



LS-PREPOST Training

8th International LS-DYNA Users Conference

Post Conference Training

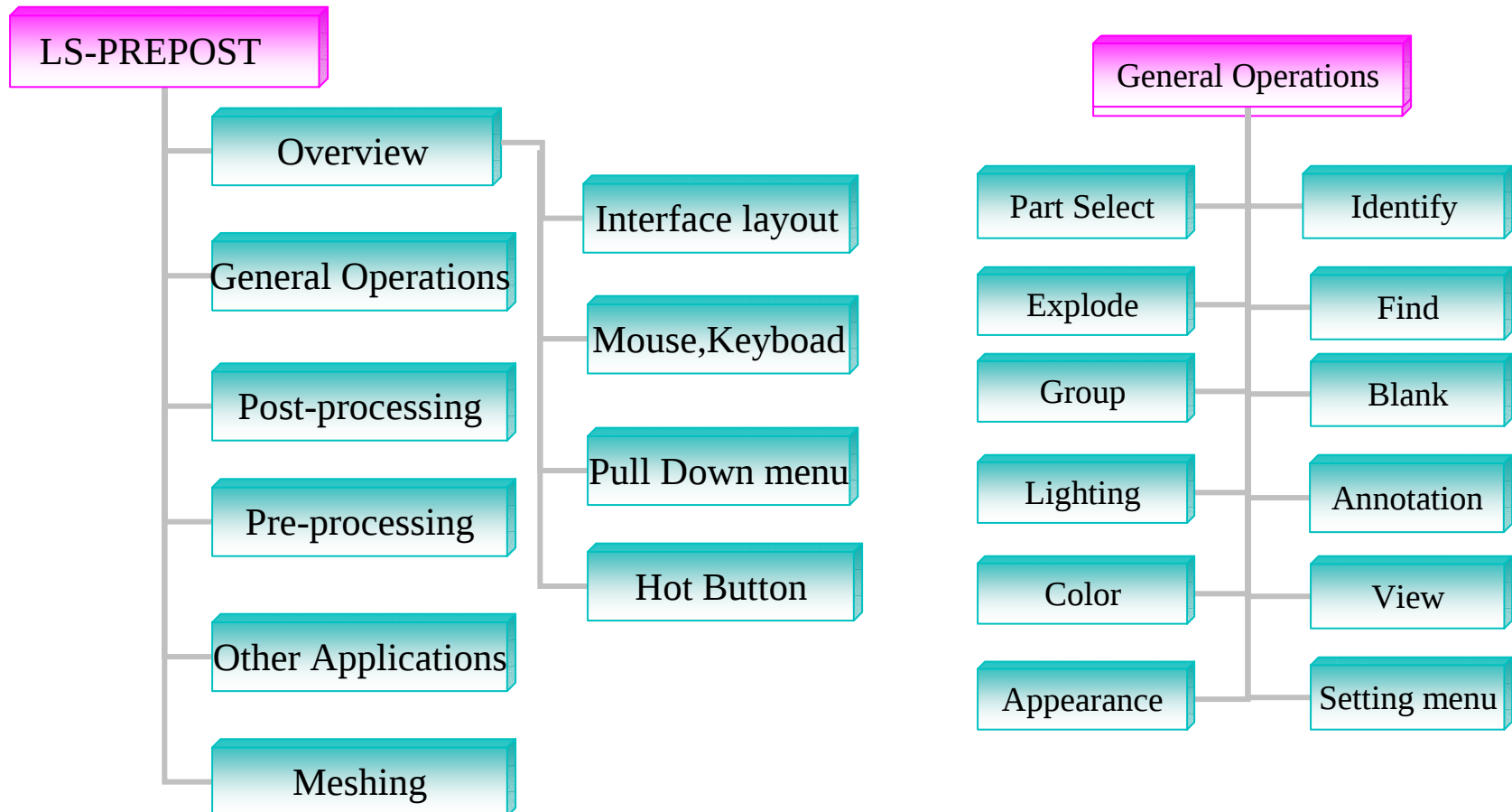
May 5 - 6, 2004

Introduction

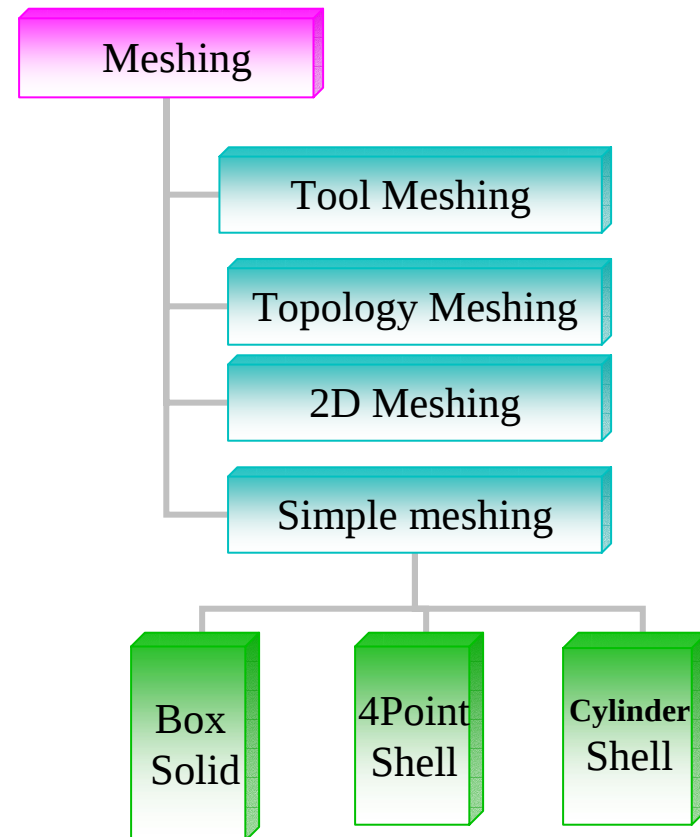
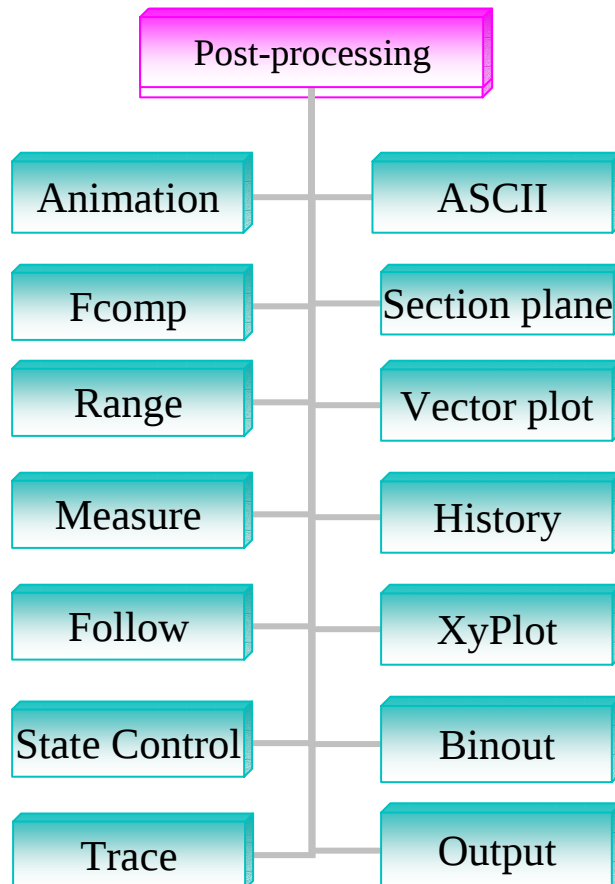
LS-PREPOST is an advanced pre/post processor for LS-DYNA

- ❑ Full Support of LS-DYNA Keywords
- ❑ LS-DYNA Model Visualization
- ❑ Model Creation and Editing
- ❑ Post-processing
 - ❖ State results animation
 - ❖ Fringe component plots
 - ❖ History XY plots

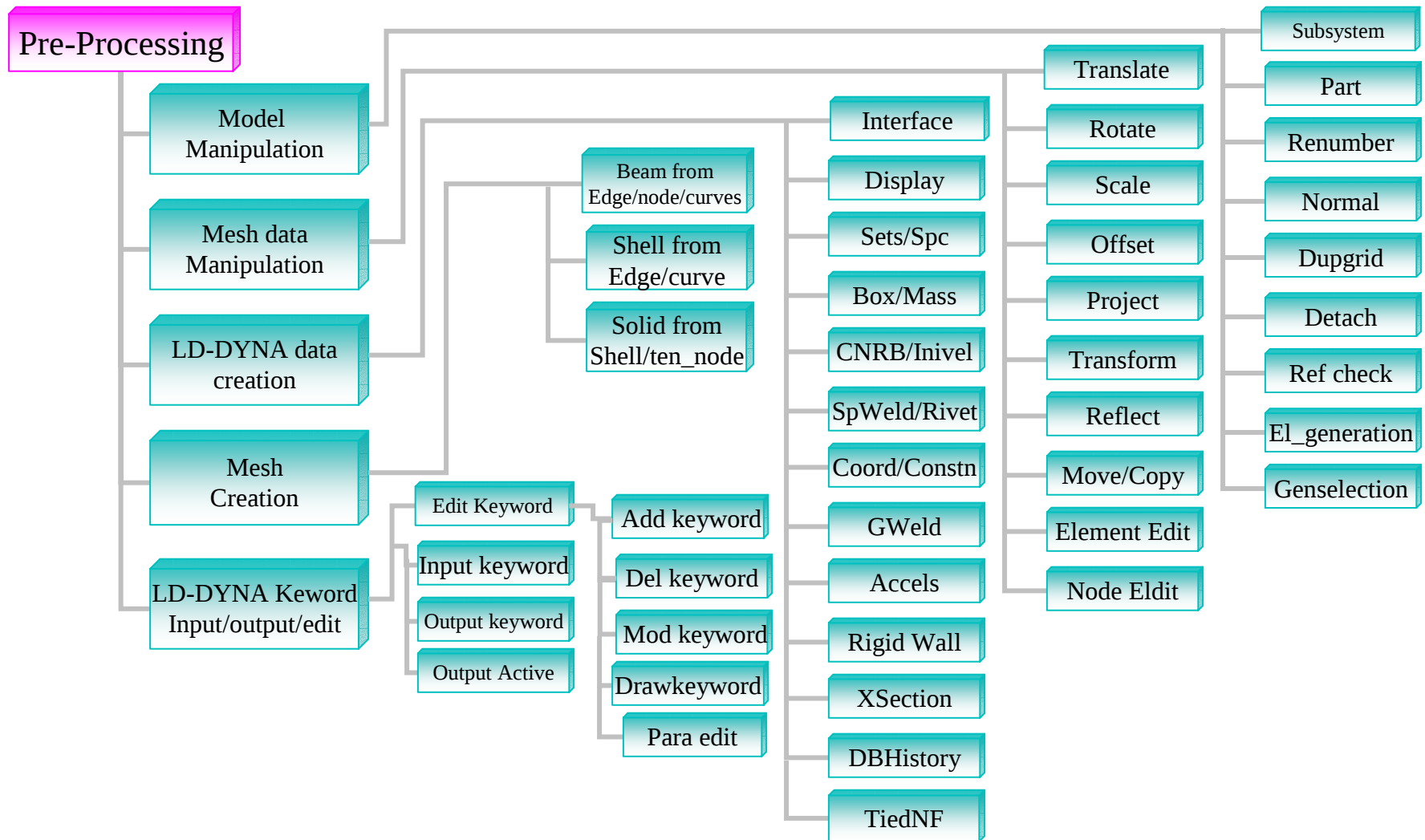
Course Outline



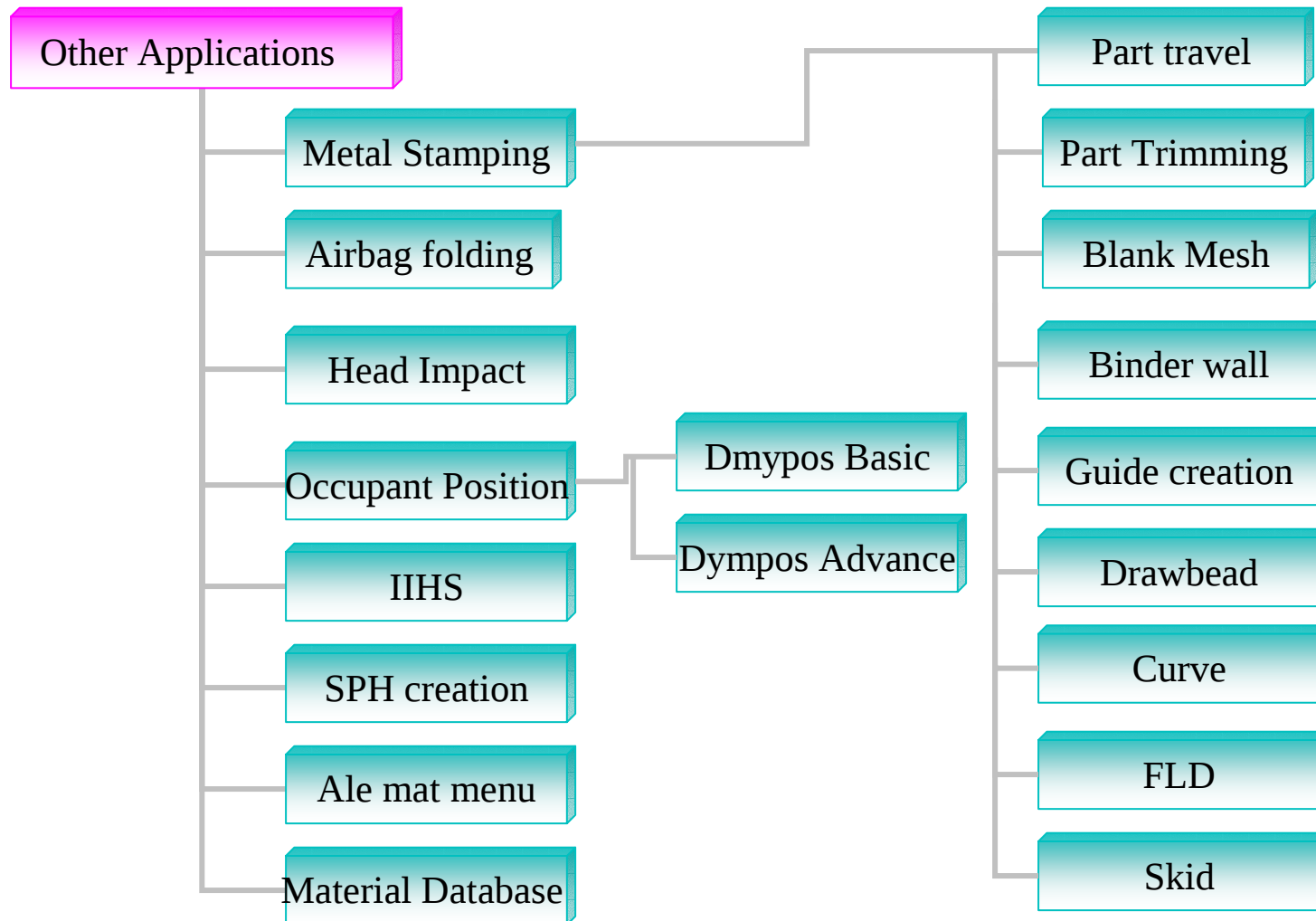
Course Outline (cont'd)



