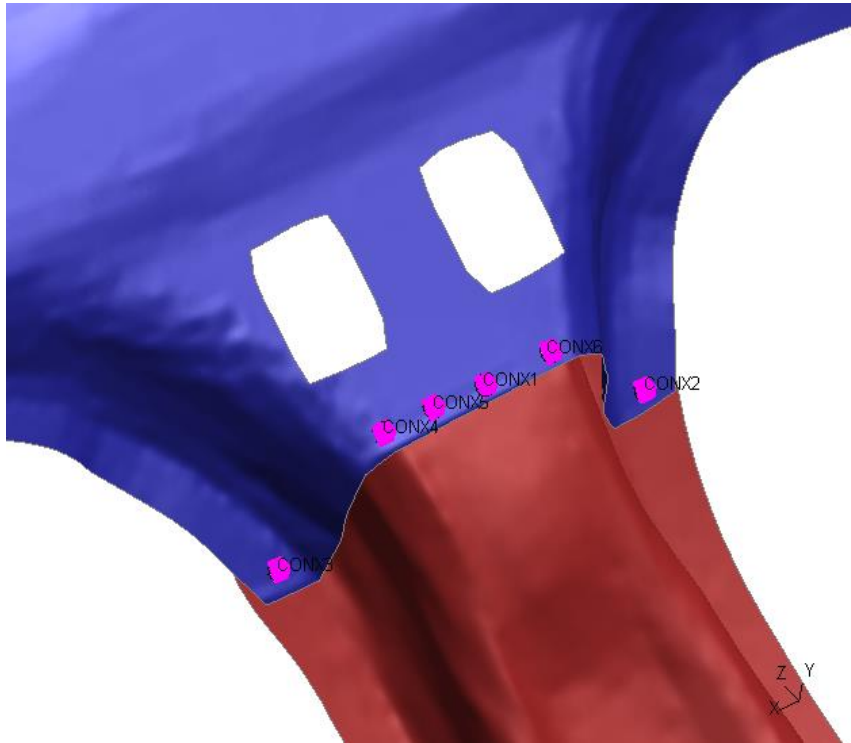
target part with HAZ weld

Source part imported and ready to merge onto HAZ weld ring. Mesh change on one part only.

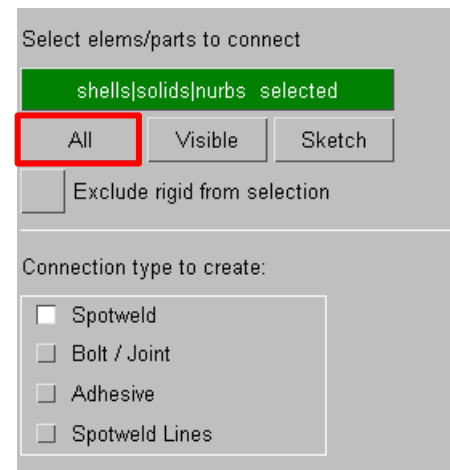


Welding to SOLIDS, TSHELLS & SHELL_NURBS_PATCH

PRIMER spot-welder will now form segments on external faces of shell/solid/tshell parts and on shell_nurbs_patch enabling welds (or adhesive) to be made to them.



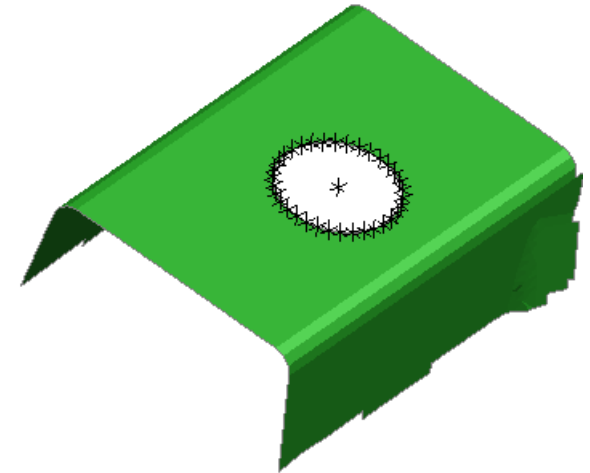
Selection of all elements for weld creation in spotweld mode now considers shells, solids and shell_nurb_patch under “All”.



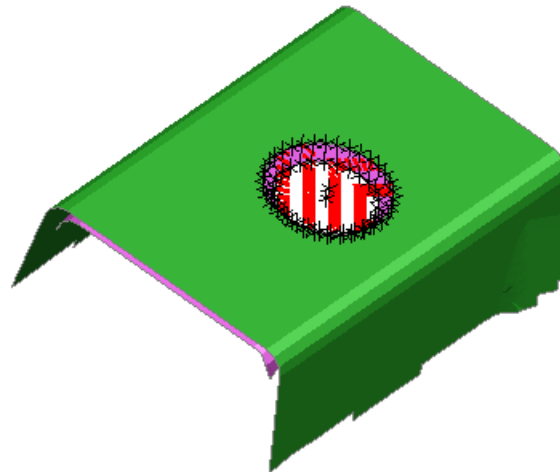
NRB Bolt Connections Across Includes use SET_COLLECT

If NRB bolt connection spans include files, PRIMER will use SET_COLLECT to avoid SET_LIST with latent nodes which can get lost e.g. accidental cleanup on stand alone include.

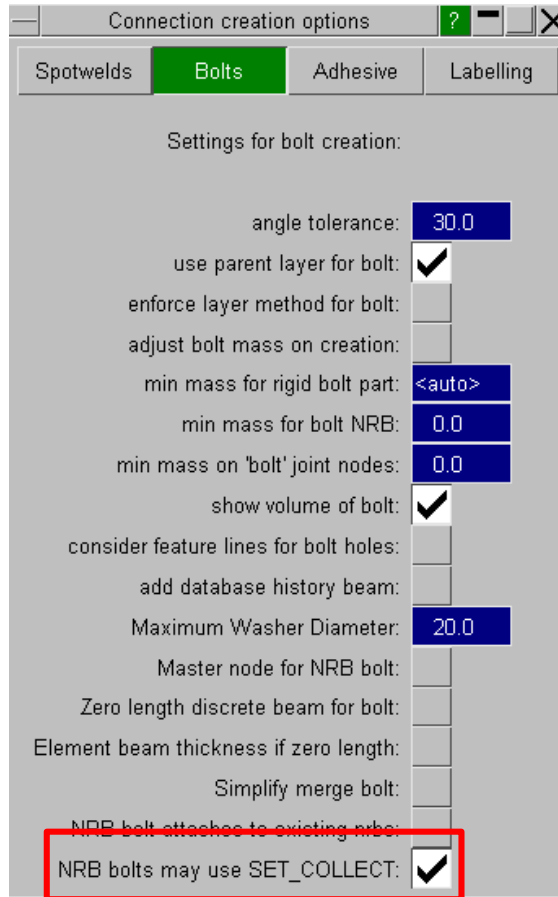
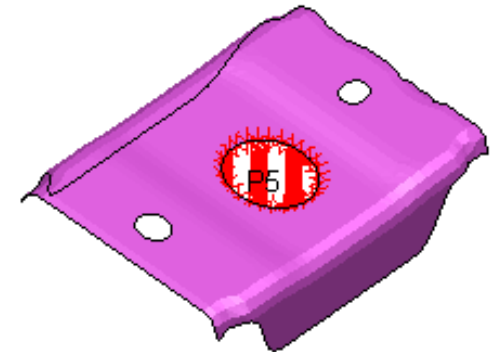
Include 1 read stand alone has nset



Master file shows NRB from complete node set



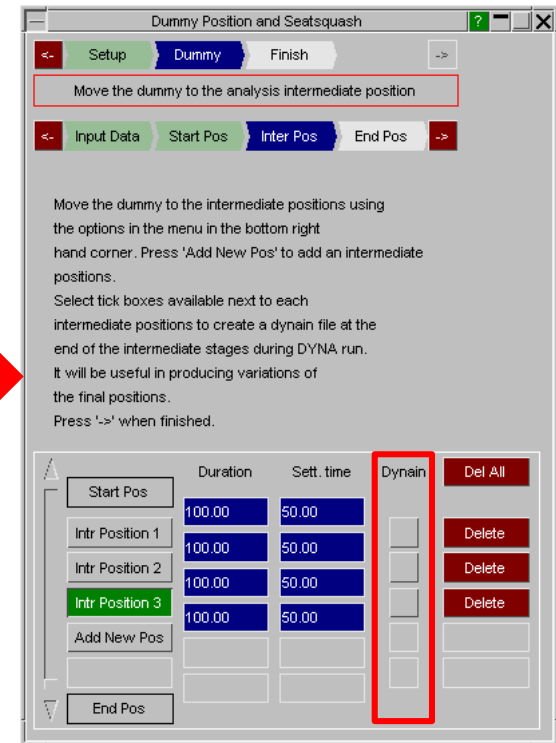
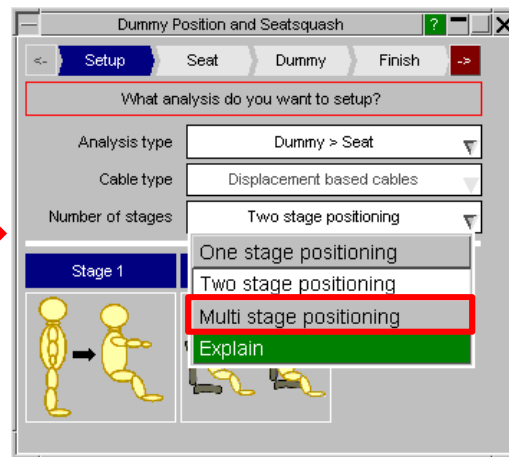
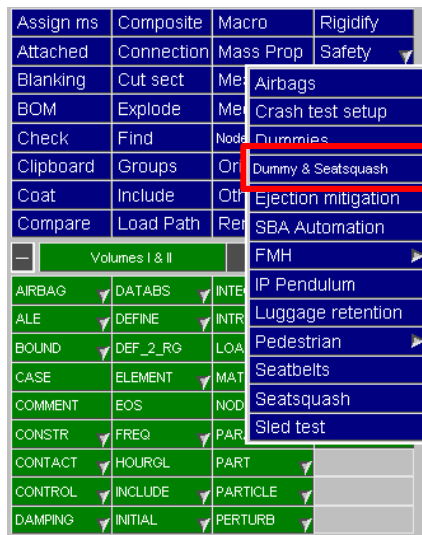
Include 2 read stand alone has nrb



Multi Stage Positioning

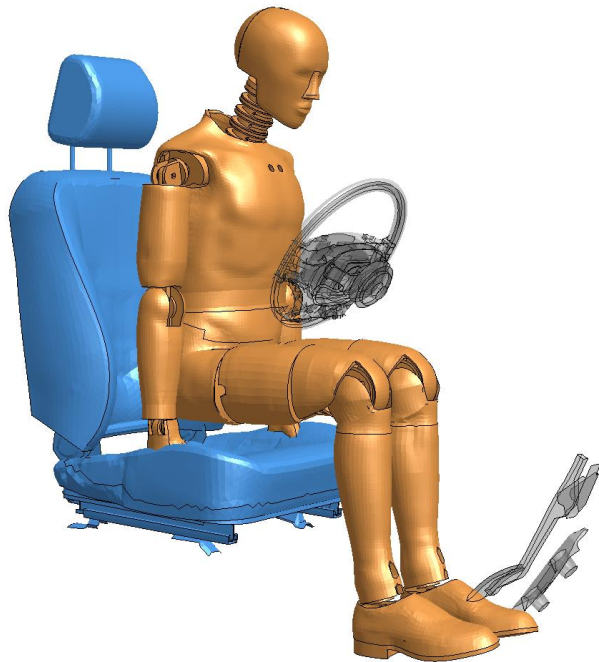
Multi Stage Positioning

- A new option “Multi stage positioning” is available within the “Dummy position and Seatsquash” panel.
- This option allows you to define multiple intermediate stages for a dummy position and seatsquash simulation.
- Select the tick boxes available next to each positions to write out a dynain file at the end of the intermediate stages during the LS-DYNA run. This could be helpful in producing variations of the final position.

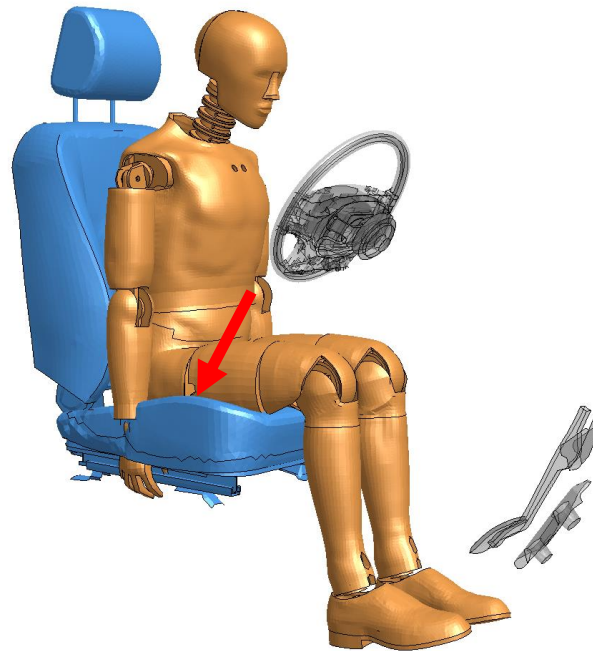


Multi Stage Positioning

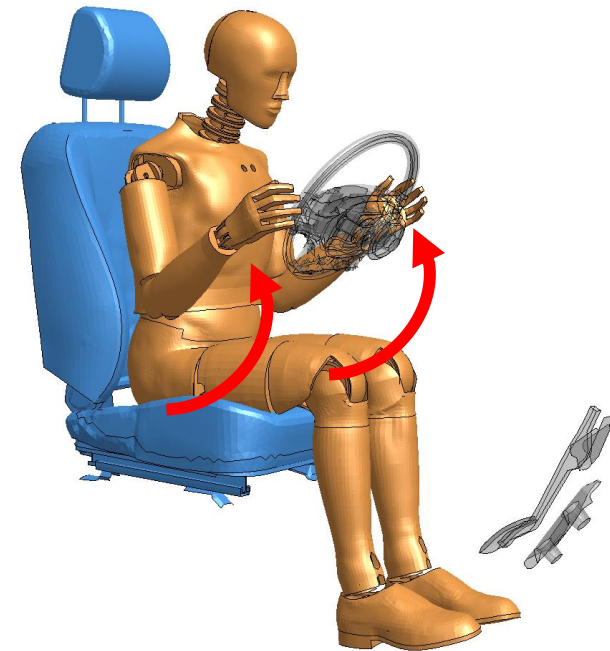
- Example of multi-stage positioning – all in one analysis:



Stage 1 – Occupant starts above seat



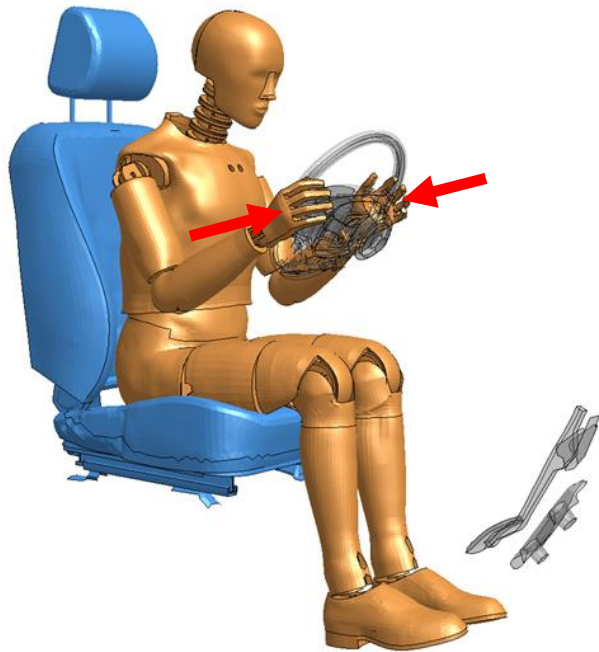
Stage 2 – Occupant is moved into seat (seat-squash)



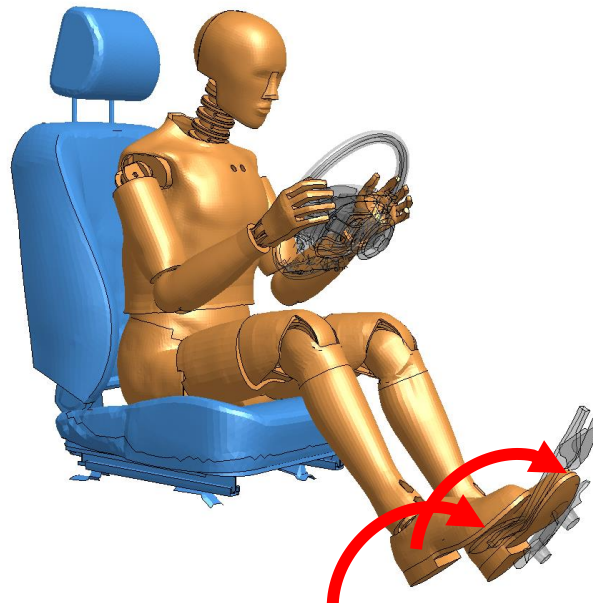
Stage3 – Arms are moved to either side of the steering wheel

Multi Stage Positioning

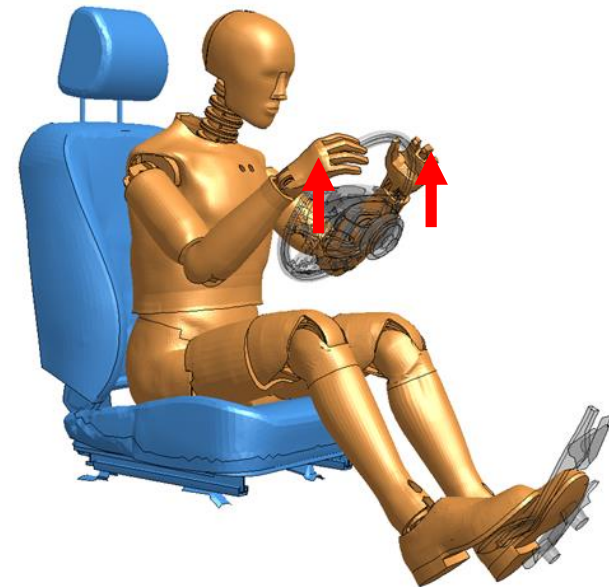
- Example of multi-stage positioning – all in one analysis:



Stage 4 – Hands moved onto steering wheel



Stage 5 – Feet moved onto pedals – **dynain file produced**



Stage 6 – Hands moved to different position on steering wheel – **dynain file produced**

Seatsquash

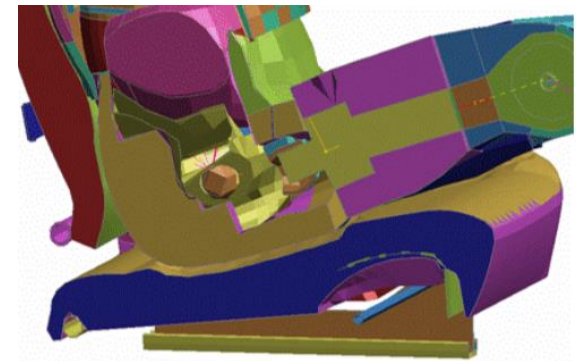
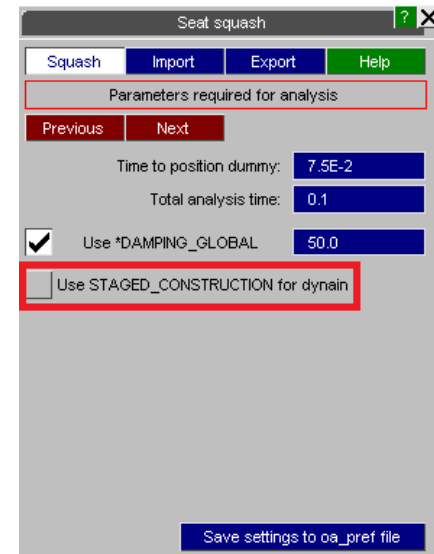
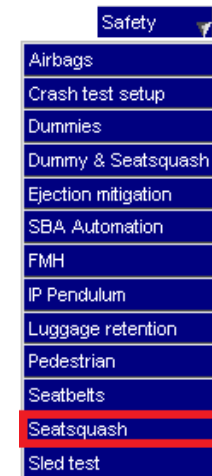
Alternative Method for Outputting dynain File

An LS-DYNA seatsquash analysis set up in PRIMER will, by default, output a dynain using the *INTERFACE_SPRINGBACK keyword.

In LS-DYNA R9, any parts that use *MAT_FABRIC using this method are not output to the dynain file, so an alternative method of outputting the dynain file has been added.

This uses STAGED_CONSTRUCTION cards and can be selected interactively on the seatsquash menu and set with the preference.

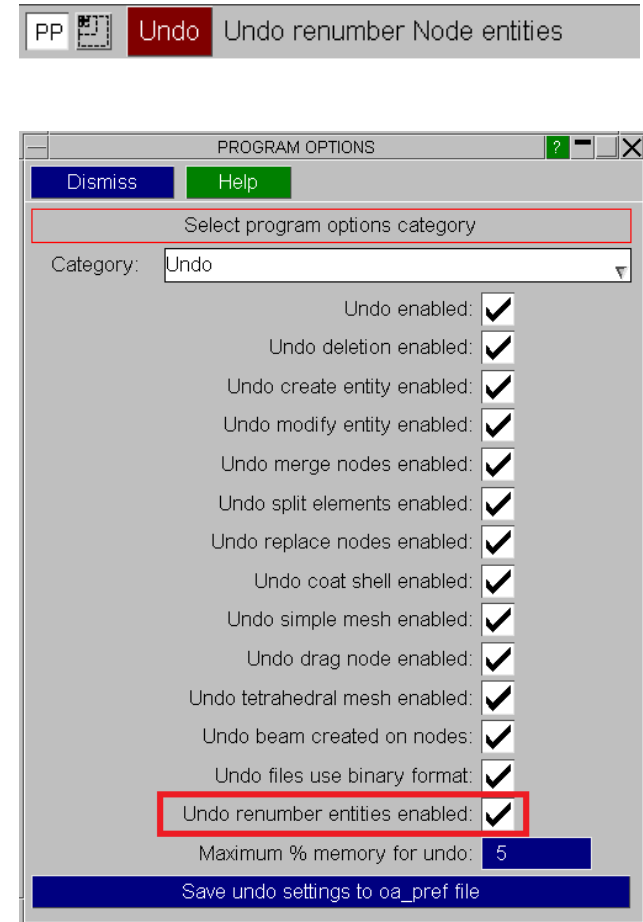
primer*squash_use_define_construction: TRUE



Undo for Renumbering

Undo for Renumbering

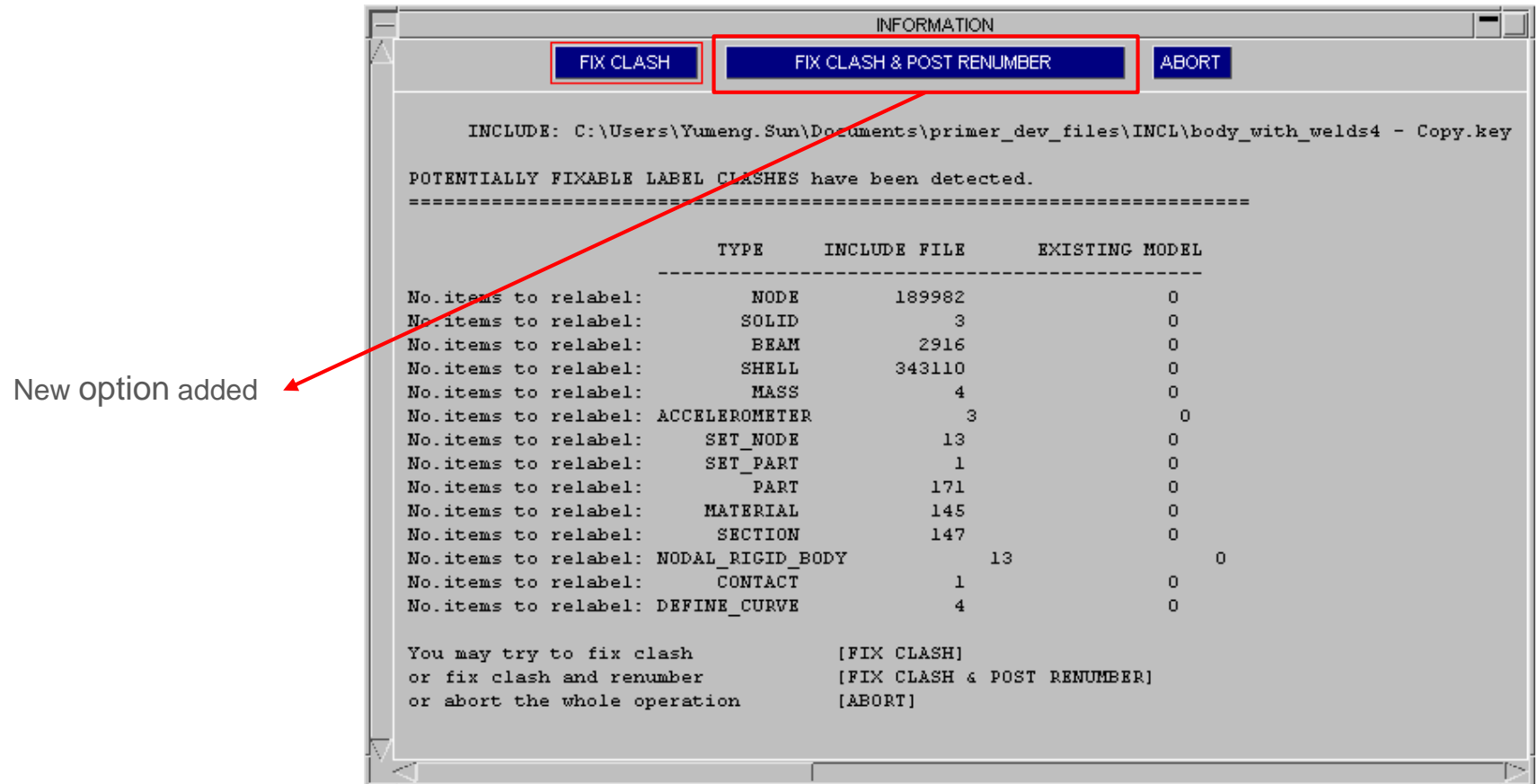
- The Undo option is now available for renumbering entities within the various renumbering panels.
 - In Model tab:
 - Renumber selection.
 - Renumber contents.
 - Renumber includes.
 - Visualize panel.
 - Renumber option in clipboard.
 - Renumber option in individual entity edit panels.
- The feature is accessible via generic UNDO button on the top menu bar in PRIMER
- A setting in the Undo options panel, and an equivalent preference can be used to turn this feature off/on (on by default).



Adding/Replacing an INCLUDE file

New [FIX CLASH & RENUMBER] Option

- When adding or replaying an include file, if label clashes are detected, users now have the option to manually renumber the corrected include file.