Course Outline (cont'd)



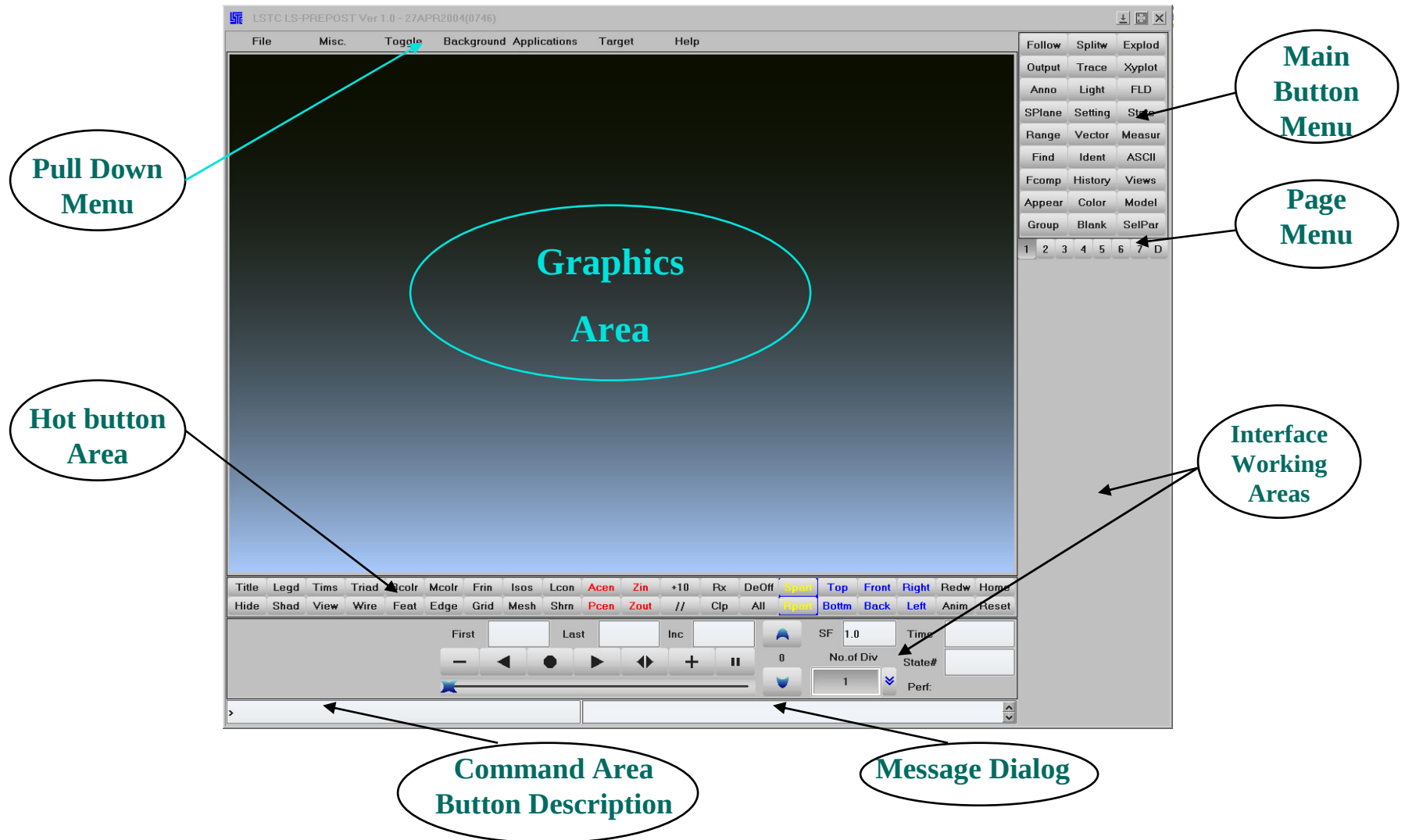
Course Outline (cont'd)



LS-PREPOST

Overview

Interface Layout



Mouse and Keyboard Operations

❑ Dynamic Model Operation

- ❖ Rotation – Left mouse button + shift key
- ❖ Translation – Middle mouse button + shift key
- ❖ Scaling – Right mouse button + shift key.
- ❖ Fast rendering mode – Use Ctrl key instead of shift key, this will render the model in edge mode.

❑ Drawing Area Selection

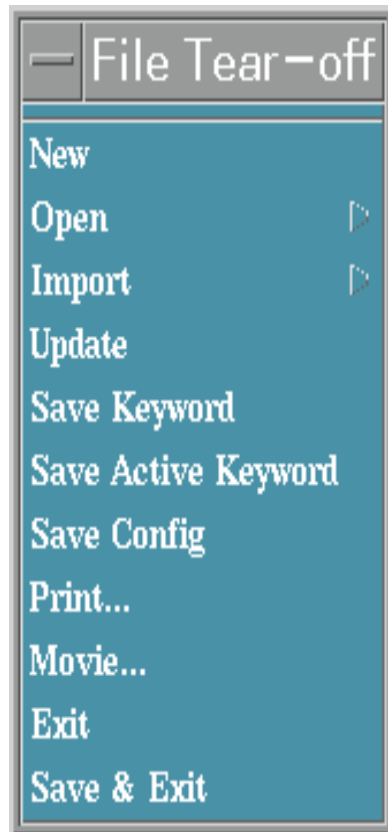
- ❖ Single Pick – Left mouse button
- ❖ Area Pick or windowing – Left mouse click, hold and drag.
- ❖ Polygon pick – Left mouse click, click point to form each side. Right click to complete.

❑ Lists and Multiple Selections

- ❖ If multiple selections are allowed then the following will apply. Press and hold Ctrl key to make multiple selections

❑ One line description of buttons – move mouse pointer to button

Pull Down Menu - File



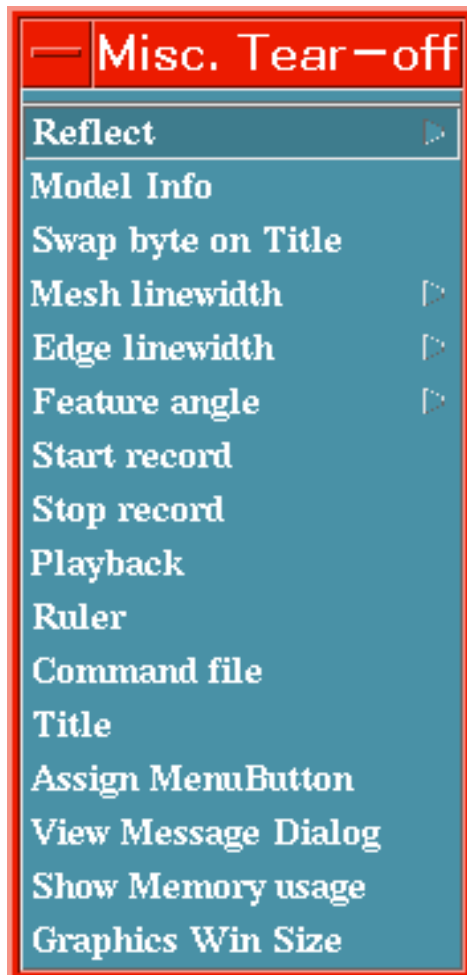
- ☐ New – Start new session, will clear out all existing models and memory
- ☐ Open – Open files, each open operation will create a new model in ls-prepost
- ☐ Import – import files will add data to current model, mainly for pre-processing operations
- ☐ Update – read up to the most recent written d3plot files
- ☐ Save Keyword – save LS-DYNA keyword file
- ☐ Save Config – save the configuration file .lspostrc
- ☐ Print – activate the printing interface
- ☐ Movie – activate the AVI file generation interface

Pull Down Menu –File->Open



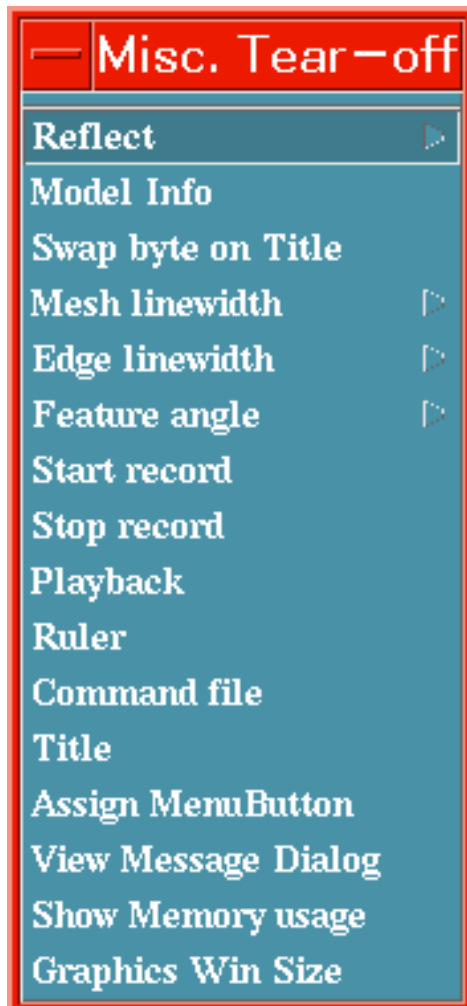
- ☐ Binary Plot – d3plot, interface file, d3thdt
- ☐ Time History – after d3plot are loaded, open d3thdt
- ☐ Command File – command session file
- ☐ Database File – lspost.db file created by cdb=input
- ☐ Crack – Crack data file
- ☐ FLD Curve – FLD curve data
- ☐ Labels – File contains labeling data
- ☐ Xydata – XY history data
- ☐ Background – Graphics window background JPEG file
- ☐ Keyword – LS-DYNA keyword input file
- ☐ Nastran File – Nastran bulk data file
- ☐ Ingrid File – Ingrid input command file
- ☐ VDA File – Geometry data in VDA format
- ☐ IGES File – Geometry data in Iges format
- ☐ LSPLIT – LS-DYNA new database format file
- ☐ Dynain Binary – Dynain file in binary format
- ☐ Ascii STL file – STL file in ascii format
- ☐ Binary STL file – STL file in binary format

Pull Down Menu – Misc



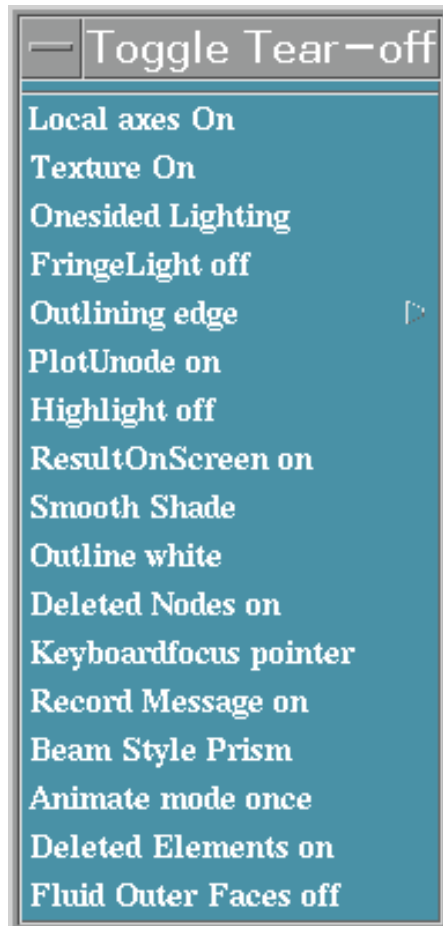
- ☐ Reflect – reflect model about global planes
- ☐ Model Info – Popup model information dialog
- ☐ Swap byte on Title – Swap the byte order for title
- ☐ Mesh linewidth – set mesh line width in pixel
- ☐ Edge linewidth – Set edge line width in pixel
- ☐ Feature angle – Set feature line computation angle
- ☐ Start record – Start recording commands for animation sequence when creating AVI file
- ☐ Stop record – Stop recording commands
- ☐ Playback – playback the recorded commands
- ☐ Ruler – Show ruler interface

Pull Down Menu – Misc



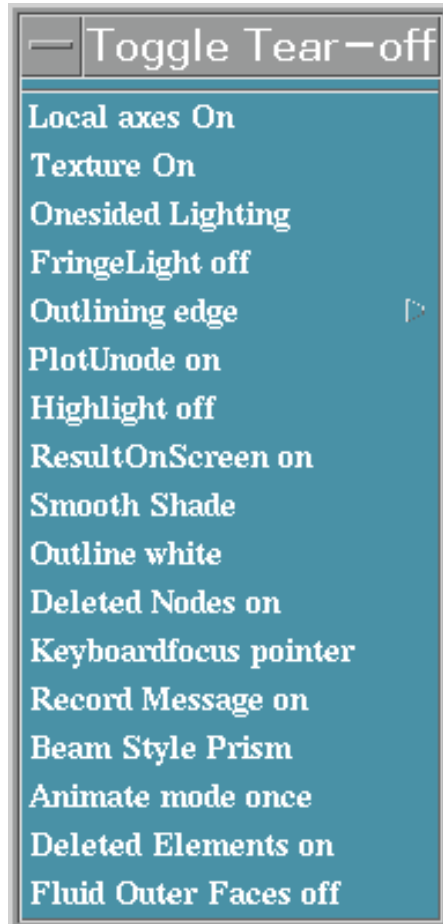
- ☐ Command file –
- ☐ Title – Change title
- ☐ Assign Menu Button – Start interface to assign menu on different buttons
- ☐ View Message Dialog – View Keyword reader message after it is closed
- ☐ Show Memory usage – Show amount of memory used for this model
- ☐ Graphics Win Size – Set the Graphics window size

Pull Down Menu – Toggle



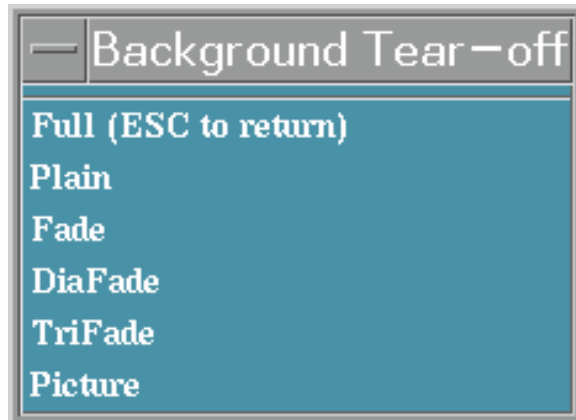
- ☐ Local axes – off/on
- ☐ Texture –switch texture mode off/on (only applicable when in Fringe mode)
- ☐ Lighting – Two-sided/One-sided
- ☐ Fringe Light – off/on When this option is off the model will be in fringe color mode. (Lighting effects are disabled)
- ☐ Outlining – Edge/feature/off, select outline type.
- ☐ PlotUnode – Toggle plot un-referenced nodes on/off
- ☐ Highlight – Toggle highlights off/on
- ☐ Result On Screen – Toggle Show results on screen on/off (only applicable if show results is selected under Identify interface)
- ☐ Shade – Flat/smooth
- ☐ Outline – Select outline color black/white

Pull Down Menu – Toggle



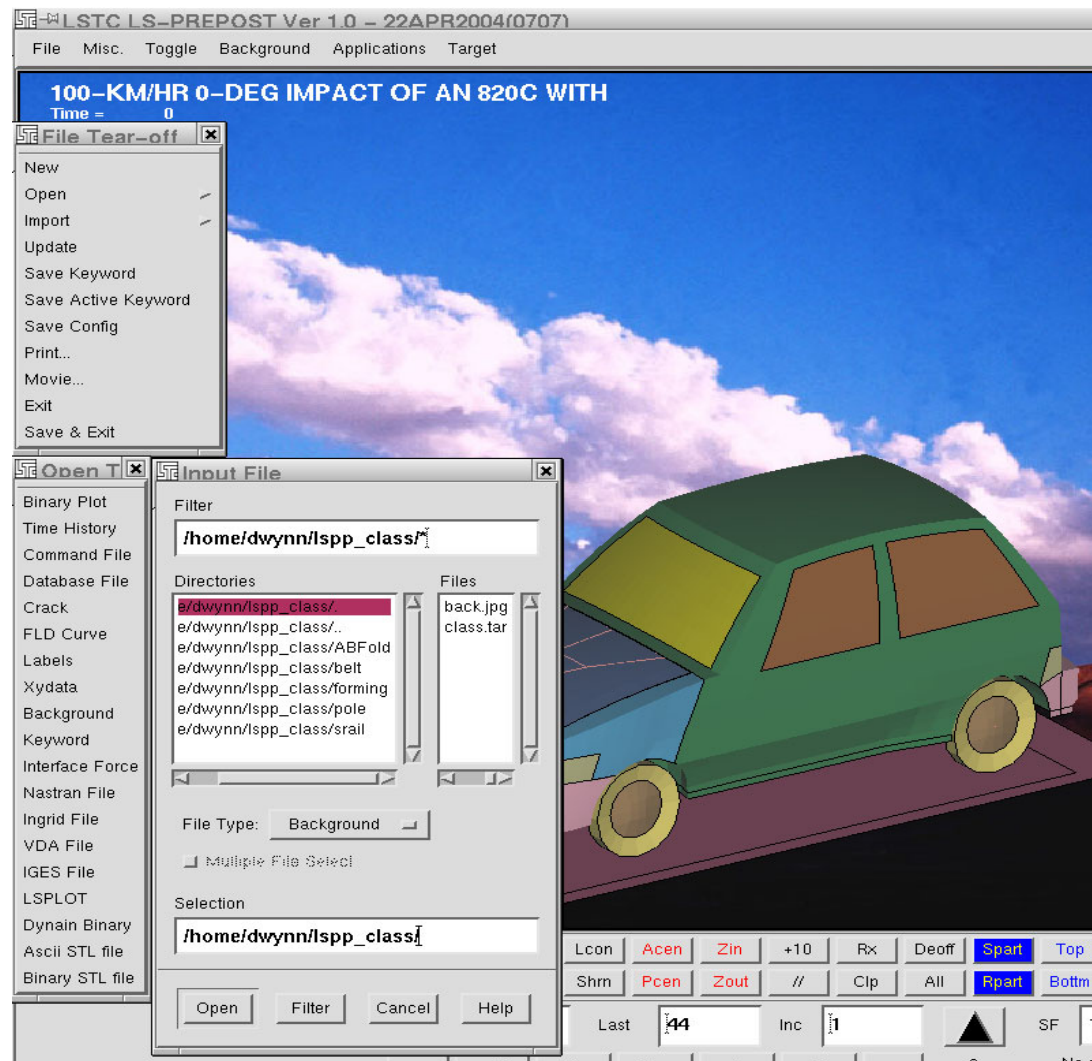
- ☐ Deleted Nodes – on/off
- ☐ Record Message – off/on When on all messages displayed in the command window will be saved to file, Default file = lspost.msg
- ☐ Beam Style – Line/Prism
- ☐ Animate mode – Loop/Once
- ☐ Deleted elements – off/on
- ☐ Fluid Outer Faces – off/on

Pull Down Menu – Background



- ☐ Full (ESC to return) - Set graphic window size to full screen
- ☐ Plain – Single color
 - ❖ Background color can be changed using Color interface on Page1
- ☐ Fade, DiaFade, TriFade
 - ❖ Faded background color can be changed using Color interface on Page1
- ☐ Picture - Jpeg file can be imported through file-background

Example



Hot Buttons

Title	Legd	Tims	Triad	Bcolr	Mcolr	Frin	Isos	Lcon	Acen	Zin	+10	Rx	Deoff	Spart	Top	Front	Right	Redw	Home
Hide	Shad	View	Wire	Feat	Edge	Grid	Mesh	Shrn	Pcen	Zout	//	Clp	All	Rpart	Bottm	Back	Left	Anim	Reset

- ❑ Hot buttons are always available, they provide a fastest access to functionalities
- ❑ Hide, Shad, View, Wire, Feat, Edge, Grid – Model drawing mode
- ❑ Mesh, Shrn – Turn mesh line and shrinking on/off
- ❑ Frin, Iso, Lcon – Set fringe/contour drawing mode
- ❑ Title, Legd, Tims, Triad – Turn on/off
- ❑ Bcolr, Mcolr – Turn background and mesh line color to white/black

Hot Buttons

Acen	Zin	+10	Rx	Deoff	Spart
Pcen	Zout	//	Clp	All	Rpart

- ☐ Acen – Automatically center model to fit within graphics window
- ☐ Pcen – Pick a node to be the new center point for model rotation
- ☐ Zin/Zout – Zoom in, Zoom out the model
- ☐ +/- 10, Rx, Ry, Rz – Rotate 10 degrees about the global X, Y, Z axes respectively. Right click on Rx, Ry, Rz to switch rotation axis.
- ☐ // Toggle between parallel view and perspective view
- ☐ Clp – Clear all highlighted nodes/element/parts, etc.
- ☐ All – Turn all blanked elements back on
- ☐ Deoff - Turn on/off all other entities in graphics window
- ☐ Spart – Activate Selpart interface
- ☐ Rpart – Restore last blanked part (blanked in Selpart interface)

Hot Buttons

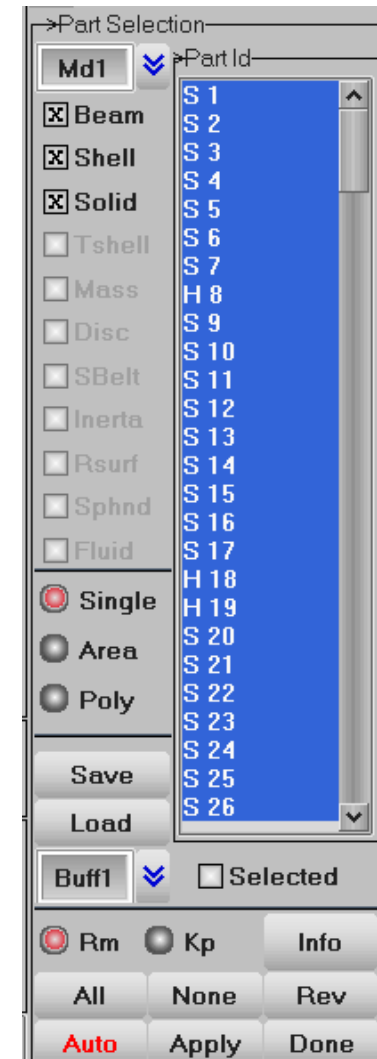
Top	Front	Right	Redw	Home
Bottm	Back	Left	Anim	Reset

- ☐ Top, Bottom, Front, Back, Right, Left – View model in the 6 global views
- ☐ Redw – Redraw the graphics window.
- ☐ Anim – Activate the animation interface and start/stop animation
- ☐ Home – re-position the model in the default orientation (Top view) and fit to screen
- ☐ Reset – Reset to default drawing mode

Part Selection

Purpose: To turn on and off parts

- ☐ List provide all parts ID with name, use extended selection
- ☐ Element type can be turn on and off
- ☐ Use single pick, area select, or polygon select
- ☐ Default is to blank part, Kp is to keep part
- ☐ Info button provide quick information on the parts that are displayed
- ☐ Current displayed parts can be saved in buffers, up to 10 buffers, load saved buffer to become current display list
- ☐ Can select different model for multiple models display



Explode Parts

Explode/separate parts for better visualization

- ☐ Factor define scale factor for parts to move
- ☐ Direction define the direction for the parts to be moved
- ☐ All/Part/PrtGroup select parts that are to be exploded/moved.
- ☐ Apply Apply the settings to explode parts
- ☐ Clear Clear the part selection
- ☐ Reset Reset the display



The dialog box is titled "Explode Parts". It contains the following controls:

- Factor:** A numeric input field with a value of 1.0 and a dropdown arrow.
- Directions:** A dropdown menu with the value "xyz" and a dropdown arrow.
- ☐ Move
- ☒ All
- ☐ Part
- ☐ PtGroup
- Part Groups list:** An empty rectangular box.
- Buttons:** A 2x2 grid of buttons labeled "Apply", "Clear", "Reset", and "Done".

Group

Purpose: Allows users to create groups of parts.

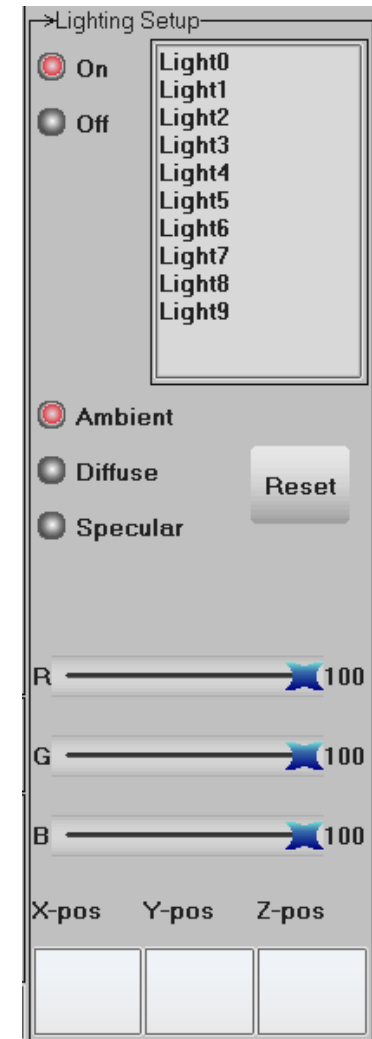
- ☐ A group is a collection of parts
- ☐ Group name can be anything up to 20 characters
- ☐ Create will put current display list to a new group
- ☐ Add – add a group to the current display list
- ☐ Subtract – subtract a group from the current display
- ☐ Save – save group names in ascii file
- ☐ Load – read previously saved groups
- ☐ Load and save very efficient for presentations
- ☐ Or, And, Xor operations are used for Adding group
- ☐ Auto Center will automatically center the selected group



Lighting Setup

Purpose: To setup lighting for the model

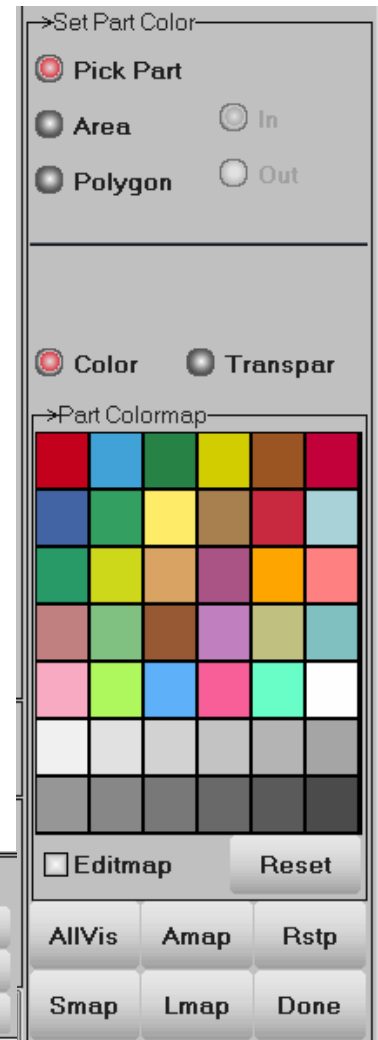
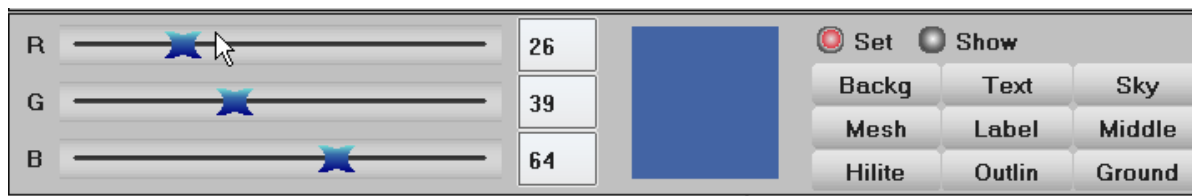
- ☐ Two standard lights are set as default –
 - ❖ Front – position (0.0, 0.0, 1.0)
 - ❖ Back – position (0.0, 0.0, -1.0)
- ☐ User can turn on up to 9 lights
- ☐ Set red, green, blue values (0 to 1) of the components
 - ❖ Ambient
 - ❖ Diffuse
 - ❖ Specular
- ☐ Set position of the light
- ☐ Reset to default values



Color and Transparency

Allows users to change

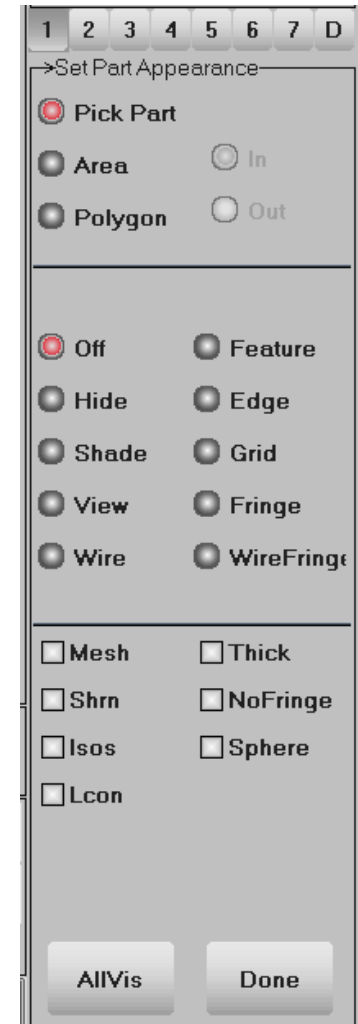
- ☐ Part color and transparency
- ☐ Background color
- ☐ Select color from color palate
- ☐ Make color
- ☐ Text color
- ☐ Highlight color
- ☐ Modify sky,middle,ground colors for shaded background