New [FIX CLASH & RENUMBER] option

RENUMBER SELECTED

Dismiss Apply Sketch Options

Renumber entities Post-renumber label declash - OFF

Help

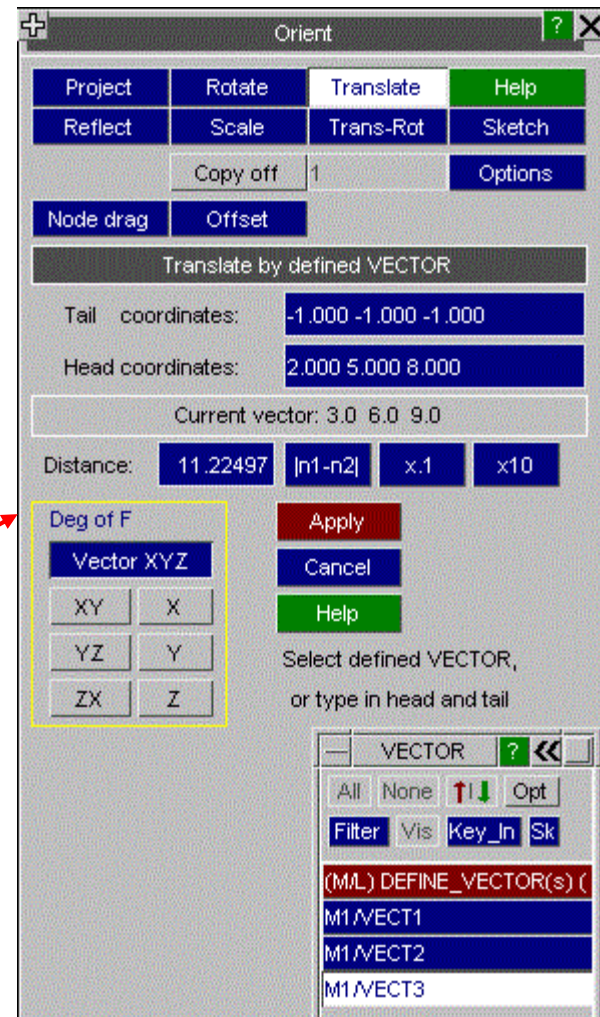
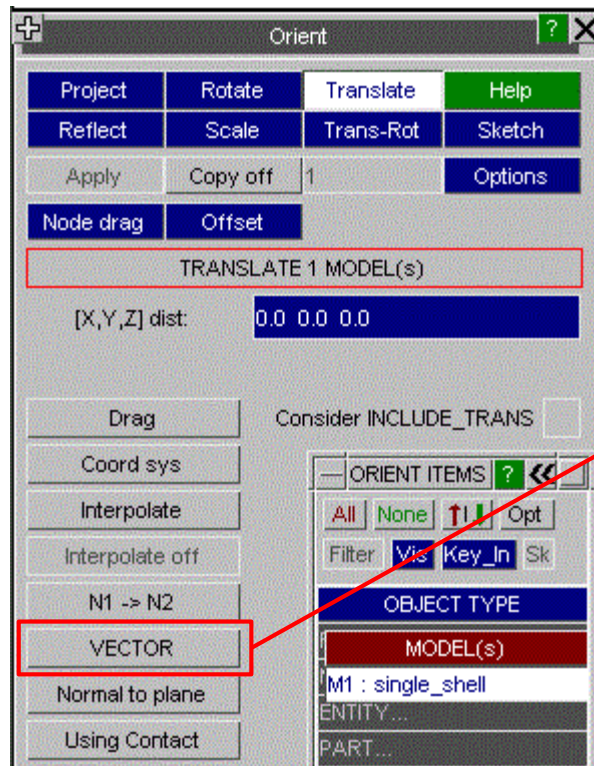
Type	Renumber		low:high	start at	offset	Information	Action
ALL TYPES	YES	NO	id low:high (num sel)	1		LIST STATUS	GLOBAL ACTION
MASS	YES	NO	460093:460096 (4/184524)	1		CLASH	Do not renumber
ACCELEROMETER	YES	NO	525122:525124 (3/10)	1		NO CLASH	No action needed
SET_NODE	YES	NO	1001457:1001469 (13/134)	1		NO CLASH	No action needed
SET_PART	YES	NO	1001434:1001434 (1/8)	1		NO CLASH	No action needed
PART	YES	NO	10001:10171 (171/546)	1		CLASH	Do not renumber
MATERIAL	YES	NO	10000:10144 (145/466)	1		CLASH	Do not renumber
SECTION	YES	NO	9999:10145 (147/471)	1		CLASH	Do not renumber
NODAL_RIGID_BODY	YES	NO	1001431:1001443 (13/116)	1		NO CLASH	No action needed
CONTACT	YES	NO	10000:10000 (1/4)	1		CLASH	Do not renumber
DEFINE_CURVE	YES	NO	320058:320061 (4/23)	1		CLASH	Do not renumber

The RENUMBER SELECTED window opens after the automatic clash fix

New Orient Translate Option

New [VECTOR] Option

- The orient tool now has a [VECTOR] option under the Translate tab that works similar to the [N1->N2] option.



New Options for Orient Dialogue Command

New option for Orient command

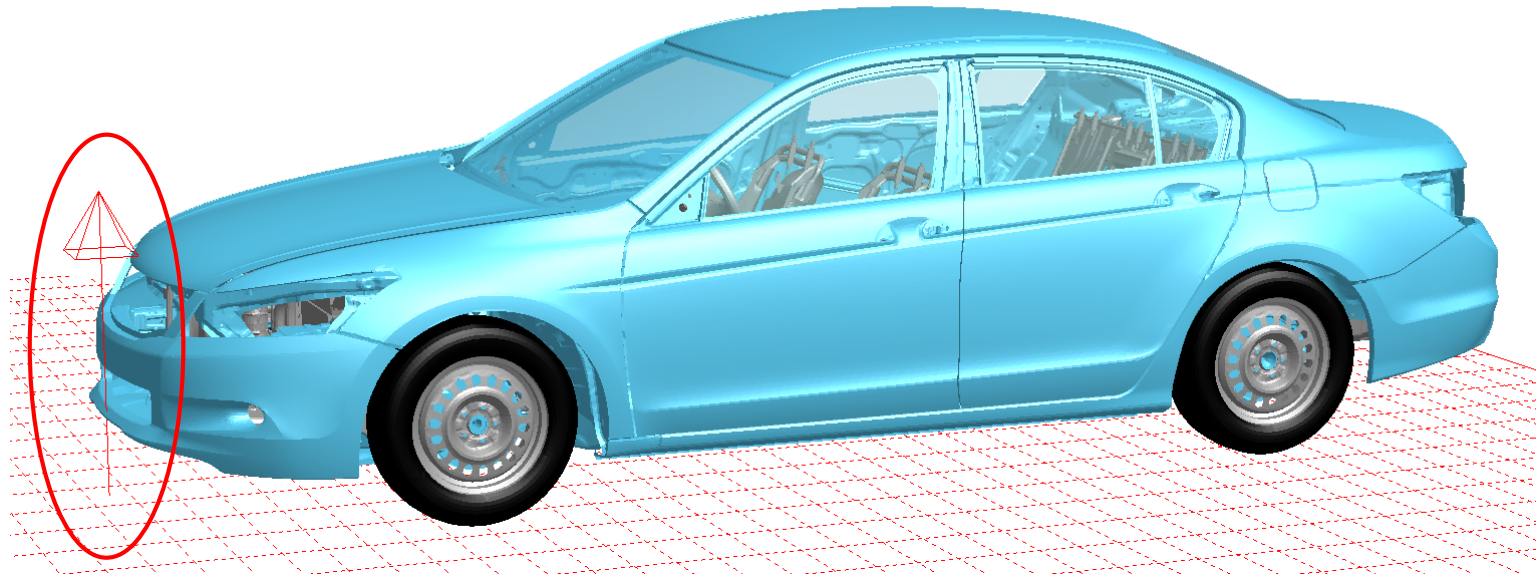
- New options for orient contact are available as dialogue input (text input) command.
- “ADD_GAP” adjusts the back off distance of the entities from the point of contact.
- “THICKNESS” (Contact Thk) over-rides the actual contact thickness if non-zero.

```
Primer >  
Primer > ORIENT  
ORIENT >>> [H for Help] ADD_GAP  
add gap: (0.0) 5.0  
ORIENT >>> [H for Help] CONTACT  
Select model for orient  
Give model number: (1) 1  
Select parts for orient [ALL/PART/SET/INCLUDE] PART  
Give list of type PART: 2  
Select model for contact (may be same or another)  
Give model number: (1) 1  
Select parts for contact [ALL/PART/SET/SHELL/SET_SHELL] PART  
Give list of type PART: 1  
Give orient vector: (1.0 0.0 0.0) 0 0 -1  
Max number of iterations: (100)  
Setting up contact for orientation ...  
Items for orient ...  
386 item(s) of type NODE  
384 item(s) of type SHELL  
1 item(s) of type PART  
Contact orient completed successfully.  
Final transform is: 0, 0, -2.39713  
ORIENT >>> [H for Help]
```

Transl into/out of contact	
Initial trans:	<input type="text" value="0.00 0.00 0.00"/> <input type="button" value="Pick 2 nodes"/>
Vector:	<input type="text" value="1.00 0.00 0.00"/> <input type="button" value="-"/> <input type="button" value="Pick 2 nodes"/>
Contact:	<input type="text" value="No contact parts/shells"/> <input type="button" value="Sk"/>
Contact Thk	<input type="text" value="0.0"/>
Increment:	<input type="text" value="<auto>"/>
Add gap:	<input type="text" value="0.0"/>
<input type="button" value="Undo"/>	
<input type="button" value="Return"/>	

Rigidwall Visualisation

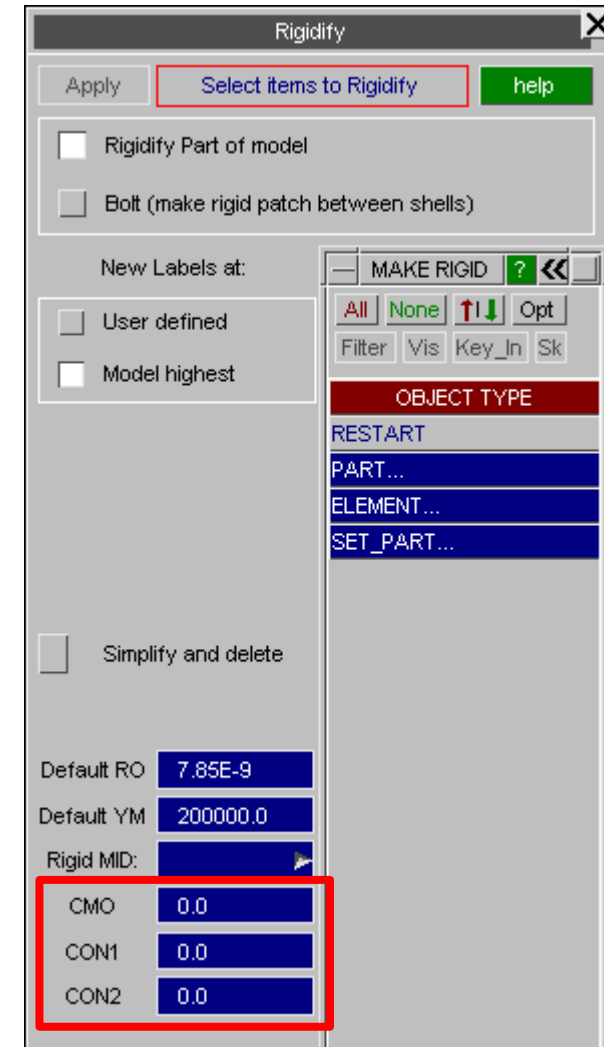
Added Visualisation of Normal Vector



Rigidify Panel Options

Rigidify Panel Options

- Extra options (CMO, CON1, CON2) are available on the rigidify panel. **(Tools->Rigidify)**.
- These allow you to set the rigid material (MAT_20) properties CMO, CON1 and CON2 directly from the panel while rigidifying parts of your model.
- These fields are available for both 'Rigidify part of model' and 'Bolt (make rigid patch)' options.



Tools->Rigidify (Part Panel)

Display and Cut-Section Options

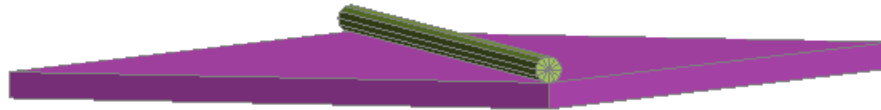
OPTT Option for True Thickness Draw of BEAMS, SHELLS

Display > Options now offers contact thickness option for shells (OPTT,SFT) and beams (OPTT).

Beam symbols

Min size: 25
Blob dia: 15

☒ True sections
☒ Use OPTT



OPTT thickness plot

OPTT = 1 for shell & beam



True thickness plot

Shell section thickness = 2

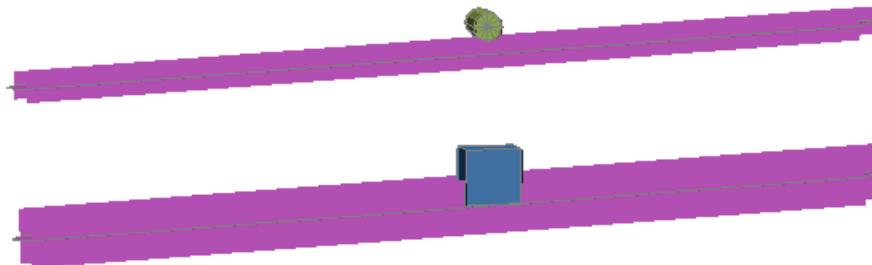
Beam section 2 x 2 rectangular

Shell symbols

☐ Thin line
☐ True flat
☐ True smooth
☒ Use OPTT/SFT

Also supported for CUT SECTION drawing

☐ No 2D element capping
☐ Use true thickness x 1.0
☐ Use fixed thickness: 1.0E-2
☐ Use Part_Contact values:
☐ Wire draw for beams
☐ Use true thickness of beam
☐ Use Part_Contact <OPTT>



FMH Markup Tool

FMH: Settings Options

FMH Impact Points Program

User-input for marking

SgRP (driver): X Y Z Sketch ?

SgRP (rear): X Y Z Sketch ?

Seat Travel: Sketch ?

Velocity: km/h ?

A-Pillar calculation: ☒

B-Pillar calculation: ☒

R-Pillar calculation: ☒

Front header calculation: ☒

Side rail calculation: ☒

Rear header calculation: ☒

A-Pillar input >>> Quit

Please only select driver side parts for parts present on both sides of vehicle.

Settings file: Select ?

READ SAVE

FMH Impact Points Program

Side rail input

Side Rail Parts ☒ Define... Sketch ?

Seatbelt Anchorage/Grab Handle (siderail) ☐ Define... Sketch ?

<<< Front header input Calculate and visualise Quit

Please only select driver side parts for parts present on both sides of vehicle.

Settings file: Select ?

READ SAVE

- Settings options added to the final panel
- Settings options on the 1st panel has been moved to the bottom of the window

FMH: New Prompts

FMH Impact Points Program

User-input for marking

SgRP (driver): X Y Z Sketch ?

SgRP (rear): X Y Z Sketch

Seat Travel: Sketch ?

Velocity: km/h ?

A-Pillar calculation: ☒

B-Pillar calculation: ☒

R-Pillar calculation: ☒

Front header calculation: ☒

Side rail calculation: ☒

Rear header calculation: ☒

A-Pillar input >>> Quit

Please only select driver side parts for parts present on both sides of vehicle.

Settings file: Select ?

READ SAVE

FMH Impact Points Program

B-Pillar input

B-Pillar (+ Cantrail) Interior Parts ☒ Define... Sketch ?

Front Door Opening Highest Point ☒ Define... Sketch ?

Front Door Opening Lowest Point ☒ Define... Sketch ?

Seatbelt Anchorage (driver) ☒ Define... Sketch ?

<<< A-Pillar input R-Pillar input >>> Quit

B-Pillar

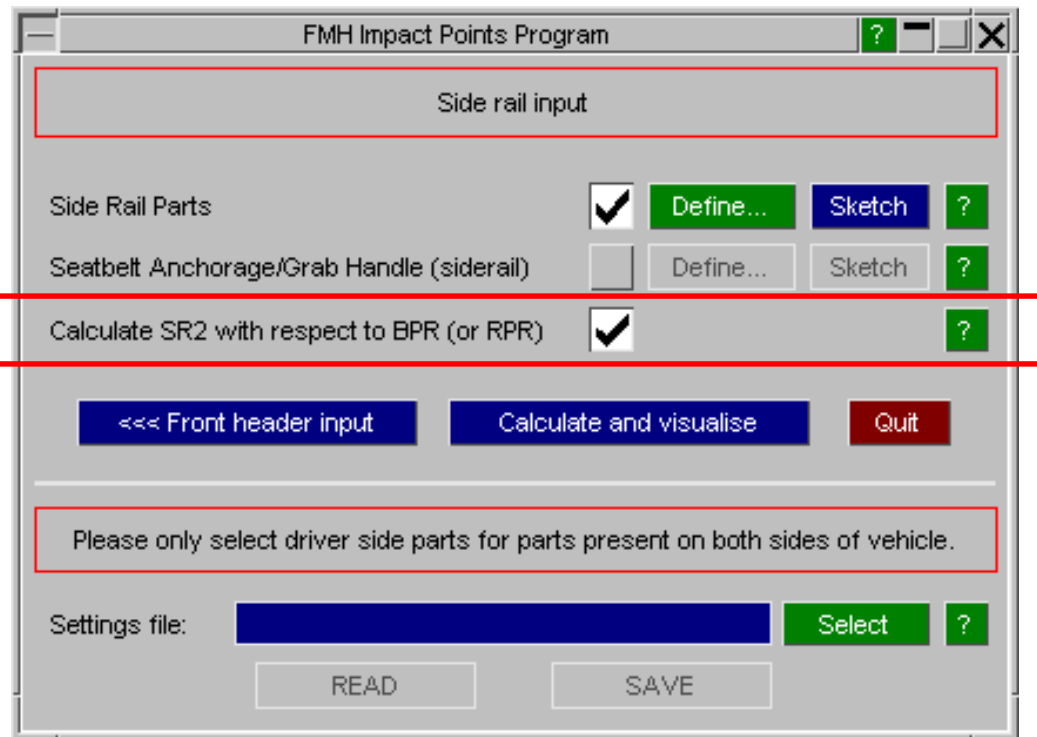
i Interior surface of B-Pillar.
This selection is required for BP1, BP3, BP4 calculation.
If window height extends beyond B-Pillar, also select the interior cantrail.

OK

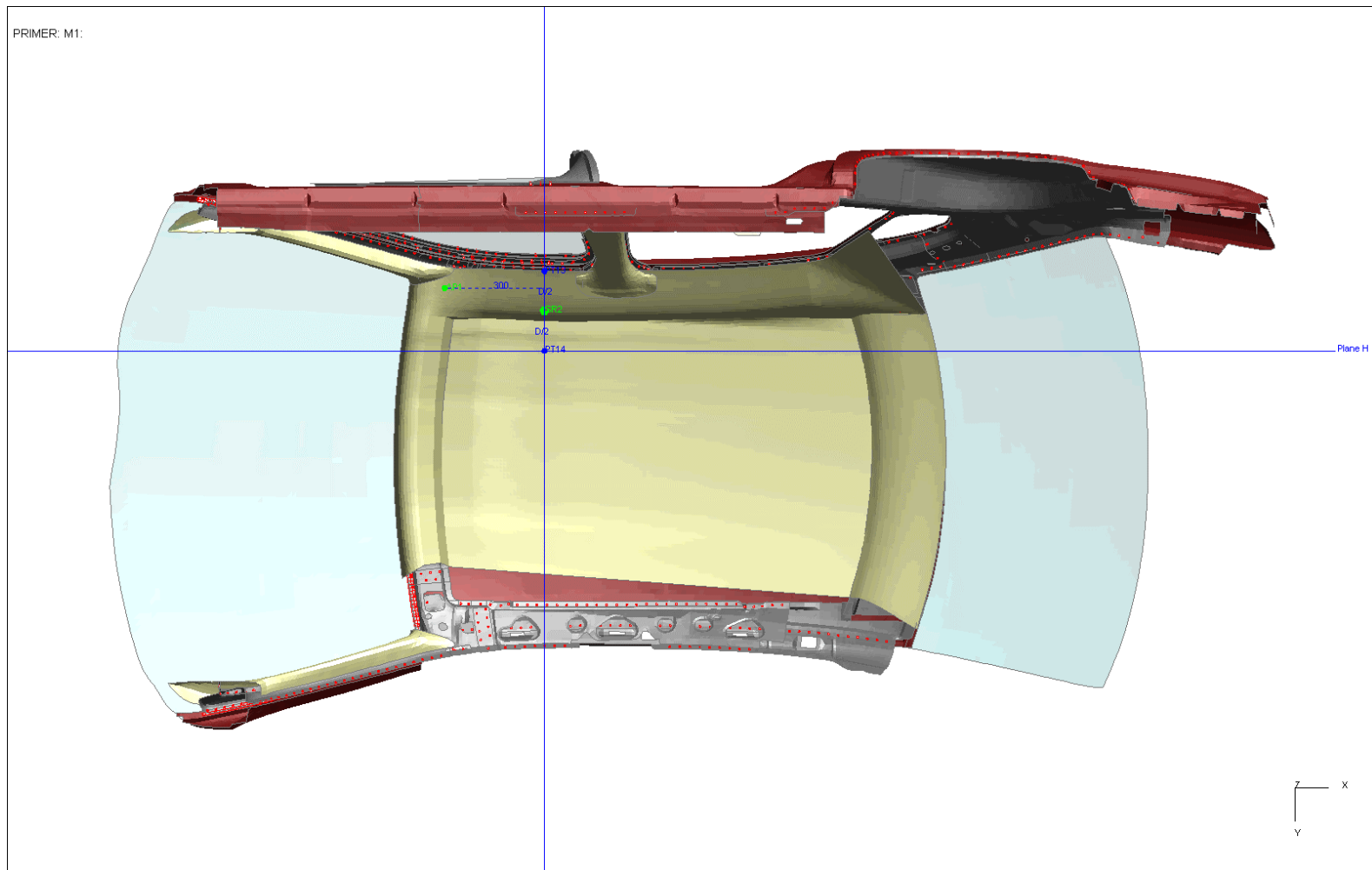
- New prompts have been added to give users more information on what to select, and also provide checking feedback.

SR2 Calculation

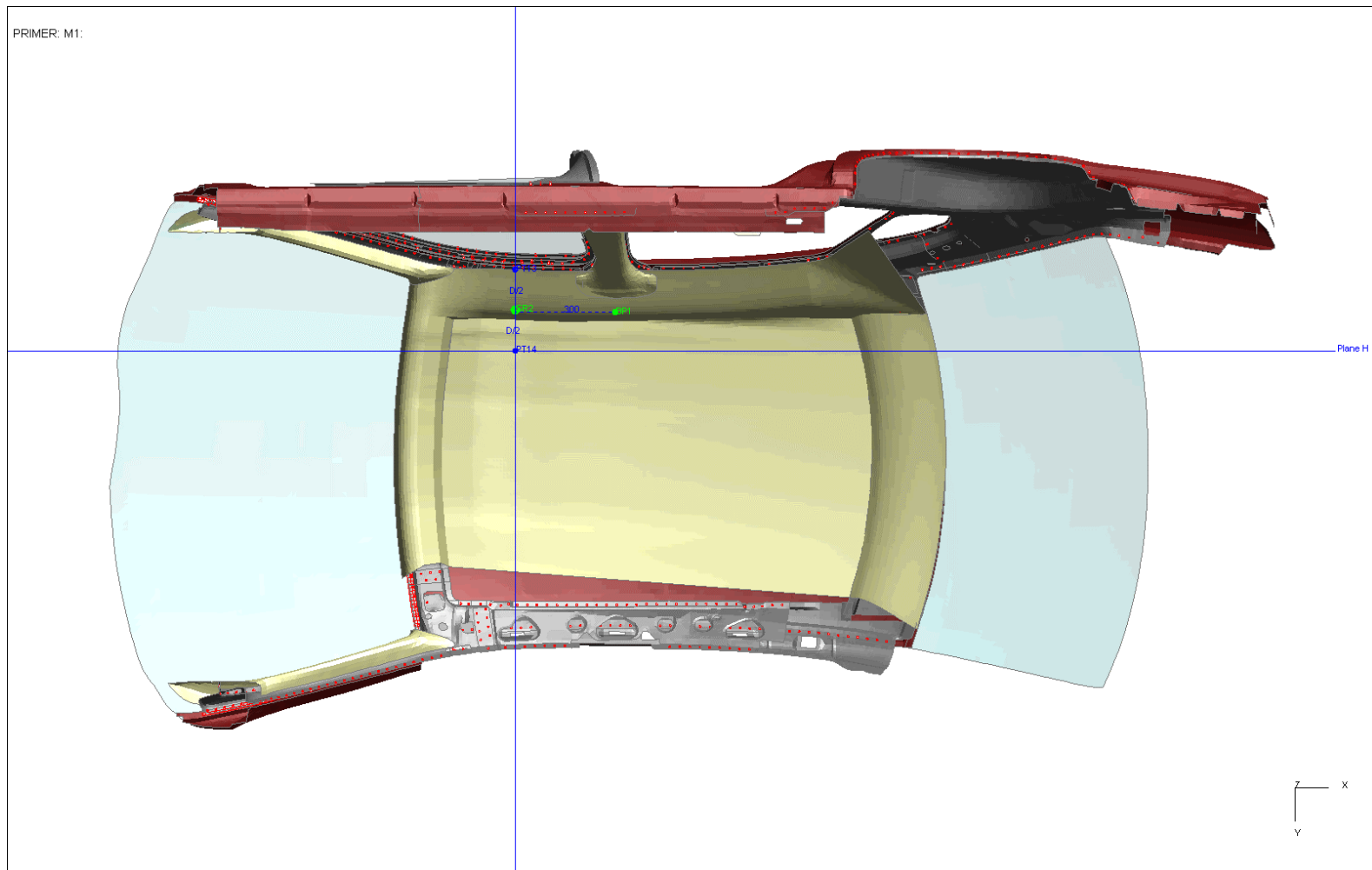
- There are now two ways to calculate target point SR2.
 - Default: With respect to APR.
 - With respect to BPR (or RPR if there is no B-pillar).