Appearance

Purpose: To set the appearance of each part

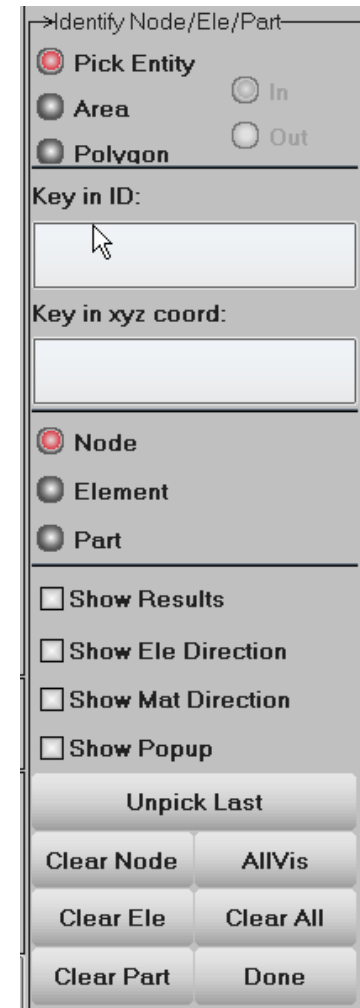
- ☐ Single pick, area select, or polygon select
- ☐ Set each part or group of part to have the selected rendering options
- ☐ Thick – button to set drawing shell as solid
- ☐ Sphere – display solid part nodes as sphere – for EFG analysis
- ☐ Allvis – allows user to set all visible parts the the selected setting
- ☐ Hot button rendering option turn off while in Appear interface



Identity Nodes/Elements/Parts

Purpose: To identify nodes/elements/parts

- ☐ Use single pick, area selection or polygon selection
- ☐ Can key in the Ids,
- ☐ Or key in xyz values to show position
- ☐ Show node/element result
- ☐ Show element direction (shell and beam)
- ☐ Show orthotropic material direction
- ☐ Show all visibles
- ☐ Unpick last picked entity
- ☐ There is a maximum Ids to be shown (10000)



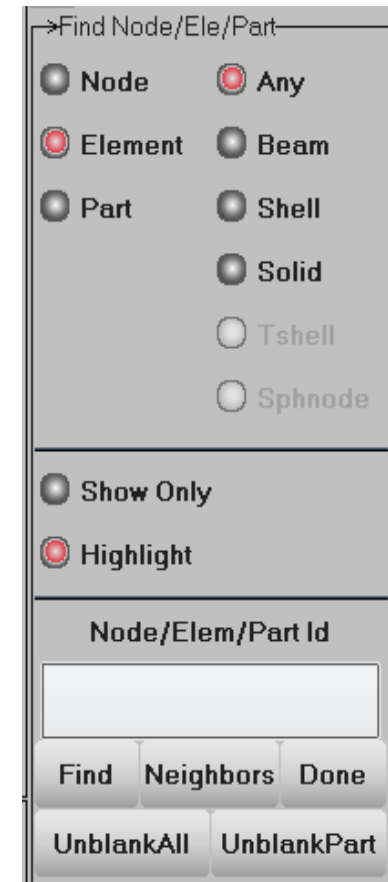
The dialog box is titled "Identify Node/Element/Part". It contains the following controls:

- Pick Entity:** A radio button is selected. To its right are two radio buttons: "In" and "Out".
- Area:** A radio button.
- Polygon:** A radio button.
- Key in ID:** A text input field with a mouse cursor pointing to it.
- Key in xyz coord:** A text input field.
- Entity Type:** Three radio buttons: "Node" (selected), "Element", and "Part".
- Display Options:** Four checkboxes: "Show Results", "Show Ele Direction", "Show Mat Direction", and "Show Popup".
- Buttons:** A row of buttons: "Unpick Last", "Clear Node", "AllVis", "Clear Ele", "Clear All", "Clear Part", and "Done".

Find Nodes/Elements/Parts

Purpose: To find a node/element/parts

- ☐ Specify node, element or part
- ☐ Find any element with ID, or specified type
- ☐ Show only will show the found entity by itself
- ☐ Highlight will highlight the found entity
- ☐ Neighbors will propagate the find to neighboring elements
- ☐ Unblank part will turn on all elements in the part that the found node/element belongs
- ☐ Unblankall will turn on all elements and all parts



The dialog box is titled "Find Node/Element/Part". It contains several sections:

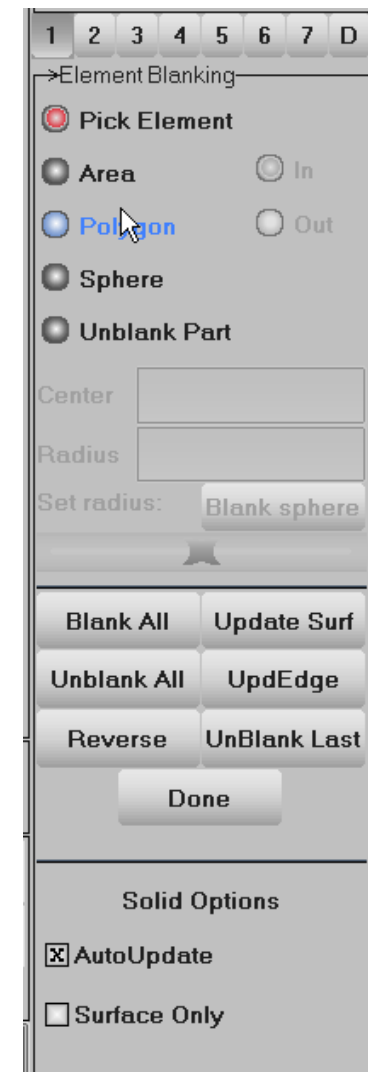
- Search Criteria:** A grid of radio buttons for selecting the search target.

☐ Node	☒ Any
☒ Element	☐ Beam
☐ Part	☐ Shell
	☐ Solid
	☐ Tshell
	☐ Sphnode
- Display Options:** Two radio buttons: ☐ Show Only and ☒ Highlight.
- Search ID:** A text input field labeled "Node/Elem/Part Id".
- Action Buttons:** A row of three buttons: "Find", "Neighbors", and "Done".
- Blanking Buttons:** A row of two buttons: "UnblankAll" and "UnblankPart".

Blank Elements

Purpose: To blank element from display

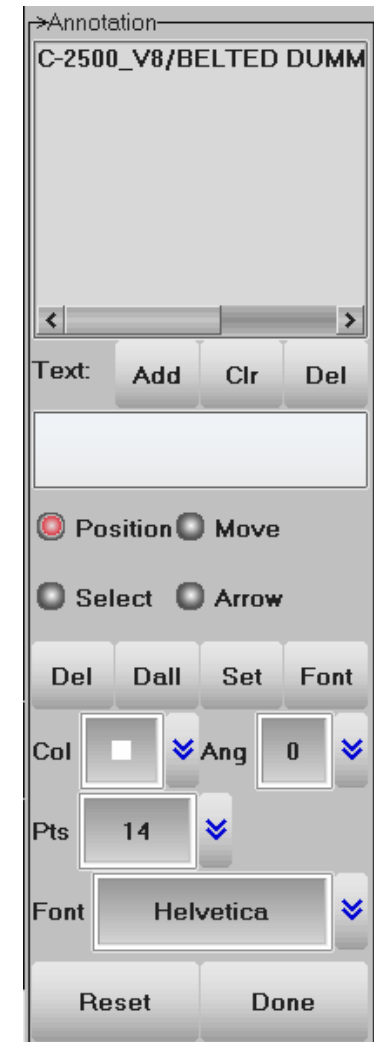
- ☐ Single pick, area, polygon selection,
- ☐ Spherical – allows user to pick a point and set a radius
- ☐ Unblank part – allows user to pick a part and unblank all elements in that part
- ☐ Update Surf – Update solid elements surface list
- ☐ UpdEdge – Update shell element edge list
- ☐ Unblank last – Undo the last blank operation
- ☐ Solid Options
 - ❖ AutoUpdate – will update solid surface list automatically
 - ❖ Surface Only – only blank the top layer of solid elements

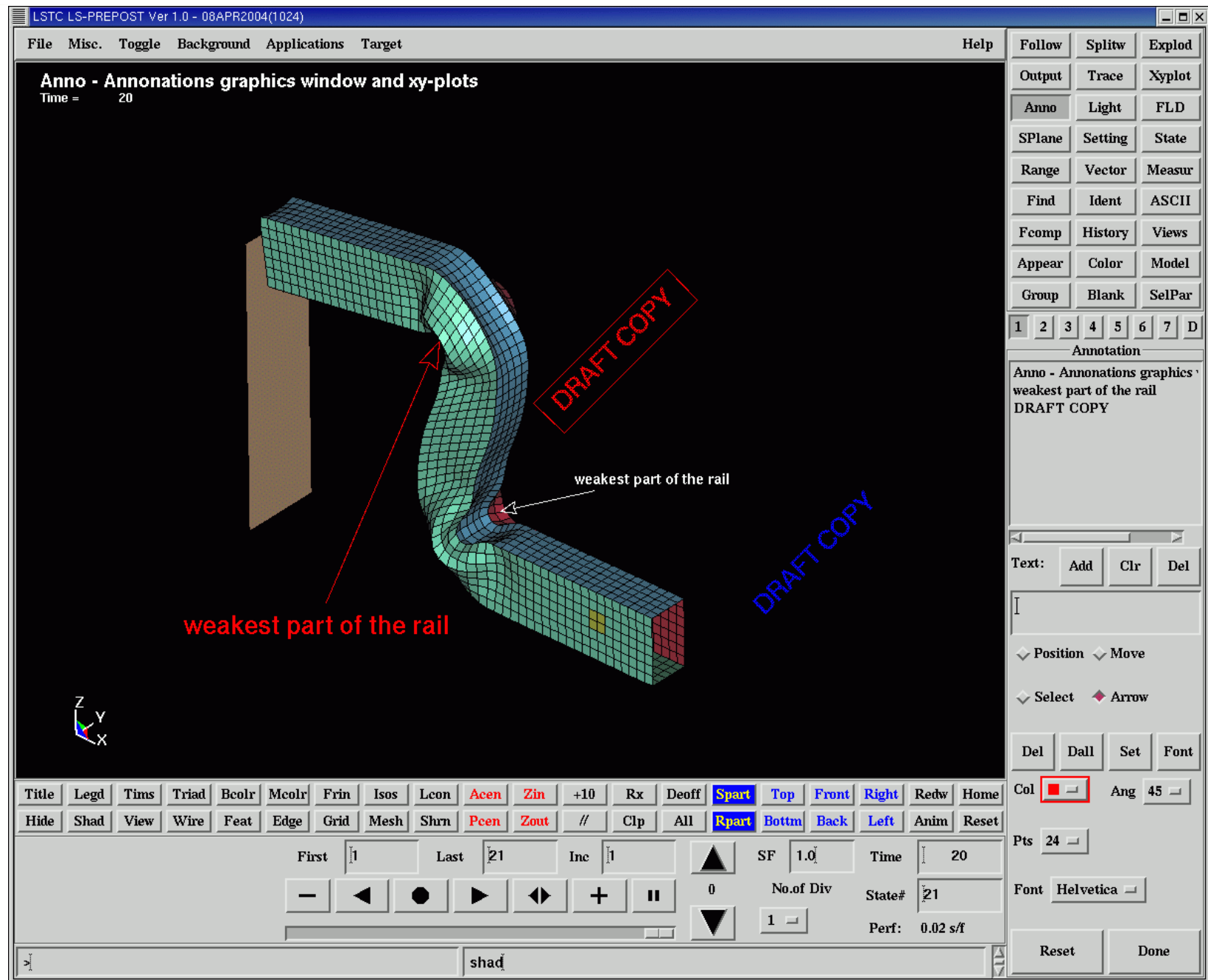


Annotate

Purpose: Annotate graphics
window and xy-plot windows

- ☐ Interactive position
- ☐ Adding arrows
- ☐ Outlining annotations
- ☐ Color choices of colors
- ☐ Moving annotations
- ☐ Font size and color
- ☐ Font style
- ☐ Angle of annotations

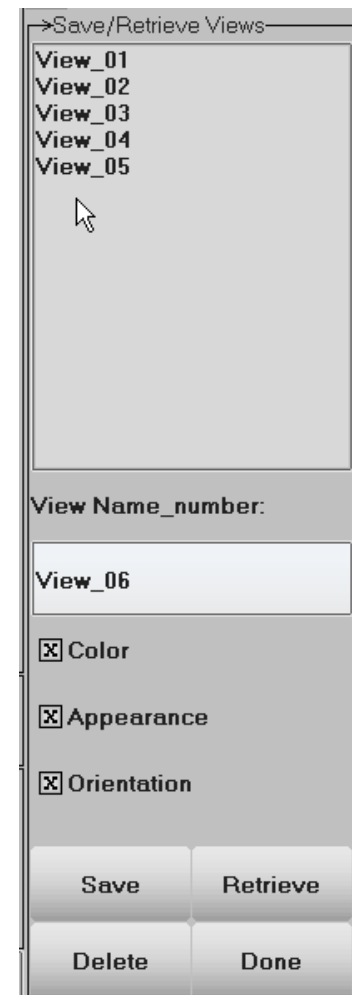




Views Interface

View allows user to save and retrieve a special setting of a model in

- ☐ Orientation
- ☐ Color
- ☐ Appearance
- ☐ The active parts and elements is also stored in the view



Save/Retrieve Views

View_01
View_02
View_03
View_04
View_05

View Name_number:

View_06

☒ Color

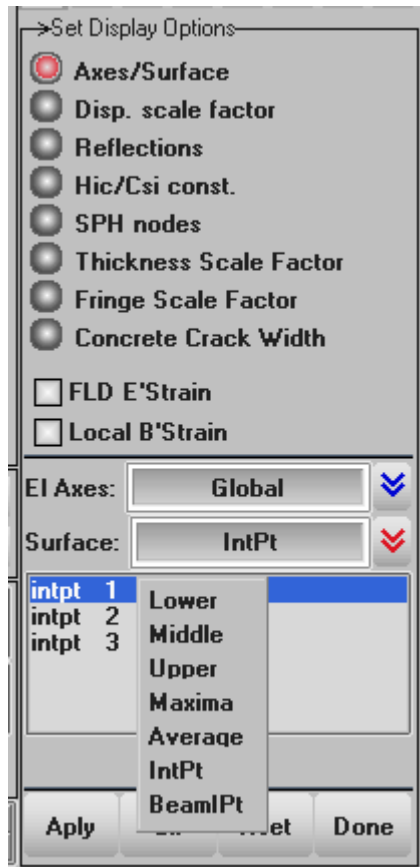
☒ Appearance

☒ Orientation

Save Retrieve

Delete Done

Settings Menu



- ❑ Menu is for setting model display parameters and items that don't fit intuitively in other menus
 - El Axes - Global/Local is for setting the calculation of component stresses and strains from the default global axes values to element axes values
 - Surface – set the current shell integration point surface, and the beam integration point. This changes what data is extracted for fringe displays: Fcomp menu. It also affects the History and Vector menus
 - Maxima and Average – enable the max value over all shell integration points, and the mean value over all points.
- ❑ Setting will only apply after the Apply button is pressed

Settings Menu

☒ Whole ☐ Part

Reflect about:

☐ XY ☐ YZ ☐ XZ

Aply Clr Rset Done

Time units:

1 sec

Gravity constant:

9.81 9.81

Aply Clr Rset Done

SPH Radius Scale:

0.3 0.3

SPH Sphere divs:

12 12

Style: flat

☐ Fixed Radius

Aply Clr Rset Done

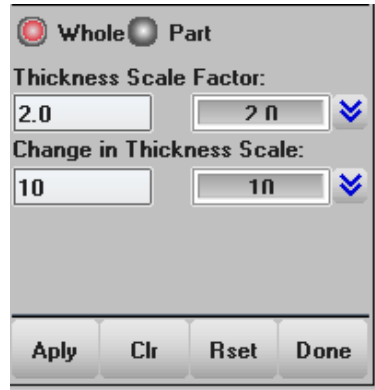
☐ Reflect menu is for making the display produce a reflection of the view about the global planes.