SR2 with respect to APR



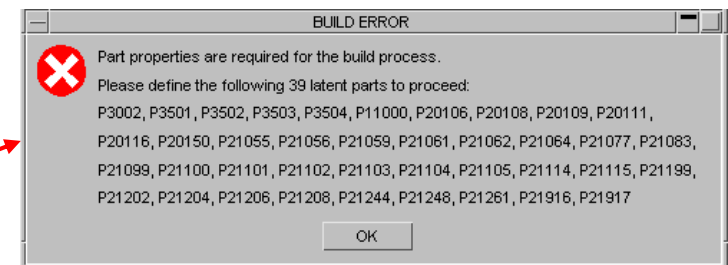
SR2 with respect to BPR



FMH: Dealing with Latent Parts

FMH Impact Points Program										
Impact points visualisation										
<input checked="" type="checkbox"/>	X	Y	Z	Horizontal approach angle			Vertical	Velocity	Sketch all	
				Max	Min	Actual	Angle			
AP1	2211.89	-507.74	1247.08	255.00	192.00	223.50	AUTO	6666.67	Visualise	?
AP2	2058.12	-572.39	1159.08	255.00	192.00	223.50	AUTO	6666.67	Visualise	?
AP3	1932.33	-608.33	1093.00	255.00	192.00	223.50	AUTO	6666.67	Visualise	?
BP1	2723.90	-435.02	1350.84	345.00	195.00	270.00	AUTO	6666.67	Visualise	?
BP2	2684.68	-652.43	1066.10	345.00	195.00	270.00	AUTO	6666.67	Visualise	?
BP3	2722.69	-616.81	1131.67	345.00	195.00	270.00	AUTO	6666.67	Visualise	?
BP4	2737.85	-683.26	1022.09	345.00	195.00	270.00	AUTO	6666.67	Visualise	?
RP1	3559.36	-376.30	1331.62	345.00	270.00	307.50	AUTO	6666.67	Visualise	?
RP2	3478.89	-664.88	1098.62	345.00	270.00	307.50	AUTO	6666.67	Visualise	?
FH1	2156.16	-391.36	1254.89	180.00	180.00	180.00	AUTO	6666.67	Visualise	?
FH2	2140.61	-243.84	1272.01	180.00	180.00	180.00	AUTO	6666.67	Visualise	?
SR1	2361.89	-446.82	1325.48	270.00	270.00	270.00	AUTO	6666.67	Visualise	?
SR2	2511.89	-443.72	1338.42	270.00	270.00	270.00	AUTO	6666.67	Visualise	?
SR3	2873.90	-442.16	1346.06	270.00	270.00	270.00	AUTO	6666.67	Visualise	?
RH	3572.81	-373.00	1329.45	0.00	0.00	0.00	AUTO	6666.67	Visualise	?
0 added										
0 added										
Back to user-input Create manually Robustness pts Read/Write csv Build ... Quit										

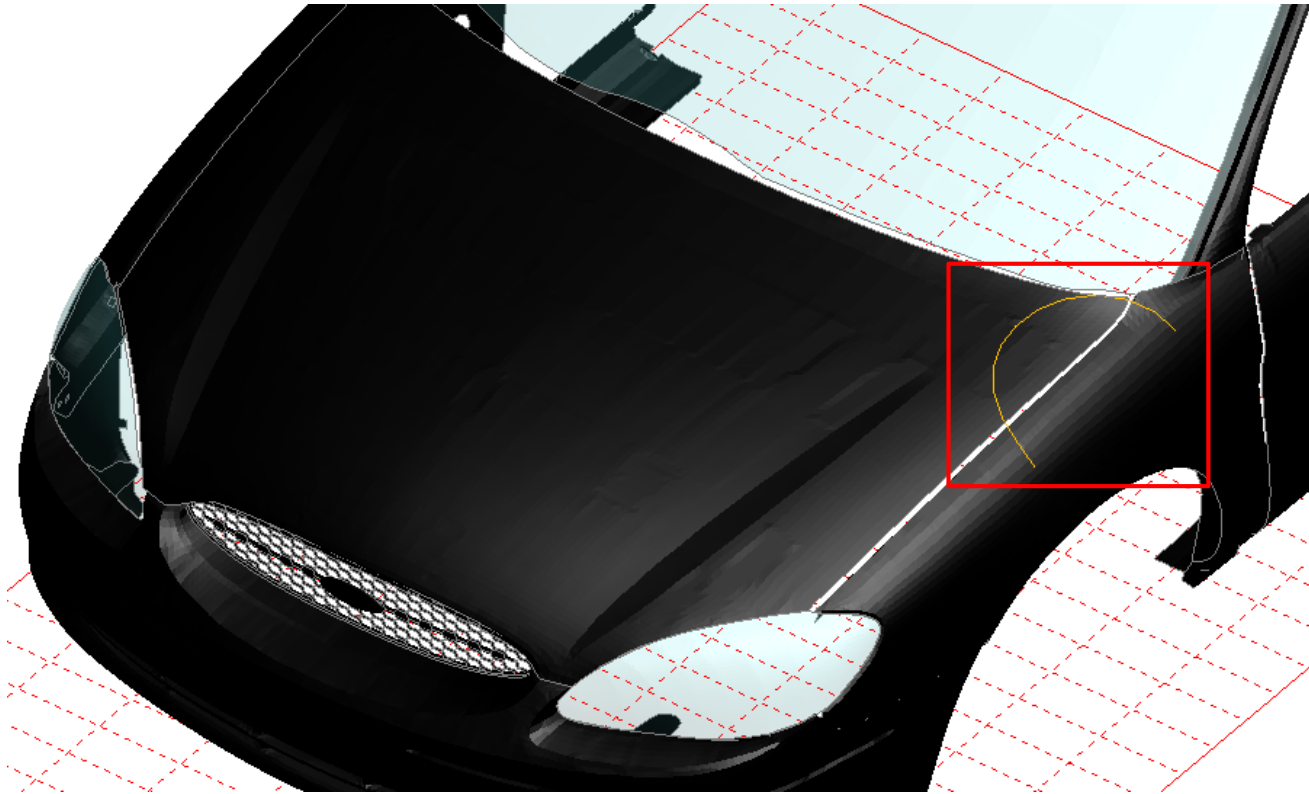
- Impact points can now be calculated for models with latent parts – this is useful for situations where you are using quick meshes from CAD packages to calculate target points, where section cards and therefore thickness is not defined.
- However with latent parts the “build” option is not available – this requires parts with correctly defined section properties.



Pedestrian Markup Tool

Pedestrian Markup Tool

The model output by the pedestrian markup tool containing the “sticks” and mark-up information now also contains the outline of the semi-circular template used to join the rear reference line with the side reference line.



Images

PRIMER 16 introduces the following changes to the handling of output image filenames:

- If an explicit filename (optionally, with path) is typed in or selected for one image type, say JPEG, that new filename and path will then become a seed for every image type. Changing to a different image type, say PNG, will not require users to browse to the user-defined path again. Instead the new seed name will be used for the PNG file with the appropriate extension (.png) and a unique increment <nnn> added if needed.
- Extensions that are an obvious mismatch will be auto-corrected and a warning issued. For example, if a user attempts to write out a BMP with the filename myfile.gif, the filename will be changed to myfile.bmp automatically. The filename myfile.image will, however, not be auto-corrected.
- If a user-chosen filename exists, clicking the 'NEW_FILE' option will not take the user back to the current working directory as was done in the past. Instead, a unique 3 digit increment <nnn> will be appended to the same name to generate a new filename.
- A sub-folder (one level only) is now also automatically created as a part of image creation, if needed.
- All of the above now also applies to postscript files (PS and PDF), although the handling of images and postscript files is kept disparate. So, choosing a new image name will not alter the default filename for movie or postscript files. Choosing a new postscript filename will, however, update the default filename for pdfs.

JavaScript API

- Various JavaScript API enhancements include:
 - The object returned from the Dummy GetPoint() method now includes a 'hpt' property that is true if the point has been automatically generated by PRIMER at the H-point.
 - There are now new classes MorphPoint and MorphBox for the existing morphing keywords as well as MorphFlow for the new post-*END keyword *MORPH_FLOW.
 - A new class function FlagVisible() is added for Part class. The API can flag all the visible or unblanked parts in a model.
 - The part class function CentreOfGravity has been re-written to allow user control of the mass calculation. The following options are supported, for example:

```
var cofg = p.CentreOfGravity({slaveparts:true, skipslave:true, transfermass:true,  
                             lumpedmass:true, nrbmass:false, timestep_mass:false,  
                             plot:true});
```
- Updates to keyword classes for R11.0 (see manual).

Model Check

Model Check Improvements

- Up to 20% faster for a typical model, **very** model content dependent.
- Substantially (2x or better) faster for models containing many solids and thick shells when these are used in tied contacts.
- Substantially (2x or better) faster for models containing many adhesive and spotweld connections. (The speed of use of the connection table also benefits from this.)

Much of the speed up comes from further parallelisation of bottle-necks, and the improvement will depend heavily on model size, content and organisation. Also on the number of available cores.

Generally the larger the model the more it will benefit from parallelisation, as splitting tasks up over cores has a fixed overhead, but larger models require more useful work. In a large model the ratio of

(useful work / overhead)

improves, because the length of a “run” of useful work per core gets longer. Large models will also benefit more from a greater number of cores for the same reason. PRIMER currently limits this sort of parallelism to $\min(\text{\#cores}, 16)$ threads by default, but this limit can be changed.

Hyper-threading (2 logical cores per physical core) helps a little, perhaps 10% in a large model.

Element Quality Checks

Element Quality Checks

Two new element quality checks have been added:

- Max taper (SHELL elements only)

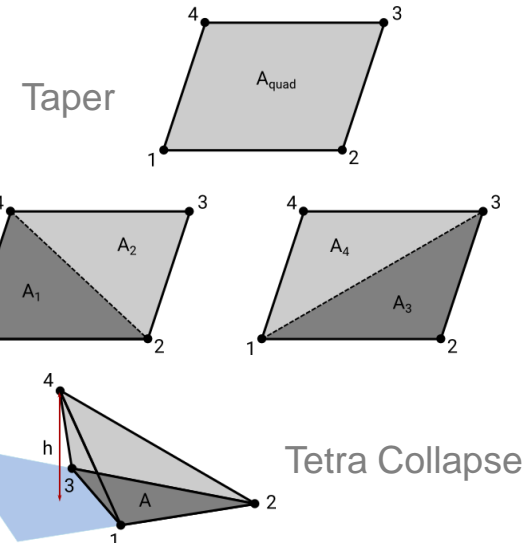
$$taper = 1 - \left(\frac{2A_i}{A_{quad}} \right)_{max}$$

- Min tetra collapse (SOLID elements only)

$$tet\ collapse = \left(\frac{h_i}{1.24\sqrt{A_i}} \right)_{min}$$

Checking can be toggled on/off as well as the failure criteria and weighting factors specified via the 'CHECK OPTIONS' panel.

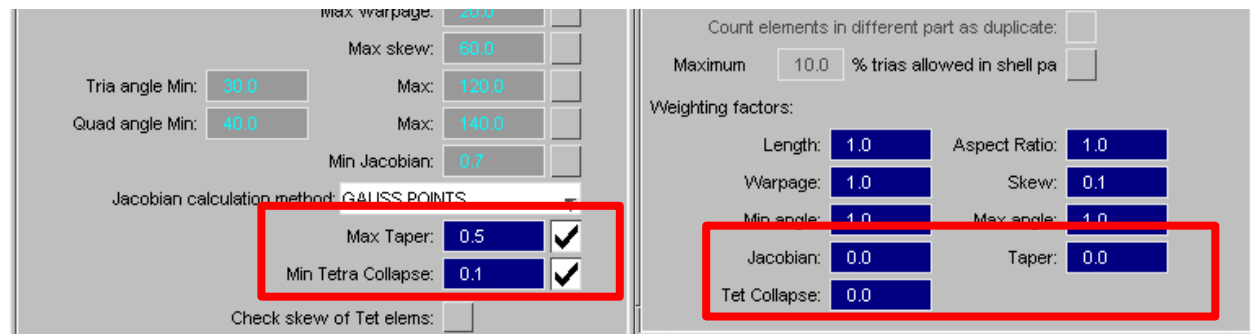
(weighting factors are used in giving quality checks precedence in 'Quality Imperfection' and 'Failed Criteria' CT/SI plots)



Taper & Tet Collapse values can also be obtained via the JavaScript API:

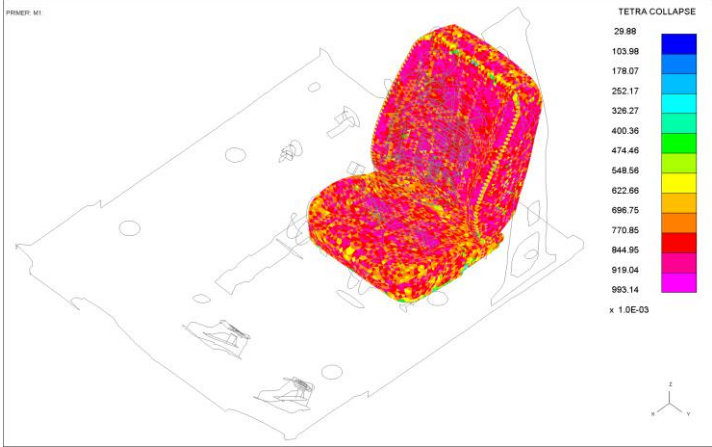
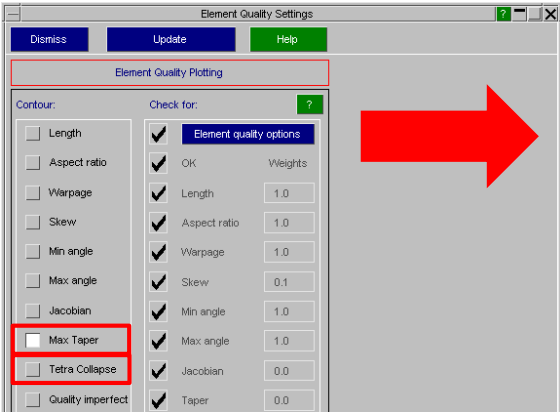
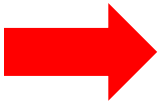
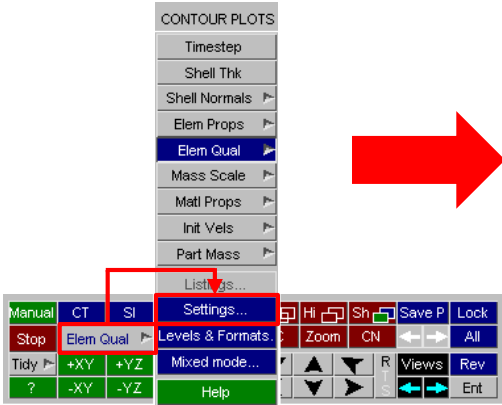
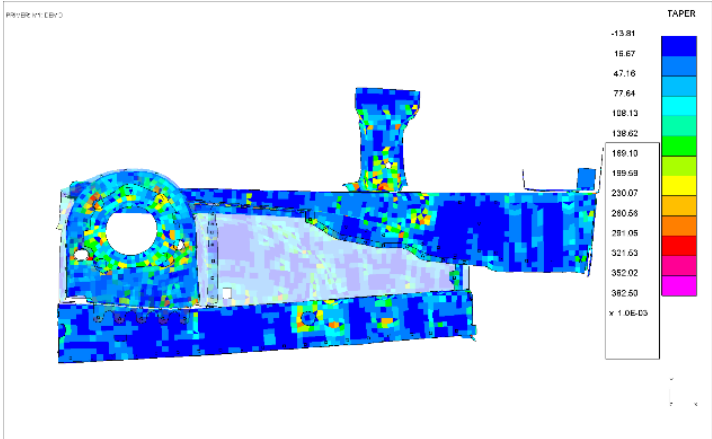
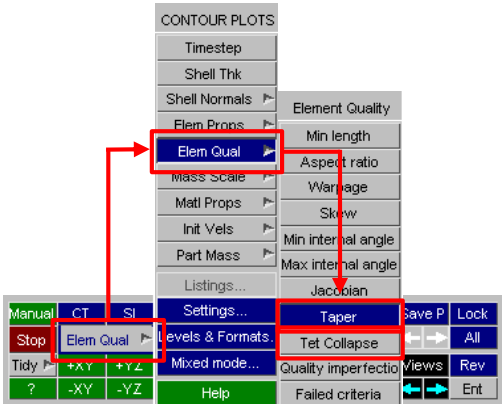
s.Taper() - (where s is a SHELL object)

s.TetCollapse() - (where s is a SOLID object)