- o The reflections can be for the whole model or selected parts

☐ Hic/Csi menu for changing the default time scale and gravity constant

☐ SPH – Smooth Particle Hydrodynamics menu allows the appearance of the particles to be changed

Settings Menu

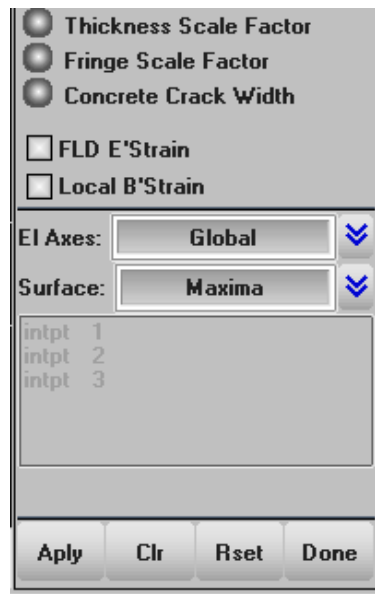


Whole ☒ Part ☐

Thickness Scale Factor:
2.0 2 n

Change in Thickness Scale:
10 1n

Aply Clr Rset Done



☐ Thickness Scale Factor
☐ Fringe Scale Factor
☐ Concrete Crack Width

☐ FLD E'Strain
☐ Local B'Strain

El Axes: Global

Surface: Maxima

intpt 1
intpt 2
intpt 3

Aply Clr Rset Done

- ☐ Thickness scale menu is related to displaying thin shells, 4 node quads, as solids. This state is set in the Appear menu
 - o Even displayed as solid may not have sufficient thickness to be prominent, so scaling the thickness helps
 - o Scaling change of thickness again allow these changes to be seen.
- ☐ Fringe Scale – allows small or large changes in fringe values to modified, this is active for all fringes after setting
- ☐ Crack Width – is associated with the D3CRCK file, sets minimum crack width in mm to be displayed – MAT 84 only
- ☐ FLD E'strain - changes the display of strains for FLD fringe components and FLD plot to Engineering or True values
- ☐ Local B'strain – Converts global strains for solid elements into element axes values

LS-PREPOST

POST-PROCESSING

Animation

Purpose: Control Animation state

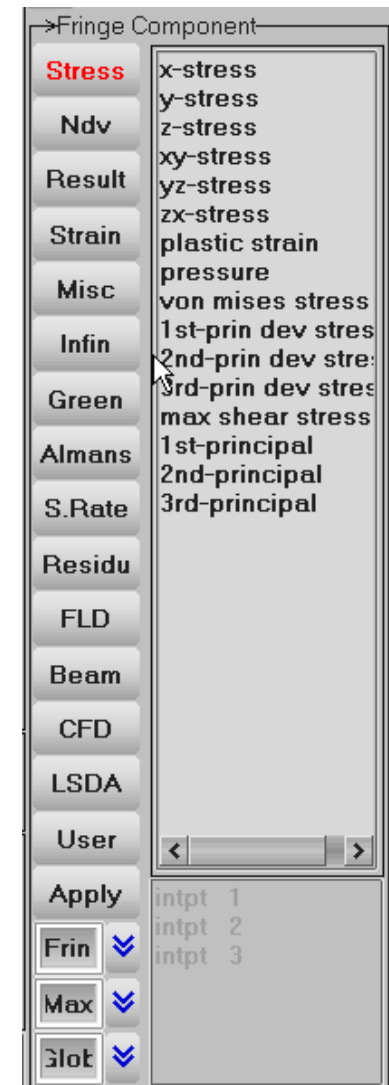
- ☐ First, Last, Inc – Animation state control
- ☐ SF – scale factor for Eigenvector (mode shape) animation
- ☐ No. of Div – set no. of interpolated steps within each mode
- ☐ Time, State#, Perf: information on the time value, state number and performance
- ☐ Up and down buttons to control animation speed
- ☐ Slider bar to position to state
- ☐ Left/right arrow to control slider bar behavior



Fcomp – Fringe Component Selection

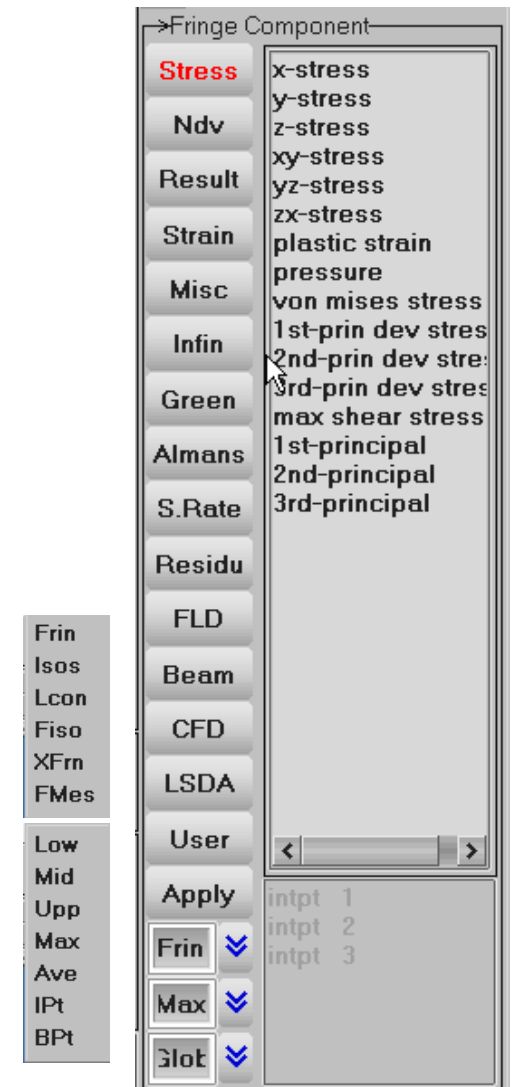
Purpose: To select fringe components

- ☐ Stress – regular stress components
- ☐ Ndv – nodal displacement and velocity
- ☐ Result – stress resultant components
- ☐ Strain – regular strain components
- ☐ Misc – miscellaneous components like pressure, temperature, internal energy, shell thickness, etc.
- ☐ Infin – infinitesimal strains
- ☐ Green – Green-St. Venant strains
- ☐ Almans – Almansi strains
- ☐ S.Rate – strain rates



Fcomp – Fringe Component Selection

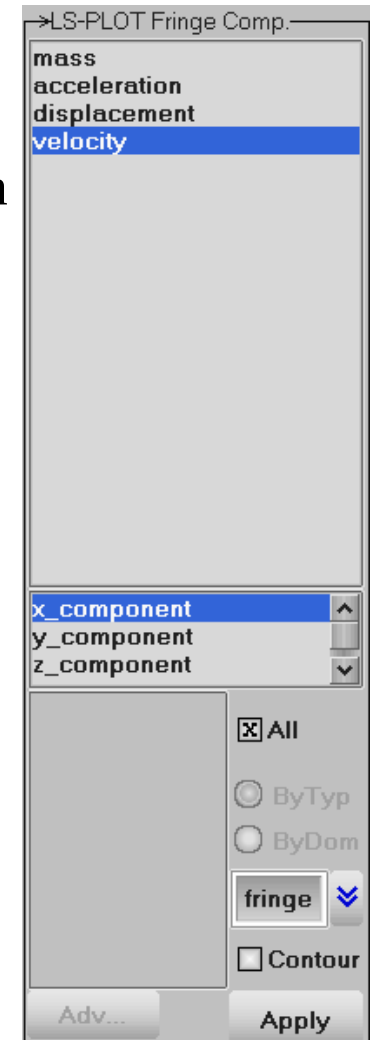
- ☐ Residu – Residual elastic strains
- ☐ FLD – Forming Limit Diagram strain components
- ☐ Beam – Beam element forces and stress resultants
- ☐ CFD – Navier-Stokes fluid components
- ☐ LSDA – New LSDYNA database fringe component
- ☐ User – user input element/node fringe component
- ☐ Apply – Not used
- ☐ Frin – choose different type of rendering
 - ❖ Isos – iso-surfaces for solid, Lcon – line contours
 - ❖ XFrn – Fringe max value across all surfaces
 - ❖ Fmes – Fringe color on the mesh
- ☐ Low, Mid, Upp, Max, Ave – location of shell surfaces
- ☐ Ipt – integration points
- ☐ Bpt – Beam element integration points
- ☐ Glob – Global or local



Fcomp - LSPLLOT

•Processing new database from LS-DYNA

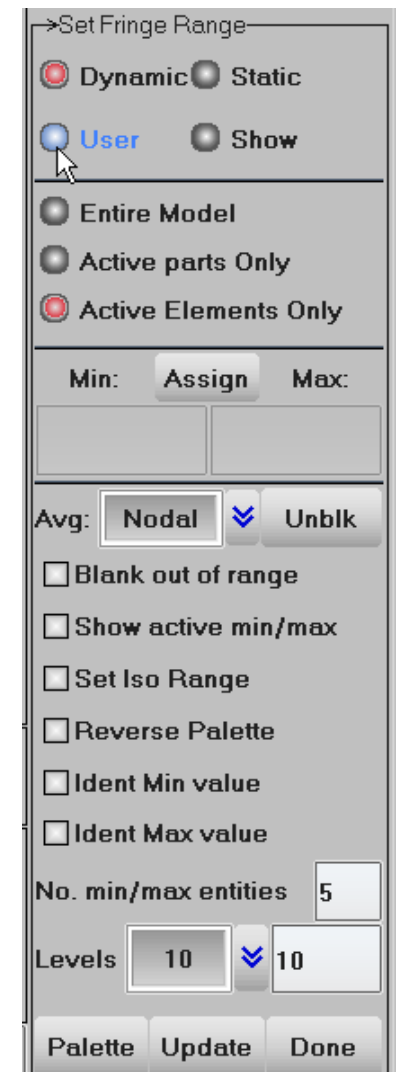
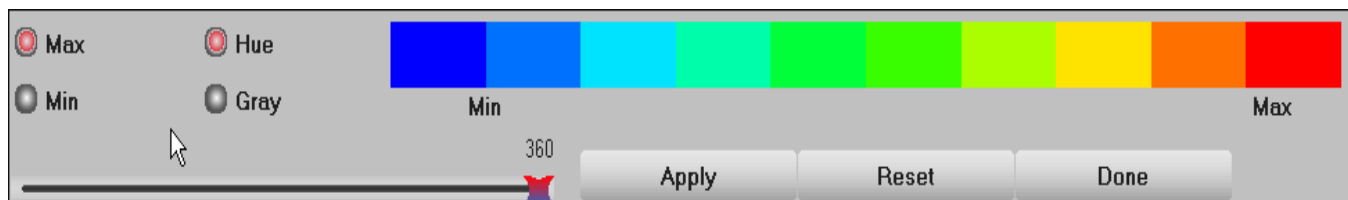
- ☐ LSPREPOST automatically detects the loaded database and switches to the same fringing menu between D3PLOT and LSPLLOT.
- ☐ Even though LSPLLOT is still at its early stage, yet LSPREPOST will always give you the up-to-date support for your post-processing needs when using LS-DYNA.
- ☐ One of the significant features in the new database is flexibility. You can even define your own output, where you want it and even how you would like it to be computed.



Range

Range allows user set fringe and iso-surface ranges

- ☐ Dynamic - min/max is different for each time state
- ☐ Static sets the same min/max for all the states
- ☐ User specifies a defined min/max for all the states
- ☐ Show – Shows elements within the specified range
- ☐ Choose entire model, active parts only or active elements only
- ☐ Color Palette can also be changed



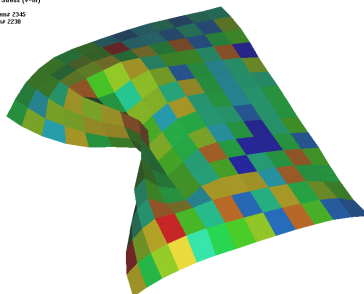
Range

- ☐ Elements out of range can be blanked out
- ☐ The color palette can be reversed with red showing min and blue max
- ☐ Min/Max elements can be identified along with a user defined number of elements to be shown
- ☐ Number of fringe levels can be set to 10, 20 and 30
- ☐ Average can be set to none, nodal and min/max

Min:	Assign	Max:
None		
Avg:	Nodal	Unblk
MinMax		
☐ Blank out of range		
☒ Show active min/max		
☐ Set Iso Range		
☒ Reverse Palette		
☐ Ident Min value		
☐ Ident Max value		
No. min/max entities		5
Levels 10		10
Palette	Update	Done

No Average

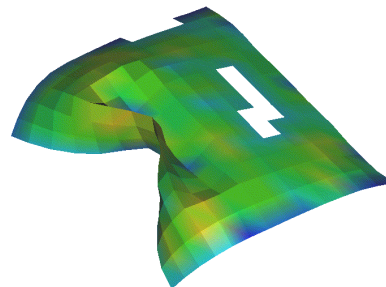
100-KM/HR 0-DEG IMPACT OF AN 820C WITH
Time = 120
Contours of Effective Stress (v-m)
min=0.000000, at elem# 2345
max=0.190000, at elem# 2238



Fringe Levels
1.801e-01
1.621e-01
1.441e-01
1.261e-01
1.081e-01
9.01e-02
7.21e-02
5.41e-02
3.60e-02

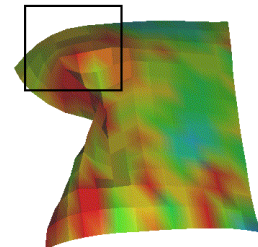
**Out of range
Elements blanked**

Time = 120
Contours of Effective Stress (v-m)
max=0.000000, at elem# 2345
min=0.190000, at elem# 2238



Fringe Levels
1.801e-01
1.621e-01
1.441e-01
1.261e-01
1.081e-01
9.01e-02
7.21e-02
5.41e-02
3.60e-02

Time = 120
Contours of Effective Stress (v-m)
max=0.000000, at elem# 2345
min=0.190000, at elem# 2238
active min=0.000000, at elem# 2225
active max=0.190000, at elem# 2238



**Min/max in selected
area shown**

Measure

Purpose: To make measurements of various items

- ☐ Items - Selection list for measuring or creating a local coordinate system
- ☐ Cancel pick – to remove last picked operation
- ☐ History list – depends on selection list
- ☐ Active Elements Only – for certain quantities (like area, volume, mass, etc.)
- ☐ Element, Part, All – measuring by element or by part, or all
- ☐ Reference Axes – select current reference axes for measurement