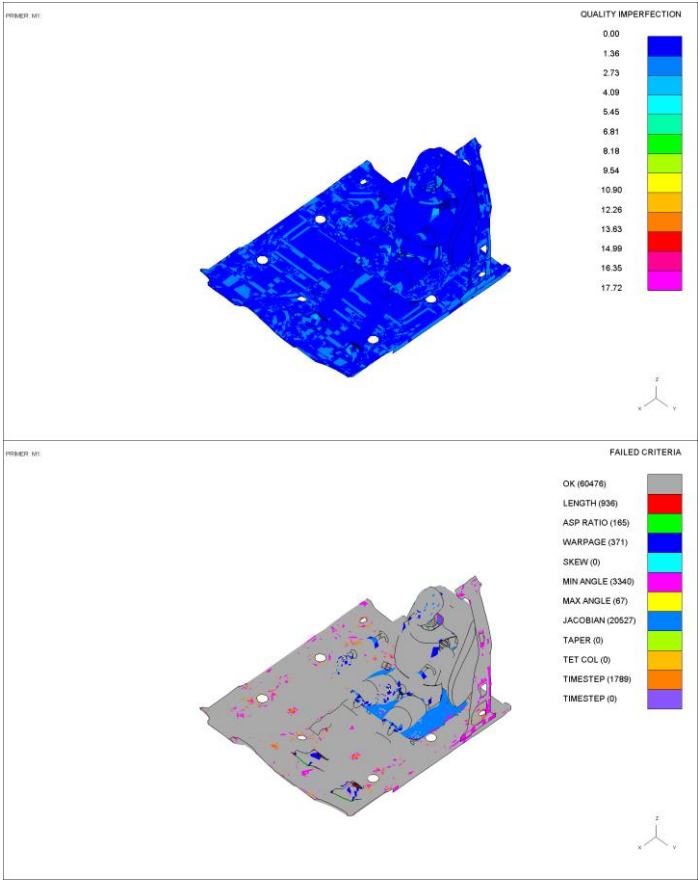
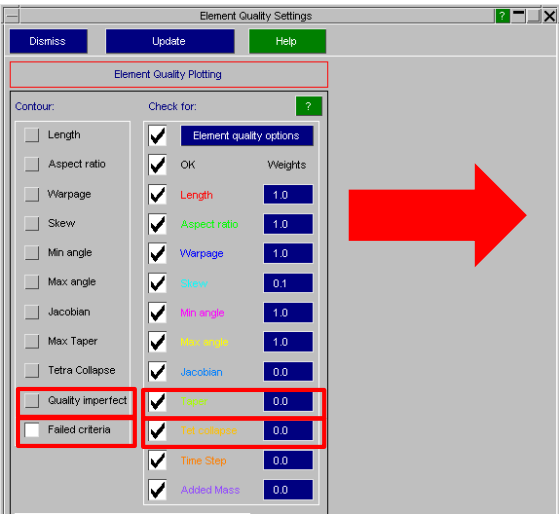
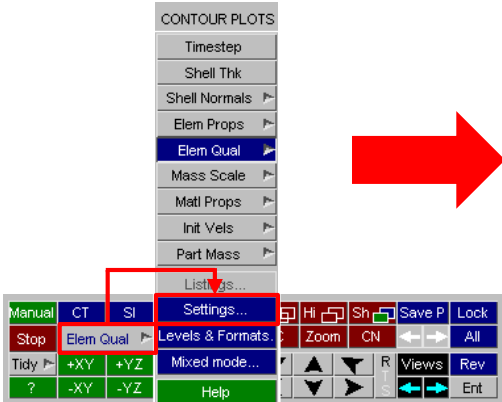
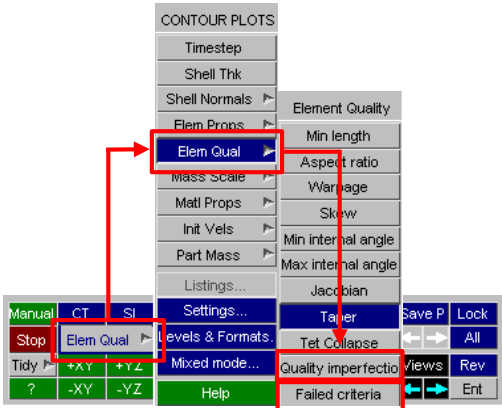
Element Quality Checks

These element checks can be displayed as contour plots:



Element Quality Checks

These checks can also feed into Quality Imperfection and Failed Criteria contour plots:



Element Quality Checks

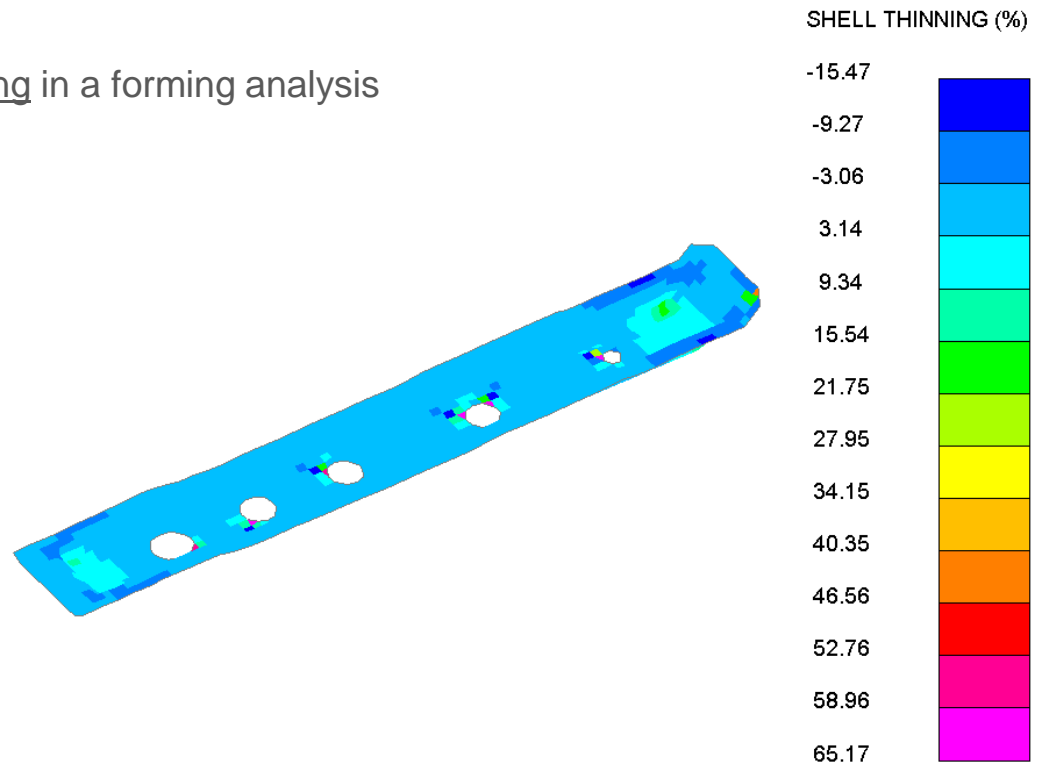
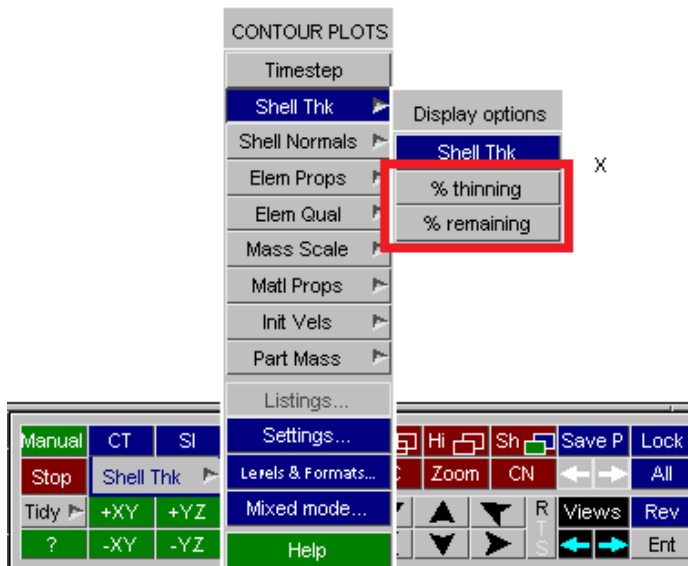
Two new options under “Shell Thk” have been added:

- % thinning

This option contours the % thickness reduction in a forming analysis

- % remaining

This option contours the % thickness remaining in a forming analysis

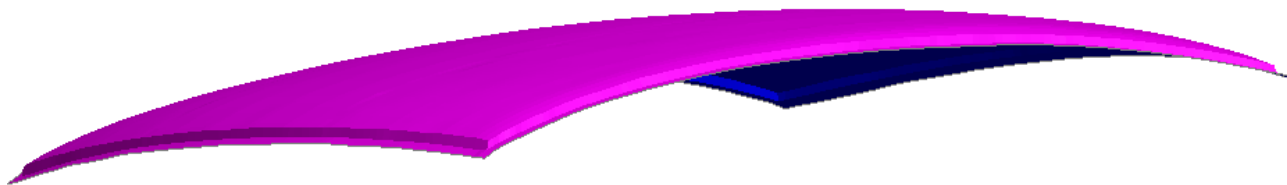
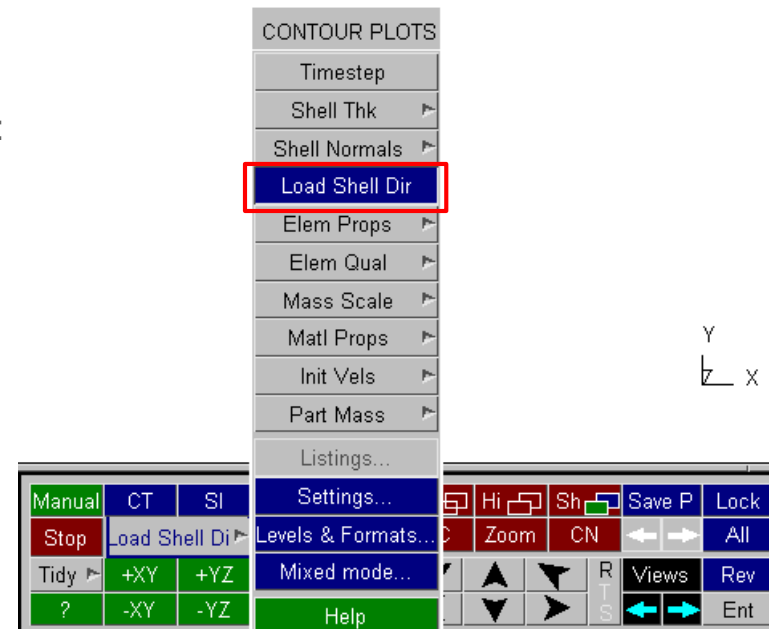


Element Quality Checks

New contour option: LOAD SHELL DIRECTION.

This adds the ability to contour LOAD SHELL initial direction:

- Blue Loading in away direction.
- Magenta Loading in towards direction.
- Yellow Loading in both directions.
- Grey No loading.



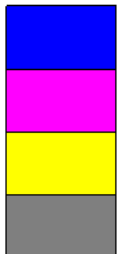
LOAD SHELL DIRECTION

AWAY

TOWARDS

BOTH

NONE



Element Quality Checks

A **JavaScript** function has been added to display a contour plot of any of the Quality Checks:

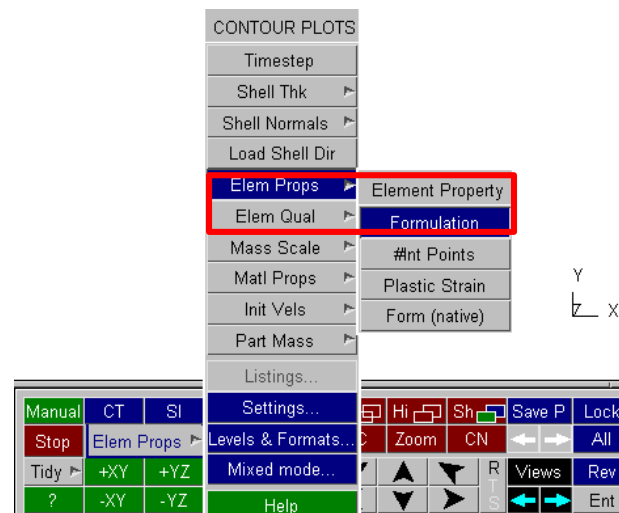
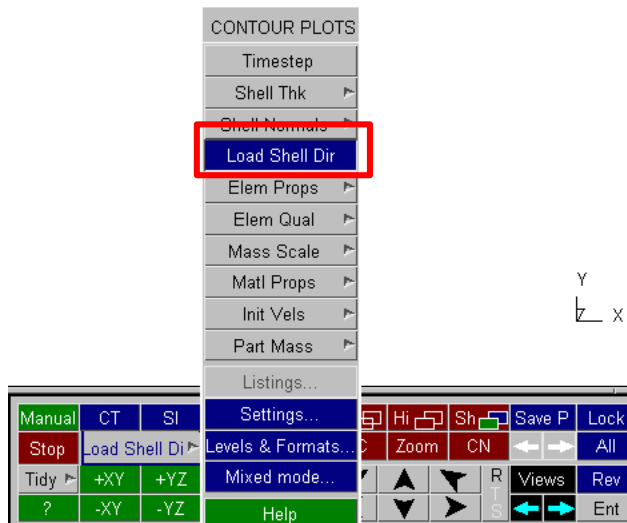
View.SetContourType

The function takes 1 or 2 arguments, depending on what you want to contour:

- Type.
- Subtype.