Measure

Purpose: To make measurements of various items

❑ Items to be measured

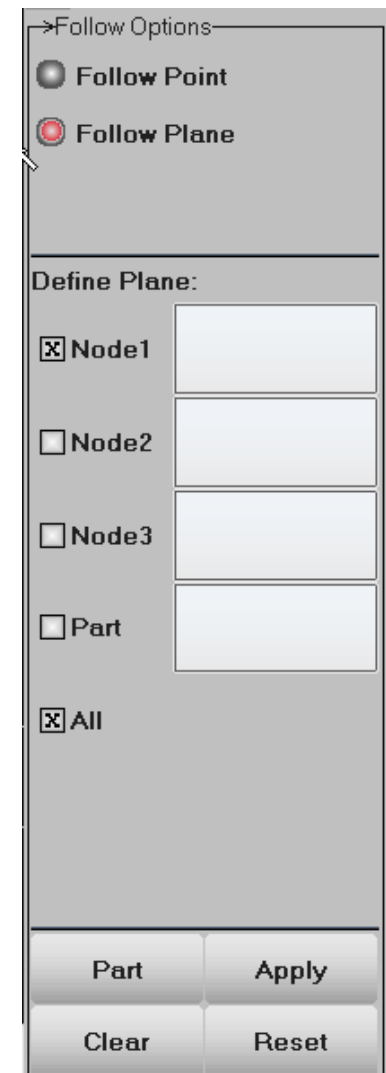
- ❖ Coordinate
- ❖ Distance – between 2 nodes
- ❖ Dist N2S – between a node and a surface
- ❖ Angle 3Node – angle between 3 nodes
- ❖ Angle 4node – angle between 2 lines formed by 4 nodes
- ❖ 3pt Radius – the radius formed by 3 points (nodes)
- ❖ Area, volume, Mass, Inertia
- ❖ AngVel – angular velocity
- ❖ Separation – distance between two parts



Follow

This interface is used for model animation and post processing

- ❑ The Definition of a point or a plane makes certain that the model is fixed at the center of the screen during the animation
- ❑ After fixing a point or a plane a fringe plot of the relative displacement of the model can be carried out
 - ❖ Select the method
 - ❖ Chose the node or nodes and Apply to follow the model view



Follow Options

☐ Follow Point

☒ Follow Plane

Define Plane:

☒ Node1

☐ Node2

☐ Node3

☐ Part

☒ All

Part Apply

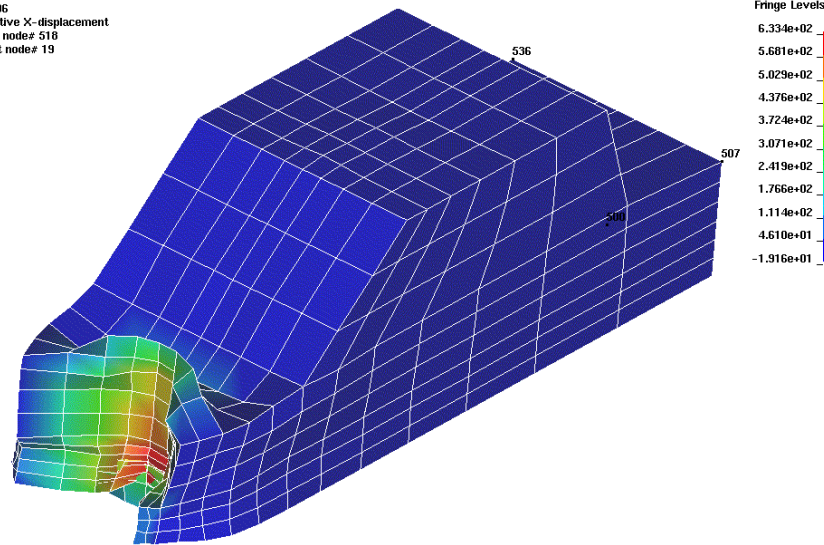
Clear Reset

Follow

Relative displacement contour plot

- ☐ Apply the follow point or plane
- ☐ Select Ndv in Fcomp
- ☐ Select Relative Disp (x, y, z or resultant displacement)

Time = 0.050006
Contours of Relative X-displacement
min=-19.155, at node# 518
max=633.372, at node# 19



>Fringe Component

Stress	x-displacement
Ndv	y-displacement
Result	z-displacement
Strain	result displace
Misc	xy-displacement
Infin	yz-displacement
Green	xz-displacement
Almans	x-velocity
S.Rate	y-velocity
Residu	z-velocity
FLD	result velocity
Beam	rx-displacement
CFD	ry-displacement
LSDA	rz-displacement
User	r-result disp
Apply	
Frin	intpt 1
Max	intpt 2
Plot	intpt 3

Relative displacement
fringes

State Control

- Direct access to a single state

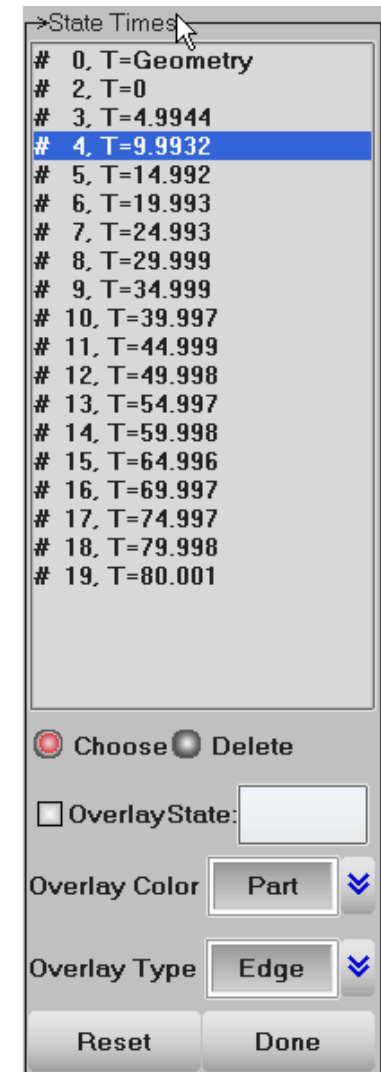
- ☐ User may directly select a state in the state list, LSPREPOST will bring the state of the model requested.

- Turn off un-wanted states

- ☐ Sometimes you may want to turn off some states such that you can focus on those you interested in.

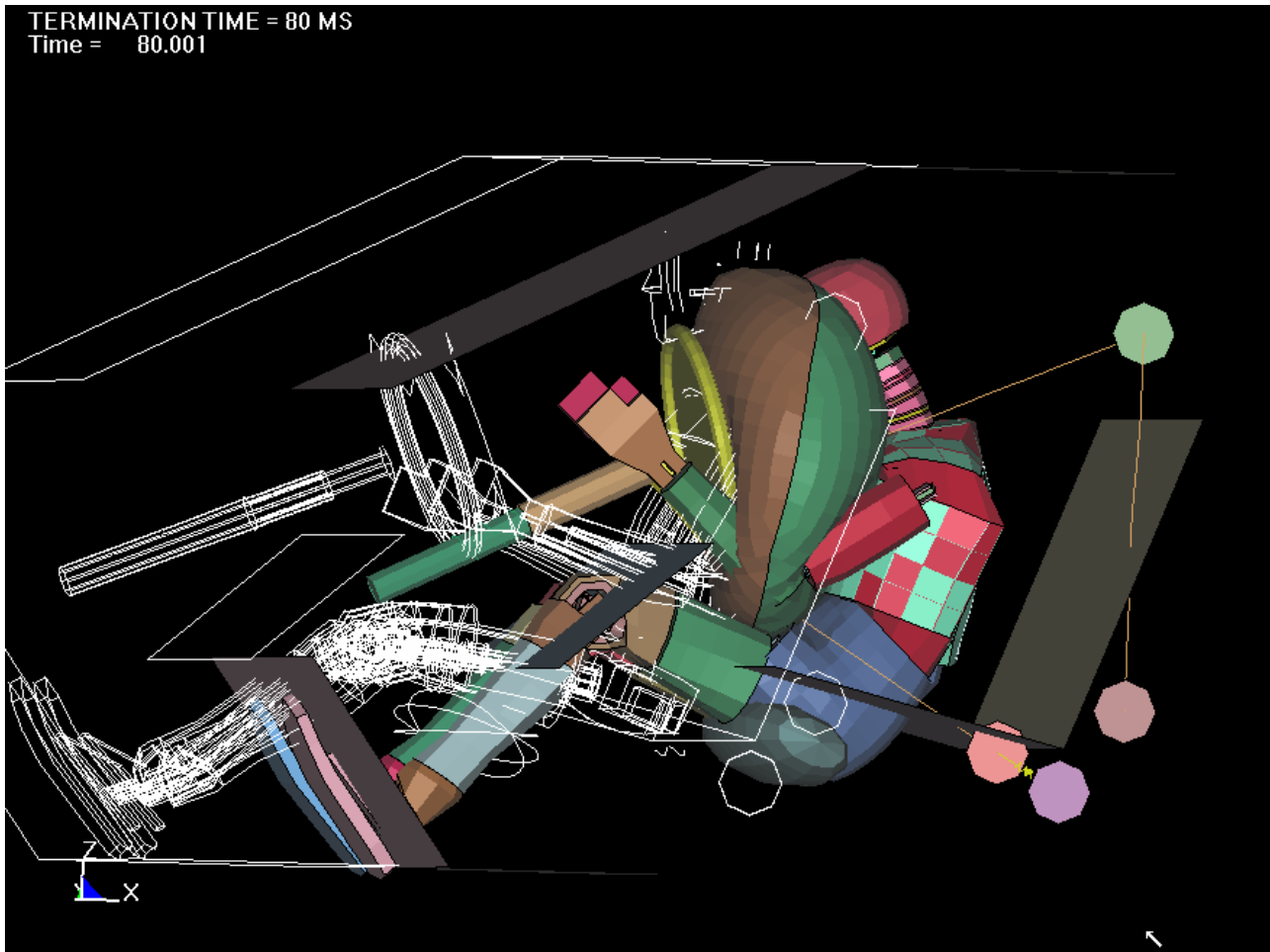
- Overlay different states in various ways

- ☐ By assigning the state to be overlaid, and how you want it to be overlaid, you can easily compare different simulation snapshots of the current model.



State Control with Overlay

TERMINATION TIME = 80 MS
Time = 80.001



→State Times←

0, T=Geometry
2, T=0
3, T=4.9944
4, T=9.9932
5, T=14.992
6, T=19.993
7, T=24.993
8, T=29.999
9, T=34.999
10, T=39.997
11, T=44.999
12, T=49.998
13, T=54.997
14, T=59.998
15, T=64.996
16, T=69.997
17, T=74.997
18, T=79.998
19, T=80.001

☒ Choose ☐ Delete

☒ OverlayState: 1

Overlay Color White ▾

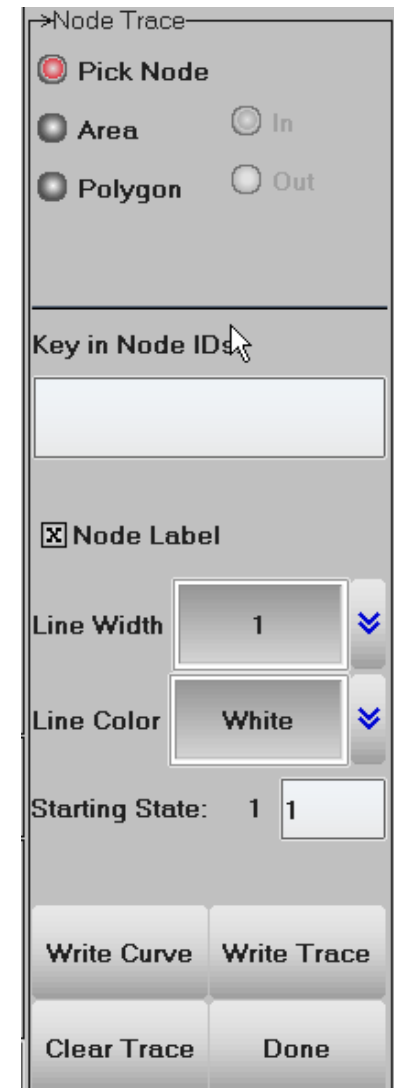
Overlay Type Feat ▾

Reset
Done

Trace Node plot

Purpose: Plot trace of nodes

- ☐ Single pick, area select, or polygon select, or key in node Ids
- ☐ Set trace plot line width and color
- ☐ Can start from any state other than state 1
- ☐ Write Trace – Time, State and X,Y,Z position
- ☐ Write Cruve – Time and displacement



Node Trace

☒ Pick Node

☐ Area ☐ In

☐ Polygon ☐ Out

Key in Node IDs

☒ Node Label

Line Width: 1

Line Color: White

Starting State: 1 1

Write Curve Write Trace

Clear Trace Done

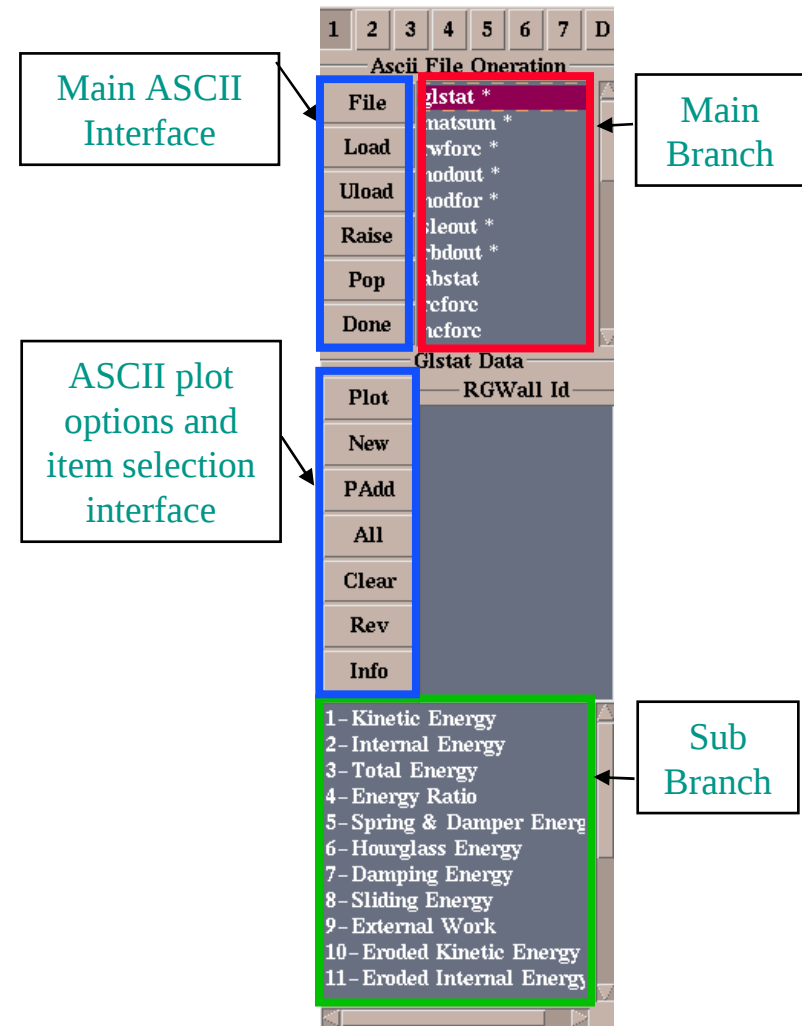
ASCII files

XY-plotting of various LS-DYNA ASCII files

Support for loading multiple ASCII files from different runs

☐ Main ASCII Interface

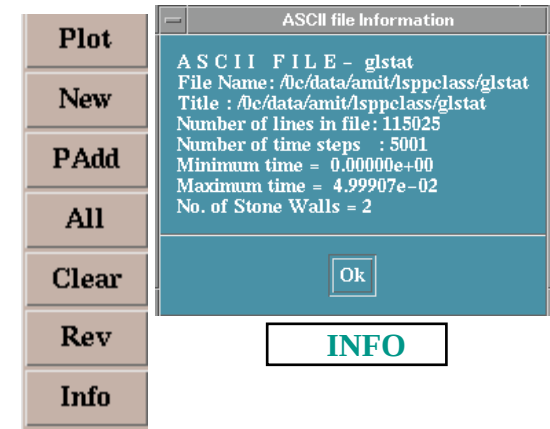
- ❖ FILE - Load Ascii file from alternate directory
- ❖ LOAD and UNLOAD – Load existing file for selected file type and unload to free memory
- ❖ RAISE – Raise all XY-Plot windows
- ❖ POP – Open and Raise all closed XY plot windows
- ❖ DONE – Exit ASCII interface