Examples:

- `View.SetContourType(View.LOADSHELLDIRECTION);`
- `View.SetContourType(View.ELEMPROPS, View.FORMULATION);`



LS-DYNA Output Files

LS-DYNA Output Files

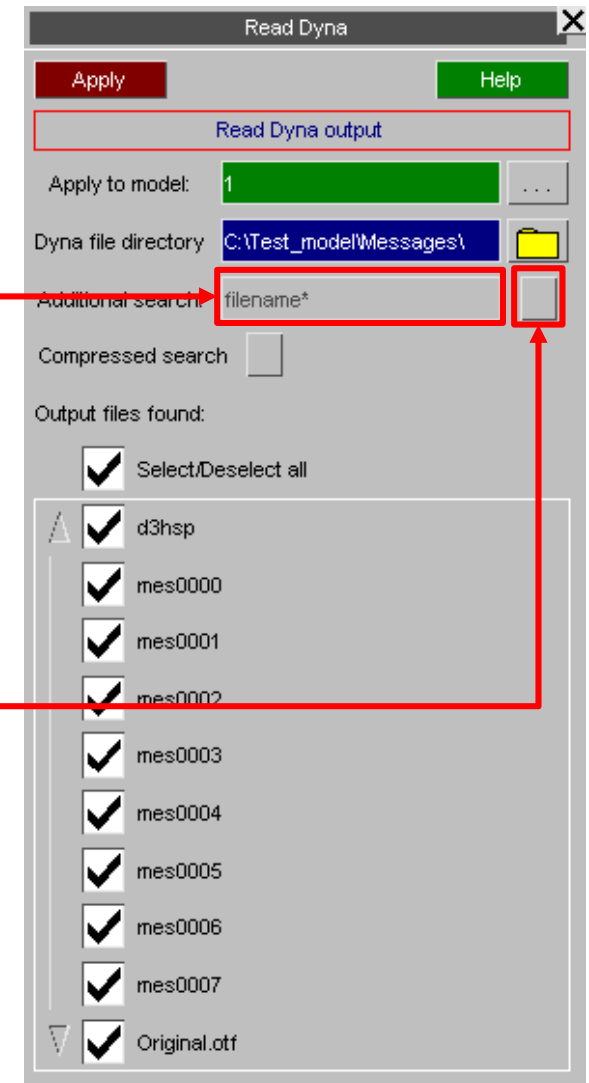
Tools→Check→DYNA Output.

More search criteria:

- mes*
- *.mes*
- *.d3hsp
- User specified 'Additional search'
- Compressed file search

Additional search:

- Turn on or off via check box
- Can use wildcards:
 - '?' – one character
 - '*' – any number of characters
- Can save as preferences:
 - 'additional_dyna_output' – search string
 - 'additional_dyna_output_search' – search turned on/off (TRUE/FALSE)



LS-DYNA Output Files

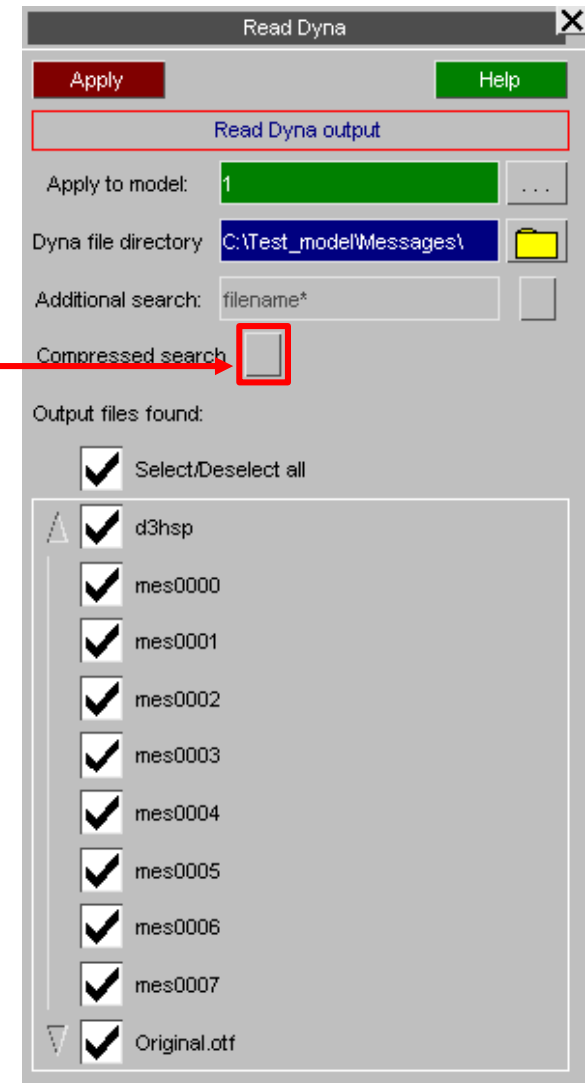
Tools→Check→DYNA Output

More search criteria:

- mes*
- *.mes*
- *.d3hsp
- User specified 'Additional search'
- Compressed file search

Compressed search:

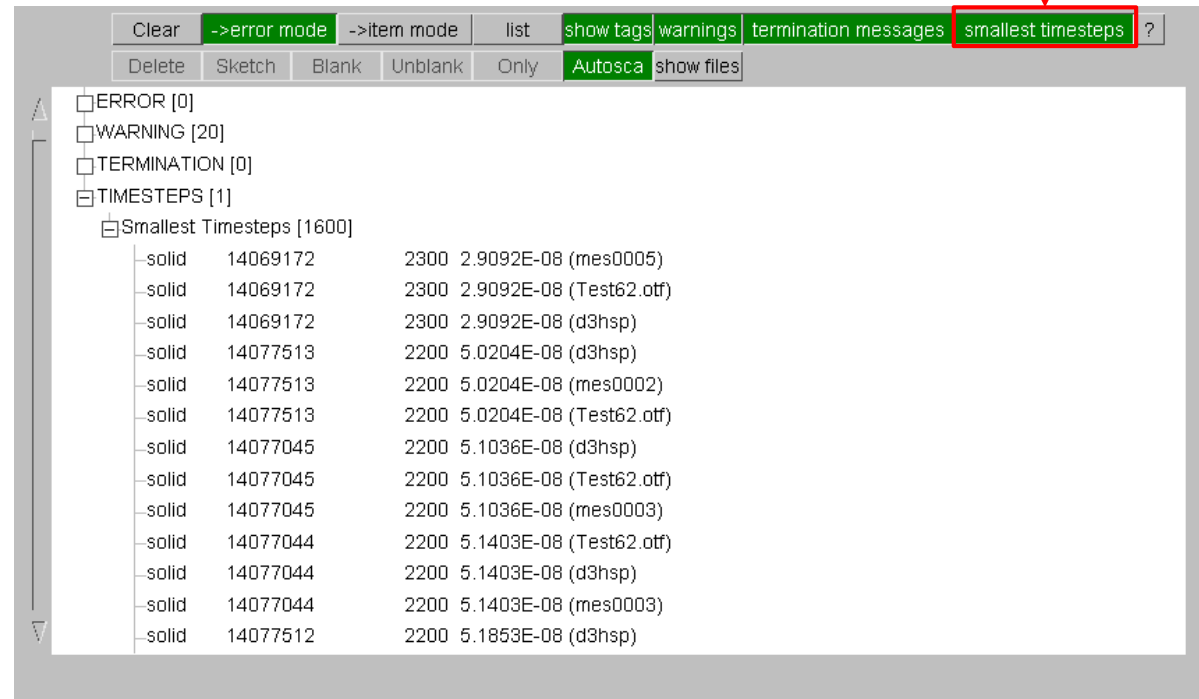
- Turn on or off via check box.
- Searches for files ending in “.gz” or “.zip”.
- When such files are selected, clicking 'Apply' will open them and present their data just like any other non-compressed file.



LS-DYNA Output Files

You can now view 100 smallest timesteps data:

- Viewing of timestep messages can be toggled on/off with the 'smallest timesteps' button.
- Error mode will show messages in order of ascending timestep value.
- Item mode will show messages in order of ascending entity label.
- As with existing warnings, if the entity being referred to exists, further operations can be performed (Only, Sketch, Edit etc.).



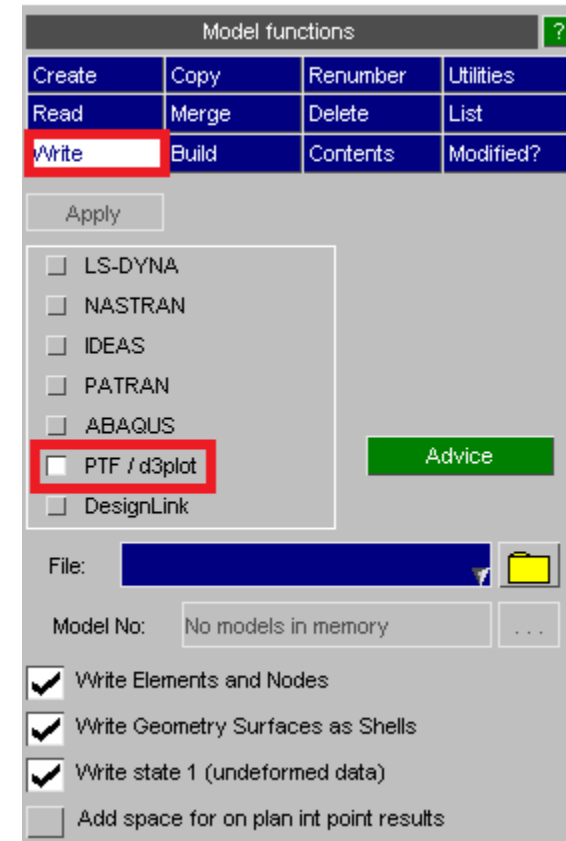
PTF / d3plot Write

PTF / d3plot Write

Prior to v16, if a model contained

*INITIAL_STRESS_(T)SHELL cards with multiple on plan points ($NPLANES > 1$), the PTF / d3plot writer would only write data for the first integration point on each layer.

This restriction has been removed in v16 and it now writes the data for each on plan point.



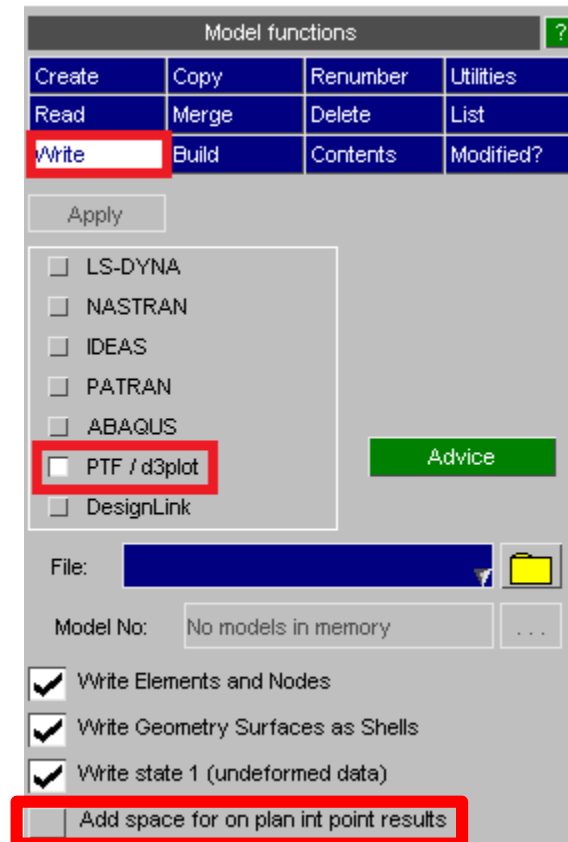
PTF / d3plot Write

A new option has been added to the PTF / d3plot writer to add space for on plan integration point results.

If the file doesn't contain any *INITIAL_STRESS_(T)SHELL cards the file will only contain one data slot for stress data per (T)SHELL. Users may want to create a JavaScript user defined UBIN component in D3PLOT to add values to the model and this limitation means they can only write it at one layer and one on plan point.

Selecting the option to add space for on plan integration point results will make PRIMER write a file with data slots available for multiple on plan points (the number of layers will still be limited to 1).