ASCII files

□ ASCII XY Plot Interface

- ❖ PLOT – Plot items from ASCII file in current XY plot window
- ❖ NEW – Plot Items from ASCII file in new XY plot window
- ❖ PADD – Add items from ASCII file to current XY plot window
- ❖ ALL – Select all items in the list
- ❖ CLEAR – Clear the selected items
- ❖ REV – Reverse the selection
- ❖ INFO – Show information on the loaded ASCII file



□ Multiple ASCII files can be loaded from alternate subdirectories

- ❖ Select ASCII file
- ❖ Load ASCII file from alternate directory
- ❖ The full path of the chosen ASCII file will be shown

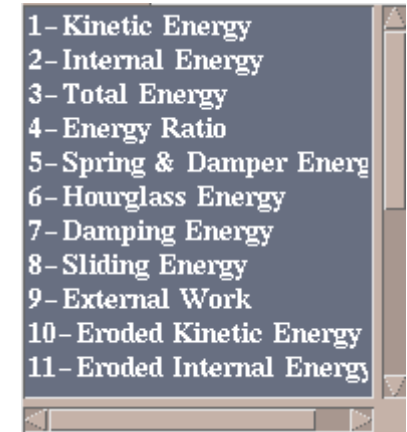


Multiple ASCII file loading with full path of file shown

ASCII files

❑ ASCII Interface SUB Branch

- ❖ The sub branch lists the data that is available in the chosen ASCII file
- ❖ Some of the ASCII files have special Toggles that can be used to process data
 - MATSUM, SECFORC, SPCFORC, NODFOR, BNDOUT, SLEOUT, and GCEOUT – [Total] used to combine multiple ASCII item selection
 - ABSTAT- [Airbag/Mat ID] Select airbag/material ID
 - ELOUT – [Clrcp] Clear component list, [Stress, Strain & Beams] component types
 - DEFORC – [Trans/Rotat] Translational force and rotational moment
 - NODFOR – [Group] combines the nodal forces of the group
 - NODOUT – [HicCsi] Open Hic/Csi
 - RBDOUT – [Local] Plot curve in local coordinate sys
 - SPCFORC – [Force/Moment] forces and moment plotting



**Component list of
GLSTAT file**

Section Plane

Purpose: To Perform section cuts in the model to study interactions/section forces/crush.

- ☐ FixSp – Section plane is fixed in space
- ☐ FixMd – Section plane is fixed to the model
- ☐ Plane definition
 - ❖ 1p+NL – base point + normal direction
 - ❖ N1-N2 - Node 1 defines the base point, vector from node1 to node2 define the normal
 - ❖ 3Nds – 3 nodes define a plane
 - ❖ 2Nds + D – define plane by 2 nodes and a direction
- ☐ Base point can be defined as position or a node
- ☐ Directions defined by:

NormX	NormY	NormZ
0.0	1.0	0.0

→Section Plane

☒ FixSp ☐ FixMd

☒ 1p+NL ☐ N1-N2

☐ 3Nds ☐ 2Nds+D

☒ BasePt ☐ BaseNd

X: 2487.46

Y: 0.00012207

Z: 999.5

Node

NormX	NormY	NormZ
0.0	1.0	0.0

Centroid Reset

☐ Keep all cuts Clear

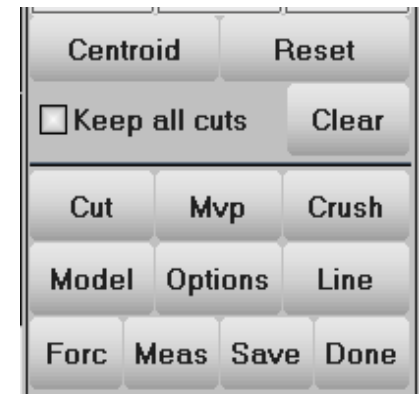
Cut Mvp Crush

Model Options Line

Forc Meas Save Done

Section Plane

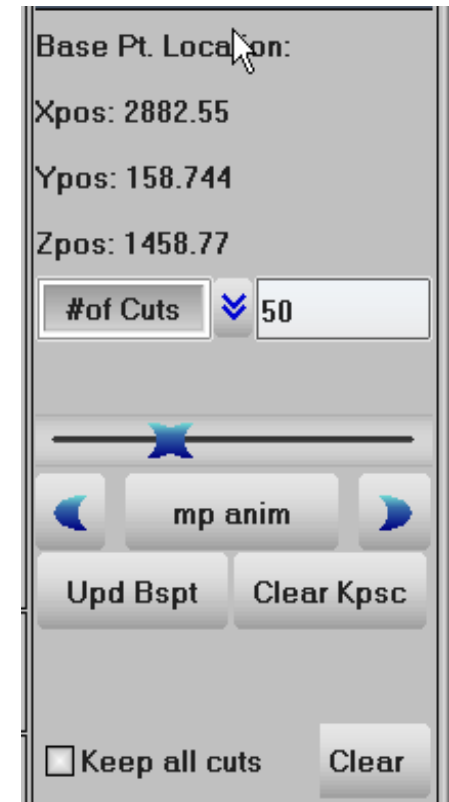
- ☐ Centroid – make centroid of model the base point
- ☐ Reset – reset and clear plane definition
- ☐ Keep all cuts – Display all the section cuts
- ☐ Clear – clear all section cuts and only keep the last cut
- ☐ Cut – perform section cut
- ☐ Model – reset display to full model view
- ☐ Force – open section force interface
- ☐ Mvp – open moving plane interface
- ☐ Options – open other options for section plane
- ☐ Crush – open crush/intrusion interface
- ☐ Line – open line plot interface
- ☐ Meas – open section cut measure interface
- ☐ Save – save plane definitions in memory and write to file or read from file



Section Plane

Mvp – Moving plane interface

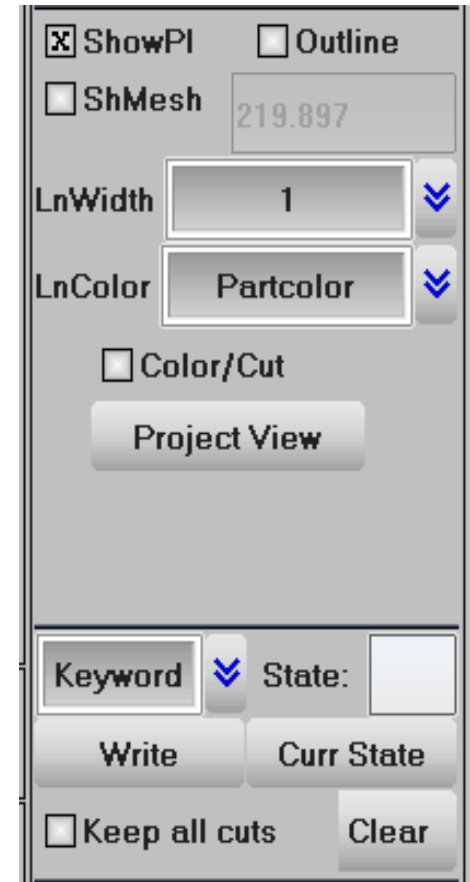
- ☐ #of Cuts to divide model along the plane normal direction
- ☐ Use  to move plane forward or backward
- ☐ Mpanim button to animate section cut
- ☐ Updbstpt – update base point with current position
- ☐ Clear Kpsc – clear all kept section cuts from memory



Section Plane

Options – Misc. options for section plane

- ☐ ShowPl – show section plane on/off
- ☐ ShMesh – show section plane as mesh
- ☐ Outline – show solid part outline after cut
- ☐ LnWidth – set section cut line width
- ☐ LnColor – set section cut line color
- ☐ Color/Cut – use different color for each cut instead of different color for each part
- ☐ Project View – view section cut in projected view normal to the plane
- ☐ Write – write section cut to file in selected format
 - ❖ Keyword, Vda curves, Iges curves
- ☐ Curr State – set current state for writing the section cut



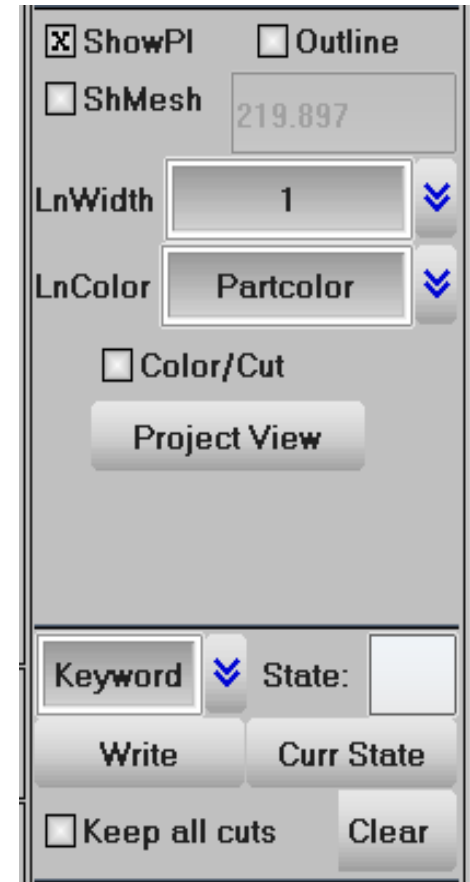
The dialog box contains the following controls:

- ☒ ShowPl
- ☐ Outline
- ☐ ShMesh
- Value: 219.897
- LnWidth: 1 (dropdown arrow)
- LnColor: Partcolor (dropdown arrow)
- ☐ Color/Cut
- Project View (button)
- Keyword (dropdown arrow)
- State: (text field)
- Write (button)
- Curr State (button)
- ☐ Keep all cuts
- Clear (button)

Section Plane

Options – Misc. options for section plane

- ☐ ShowPl – show section plane on/off
- ☐ ShMesh – show section plane as mesh
- ☐ Outline – show solid part outline after cut
- ☐ LnWidth – set section cut line width
- ☐ LnColor – set section cut line color
- ☐ Color/Cut – use different color for each cut instead of different color for each part
- ☐ Project View – view section cut in projected view normal to the plane
- ☐ Write – write section cut to file in selected format
 - ❖ Keyword, Vda curves, Iges curves
- ☐ Curr State – set current state for writing the section cut



The dialog box contains the following controls:

- ☒ ShowPl
- ☐ Outline
- ☐ ShMesh
- Value: 219.897
- LnWidth: 1 (dropdown arrow)
- LnColor: Partcolor (dropdown arrow)
- ☐ Color/Cut
- Project View (button)
- Keyword (dropdown arrow)
- State: (text field)
- Write (button)
- Curr State (button)
- ☐ Keep all cuts
- Clear (button)

Section Plane

Cursh – Intrusion plot – measure distance from node to plane

- ☐ LineWidth – Intrusion plot line width
- ☐ Line Color – Intrusion plot line color
- ☐ PlotType – select history plot type
- ☐ Node Id can be picked or key in
- ☐ Plot, New, Padd – standard lsprepost plotting buttons

Selected PartID: 7

Coord at X = 0:

X= 849.61362

Y= 0.000123

Z= 808.303894

Coord at X = End:

X= 1625.957642

Y= 0.000122

Z= 990.140747

Plot New Padd

Flip Raise Pop

Line – plot fringe values at section cut along the length of the cut

- ☐ Must load fringe value first (fcomp)
- ☐ Must perform cut first
- ☐ Pick a part for the line plot
- ☐ Mainly for metal stamping

Intrusion Plotting:

Line Width 1

Line Color White

PlotType Crush

Node ID:

Plot New Padd

Clear Raise Pop

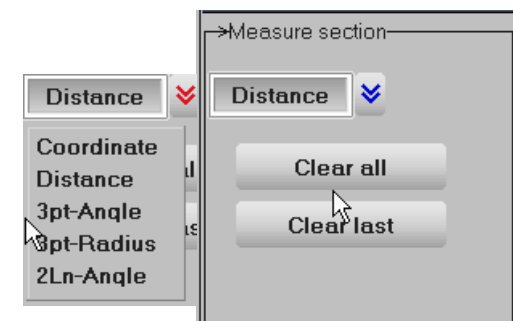
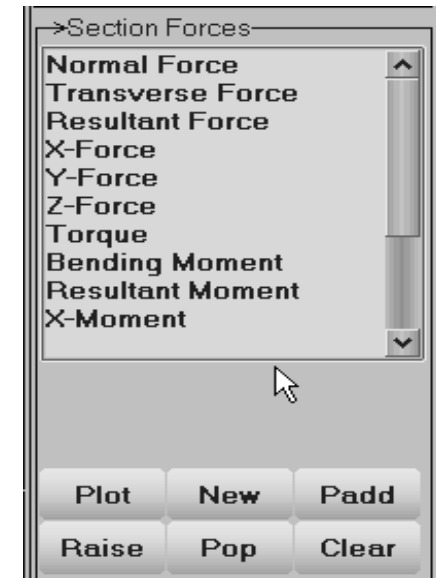
Section Plane