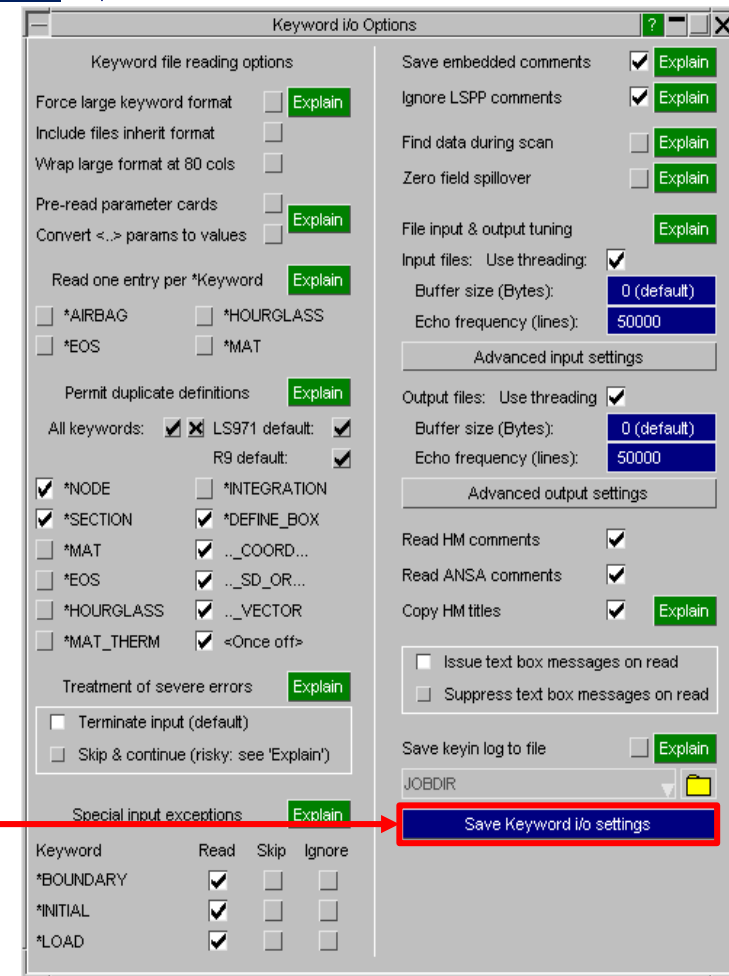
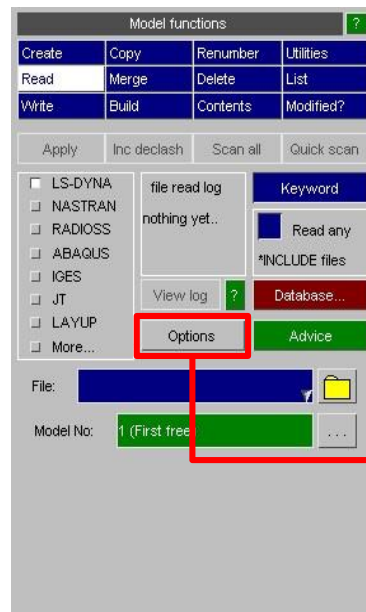
NOTE: The ELFORM field on the *SECTION card needs to be set to a formulation that supports multiple on plan integration points otherwise D3PLOT will still think it only has space for one on plan point.



Preferences

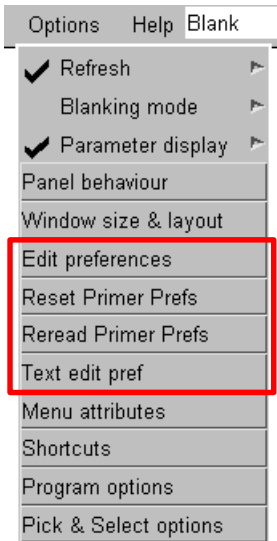
Read – Save Preferences

- “Model Read” options/preferences can be saved using the ‘Save Keyword i/o settings’ button
- They can also be edited in the edit preferences panel, under the PRIMER->input tab
- New preferences were added to allow for the ‘Save Keyword i/o settings’ button:
 - force_keyword_format
 - input_exception_BOUNDARY
 - input_exception_INITIAL
 - input_exception_LOAD
 - large_format_80cols
 - preread_parameters
 - single_card_ABAG
 - single_card_EQOS
 - single_card_HGLS
 - single_card_MATL



NOTE: ‘Customised’ selections of ‘Permit duplicate definitions’ currently cannot be saved as preferences

Text Edit of Preference File



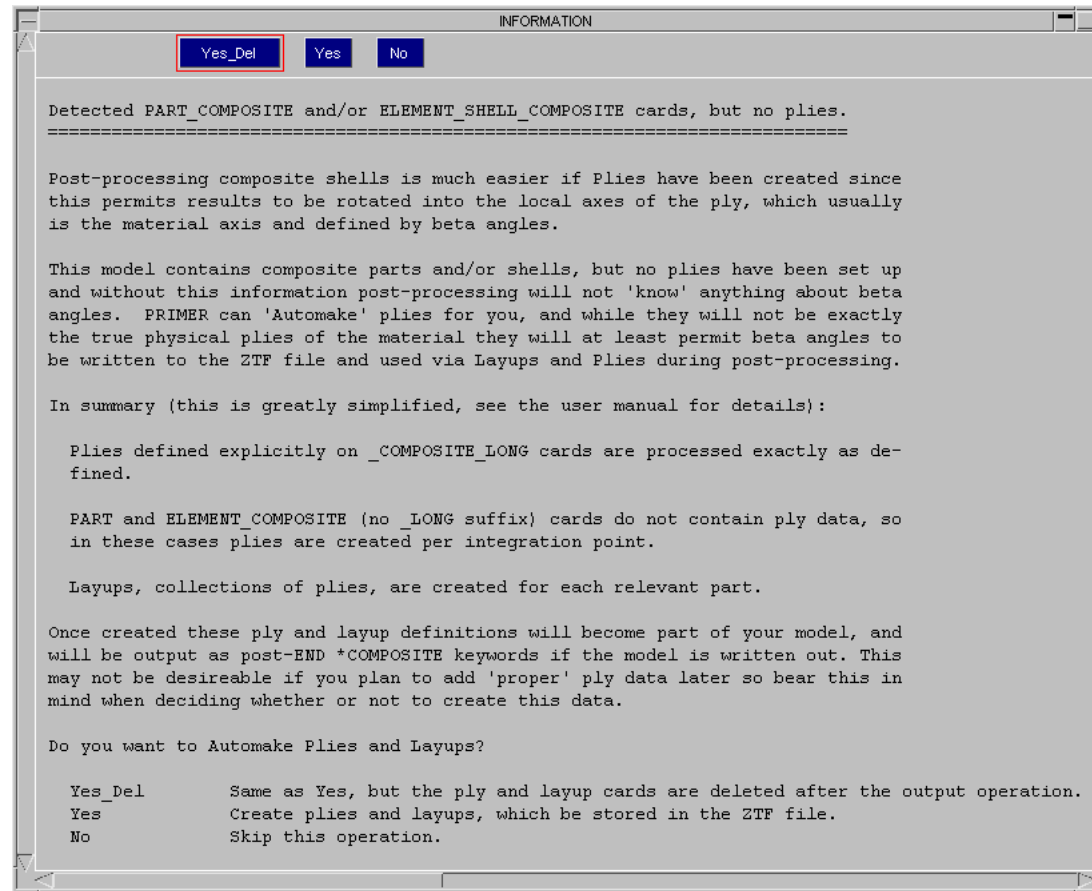
- In addition to the preference editing tool, PRIMER now supports text edit of a preference file.
- User gets choice to edit Home or Install oa_pref (if permissions allow).

How to load modified preferences to current session

- If preferences are created/modified but not deleted REREAD PRIMER PREFS will load the preference files without loss of any current settings. This process will effectively ignore deleted prefs – those settings will be unchanged.
- If preferences have been deleted RESET PRIMER PREFS will reset all possible preference settings to default and then load the preference files.
- Both options will read Admin, Install, Home and cwd preference files in that order.

New Preference for Composite ztf Output

- When writing a ztf file for a model containing composite data, without any PRIMER ply information, The below message appears, giving the user the option to create PRIMER ply information to aid post-processing of composite information.
- New preference 'ztf_output_composite_action' can be used to preselect the option on this dialogue box (set to 'Yes_Del', 'Yes', or 'No').



Licensing 16.0

Licenses for multiple programs

A PRIMER server license can be extended to be used by the post software. Short-term trial licenses for the post software are also available. Please contact your local distributor.

Situation	License details
Extend PRIMER server licenses to work with D3PLOT and T/HIS.	<p>primer + primer_post server license</p> <p>An individual using PRIMER and D3PLOT on their machine will use two PRIMER licenses.</p> <p>An individual using PRIMER, D3PLOT and T/HIS will also use two PRIMER licenses. In this instance, D3PLOT and T/HIS share a PRIMER license.</p> <p>D3PLOT and T/HIS can be prevented from using PRIMER licenses by setting the following preference in the oa_pref file.</p> <p>oasys*post_uses_primer = FALSE</p>

Licenses for multiple programs (trial access)

Situation	License
PRIMER with trial access to D3PLOT and T/HIS for server licenses.	<p>primer + post_trial server license</p> <p>An individual can use PRIMER, D3PLOT and T/HIS simultaneously on their machine. Doing so will use one 'primer' server license and one 'post_trial' server license.</p> <p>Once the trial license expires, only PRIMER can be accessed.</p>
PRIMER with trial access to D3PLOT and T/HIS for node-locked licenses.	<p>primer + post_trial node-locked license</p> <p>You can use PRIMER, D3PLOT and T/HIS simultaneously on your machine.</p> <p>Once the trial license expires, only PRIMER can be accessed.</p>
D3PLOT with trial access to T/HIS. This will enable you to use the T/HIS link.	<p>d3plot + post_trial</p> <p>You can use D3PLOT and T/HIS simultaneously on your machine.</p> <p>Once the trial license expires, only D3PLOT can be accessed.</p>

Licenses for single or multiple programs

You can buy server or node locked licenses which allow access to all programs in the Oasys Suite.

It is also possible to buy server or node locked licenses for individual programs.

Program	License required
PRIMER	primer
D3PLOT	d3plot
T/HIS	this
D3PLOT and T/HIS link	d3plot + this
REPORTER	reporter*
All programs in the Oasys Suite	primer, d3plot, this + reporter

*REPORTER is also able to run without the reporter license. In this situation the software checks for any available Oasys license and then releases it again.

Restricted-use licenses for all programs

We provide restricted-use licenses free of charge through our website www.arup.com/dyna for certain domain names, for example for students at UK Universities.

These licenses can also be sold commercially by a local distributor.

Similar licenses are available restricted to a different number of nodes or curves.

Program	License
All programs in the Oasys Suite, restricted to models with less than 10,000 nodes (PRIMER and D3PLOT) and 12 curves (T/HIS).	primer, d3plot and this licenses restricted

REPORTER

If you have licenses for any of our programs, you are licensed to use REPORTER to interact with that program.

For example, if you have a “primer” license, you can use PRIMER, or use REPORTER to create reports with PRIMER objects in them.

SHELL

The Oasys SHELL can be used without a license.

FAQs

Using the extended PRIMER license

1. License check-out priority - When customer has both the extended PRIMER license (primer + primer_post) and D3PLOT license (d3plot), which license will be checked out first when running D3PLOT?

D3PLOT will always look for a D3PLOT license first. If no D3PLOT licenses are available, it will then try and use a PRIMER license. If D3PLOT uses a PRIMER license then by default a window is displayed warning the user that they are using a PRIMER license.

2. Can a user on one machine have a unlimited number of PRIMER sessions with one primer license (as was previously the case)?

This hasn't changed if they are running versions 16 or 15. In versions 16 or 15 if a user runs multiple copies of PRIMER on the same machine then they all share a single license.

If a user runs version 14 and either version 16 or 15 at the same time then they will count as 2 separate licenses.

Using the extended PRIMER license (CON'T)

3. Can you let me know how to prevent D3PLOT and T/HIS using a PRIMER license?

To disable license sharing the following preference can be set:

d3plot*post_uses_primer: FALSE
(disables D3PLOT from using a PRIMER license)

this*post_uses_primer: FALSE
(disables T/HIS from using a PRIMER license)

oasys*post_uses_primer: FALSE
(disables D3PLOT and T/HIS from using a PRIMER license)

4. How many extended PRIMER licenses does an individual use?

D3PLOT (or T/HIS) using a PRIMER license is counted separately to a user running PRIMER so an individual on one terminal, running the following combinations of program uses these licenses:

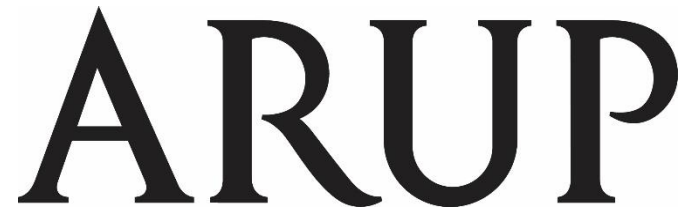
1)	1 x PRIMER only	:	1 primer license
2)	2 x PRIMER	:	1 primer license
3)	PRIMER + D3PLOT	:	2 primer licenses
4)	2 x PRIMER + 2 x D3PLOT	:	2 primer licenses
5)	PRIMER + T/HIS	:	2 primer licenses
6)	PRIMER + T/HIS + D3PLOT	:	2 primer licenses (D3PLOT and T/HIS share)

Changing from FLEXlm to LMX licensing

We are planning to discontinue the use of FLEXlm licensing in our software from version 17.0 onwards. We will instead use LMX licensing. Support for LMX licenses is included in our Oasys suite versions 15.x and 16.x software. The table shows which Oasys suite versions work with each license type.

LMX licenses will be available later in the year for testing purposes.

Oasys Suite version	FLEXlm licensing	LMX licensing
14.x and earlier	✓	✗
15.x and 16.x	✓	✓
17.x onwards	✗	✓



www.arup.com/dyna

For more information please contact the following:

UK Contact:

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

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

China Contact:

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

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

India Contact:

Arup India
Ananth Info Park, HiTec City
Madhapur Phase-II
Hyderabad
India
500081, Telangana

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

USA West Contact:

Arup Americas
c/o 560 Mission Street Suite 700
San Francisco
United States
CA 94105

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

or your local Oasys distributor