Force – Calculate and plot the section forces

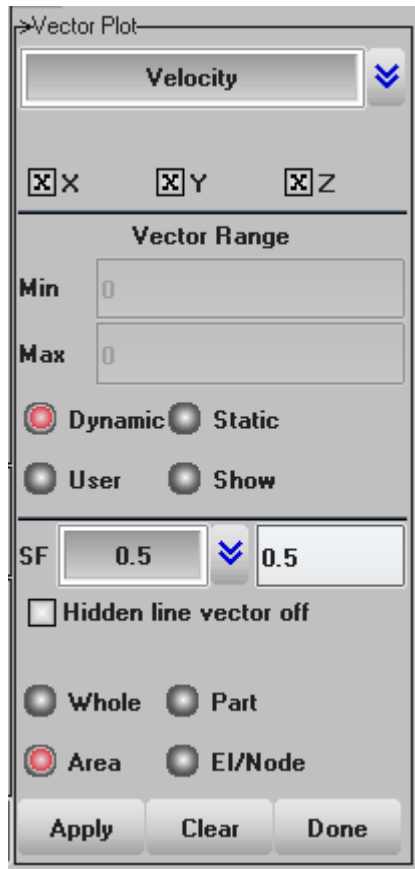
- ☐ Force and moment components
- ☐ Area can also be plotted

Measure – Section cut measure interface

- ☐ Coordinate
- ☐ Distance
- ☐ 3pt – Angle
- ☐ 3pt – radius
- ☐ 2ln- angles
- ☐ Measure can be done from any cut to any cut

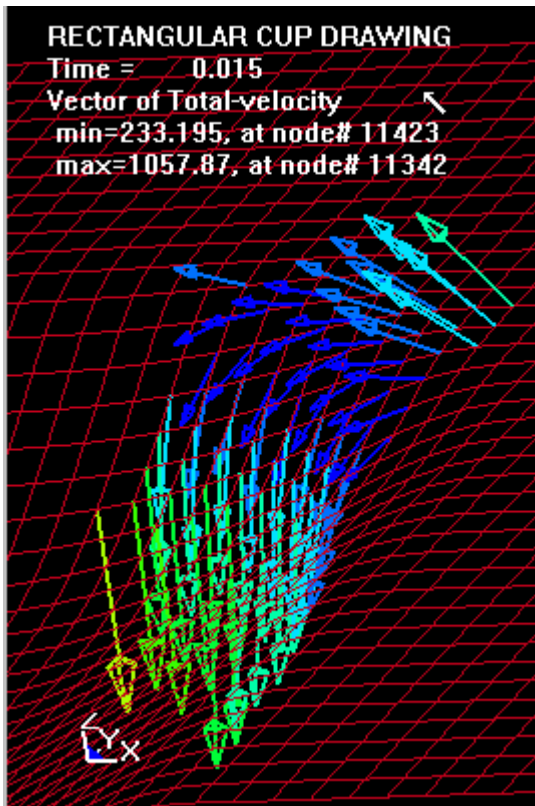


Vector Plot Menu

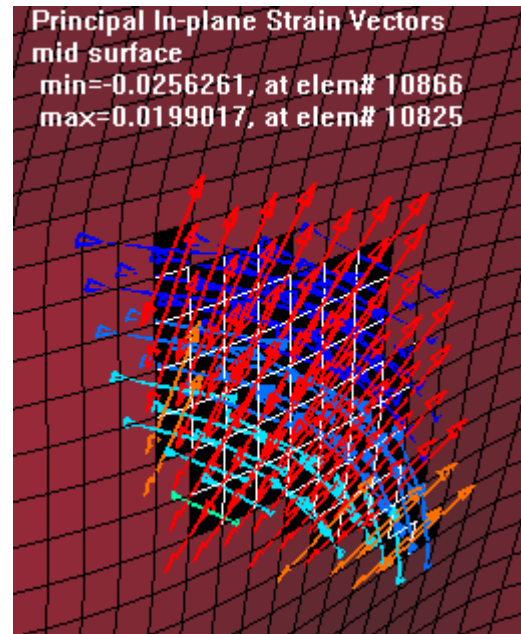


- ☐ Vector display – Shell Normal, Displacements, Velocities and Principal stress and strain directions for the d3plot file
- ☐ Additionally, Forces, Pressures and Surface shear for the Interface Force file.
- ☐ Components for displacement, velocity and force vectors can be selected
- ☐ Dynamic – sets range automatically for each state
- ☐ Static – applies the range over all the states
- ☐ User – allows range to be applied
- ☐ Show – limits the display of vectors between a range
- ☐ SF – scale factor allows large or small vectors to be adjusted
- ☐ Whole/Part/Area/El\Node – adjusts what is displayed

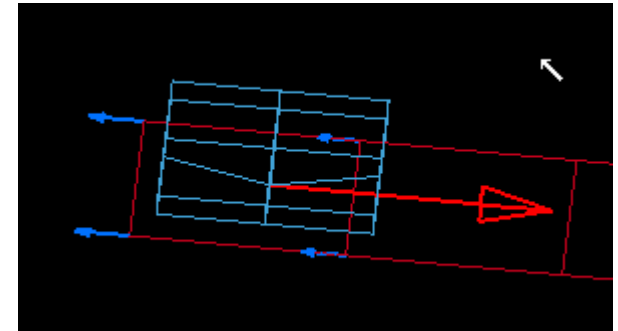
Vector Menu



Velocity



Principal Strain

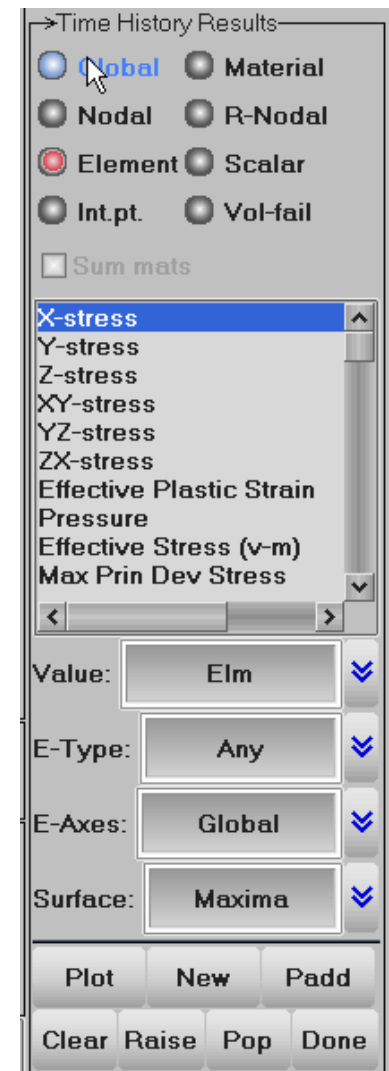


Nodal Surface Shear

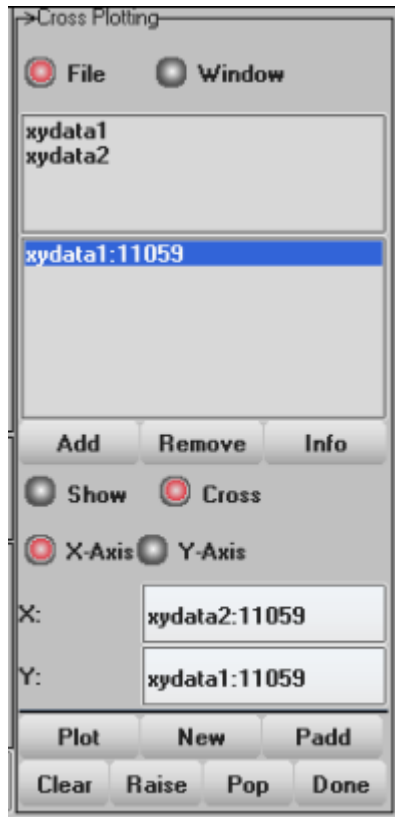
Time History Interface

History – to plot time history data

- ☐ Element data
 - ❖ Element stresses and strains
 - ❖ Pick elements to plot data
 - ❖ Value – set to element value or min/max value for the part
 - ❖ E-Type – choose element type to plot history
 - ❖ E-Axes – global or local
 - ❖ Surface – choose shell element through the thickness locations
- ☐ Scalar – plot scalar value that is being fringed (choose component in Fcomp interface)
- ☐ Int.pt – Select element integration points

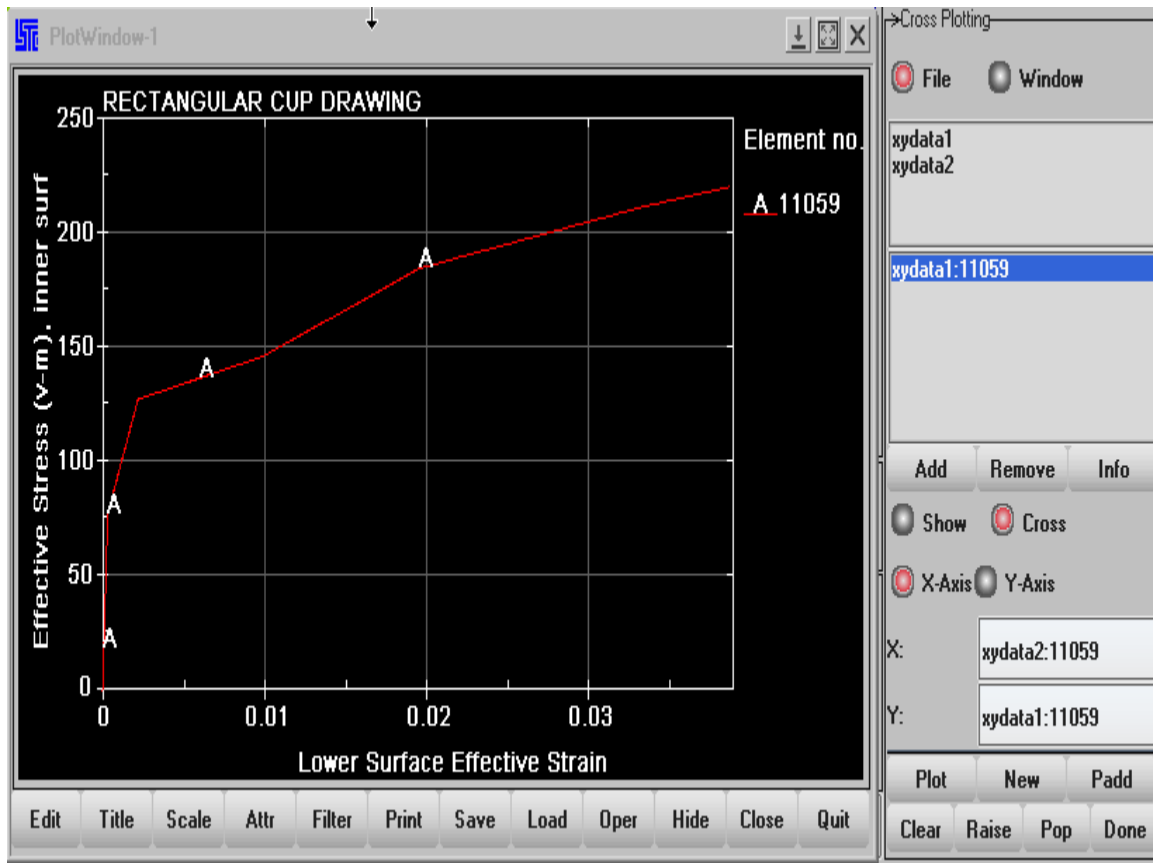


XYPlot – show and cross plotting



- ☐ Import xy plot data as: list of points, keep file, and curve file
- ☐ All xy data saved in the xy plot windows is automatically add to the file list.
- ☐ Files can be added by the File menu or Add button. Remove button deletes file from list.
- ☐ All plot windows open are also listed under the Window toggle button.
- ☐ So any point data can be shown and added to a current plot.
- ☐ When data from one plot is required to be related to another plot, so called cross plotted can be used

XY plot – cross plot



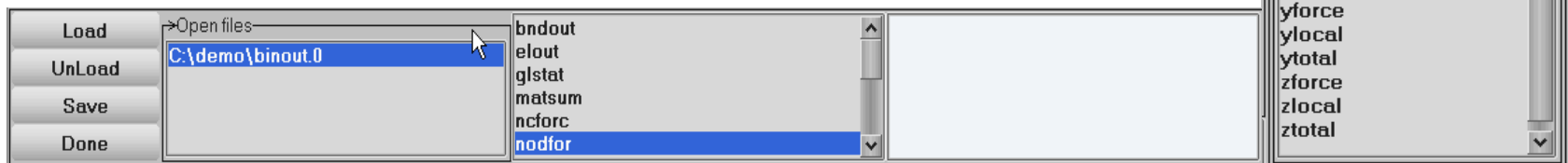
- ☐ Two files are added to file list
- ☐ Select 2nd file, and each curve in file is then listed
- ☐ Select curve into X-axis location
- ☐ Do the same for 1st file into Y-axis location
- ☐ Press plot button
- ☐ Stress versus Strain plot



Binout

BINOUT – An equivalent of ASCII files but in LSDA binary format

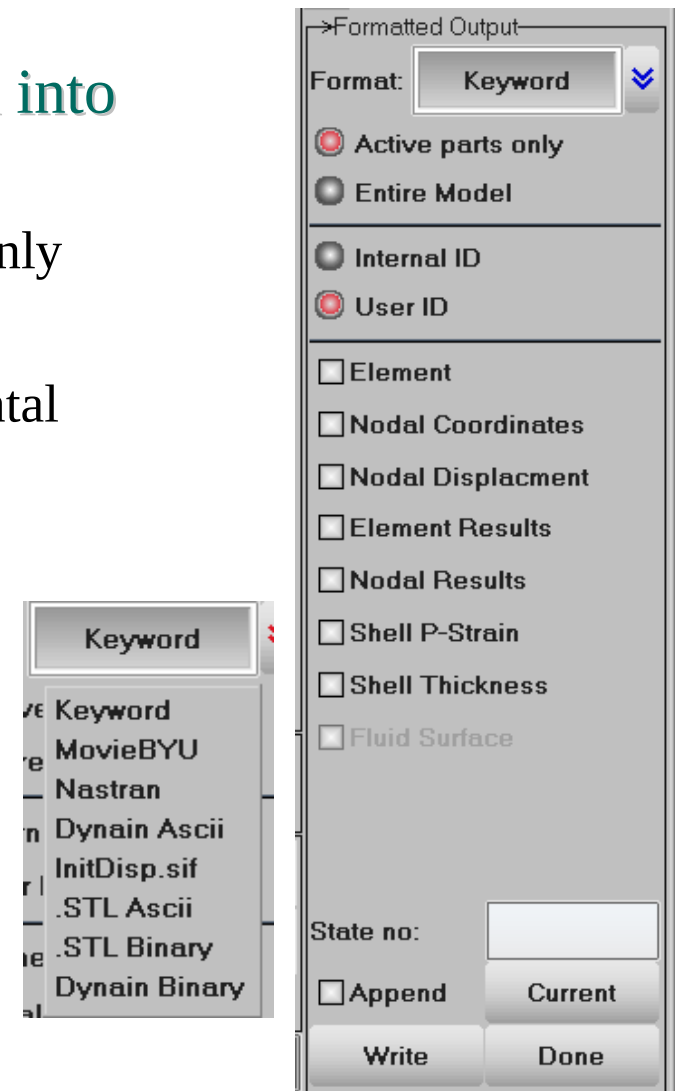
- ☐ *DATABASE_{options} gives users to turn on this feature.
- ☐ LSPREPOST gives most up-to-date support for BINOUT
- ☐ BINOUT file tells lsprepost what data have been stored
- ☐ The user interface is capable of handling multiple BINOUT files, that user can compare from one simulation to another.



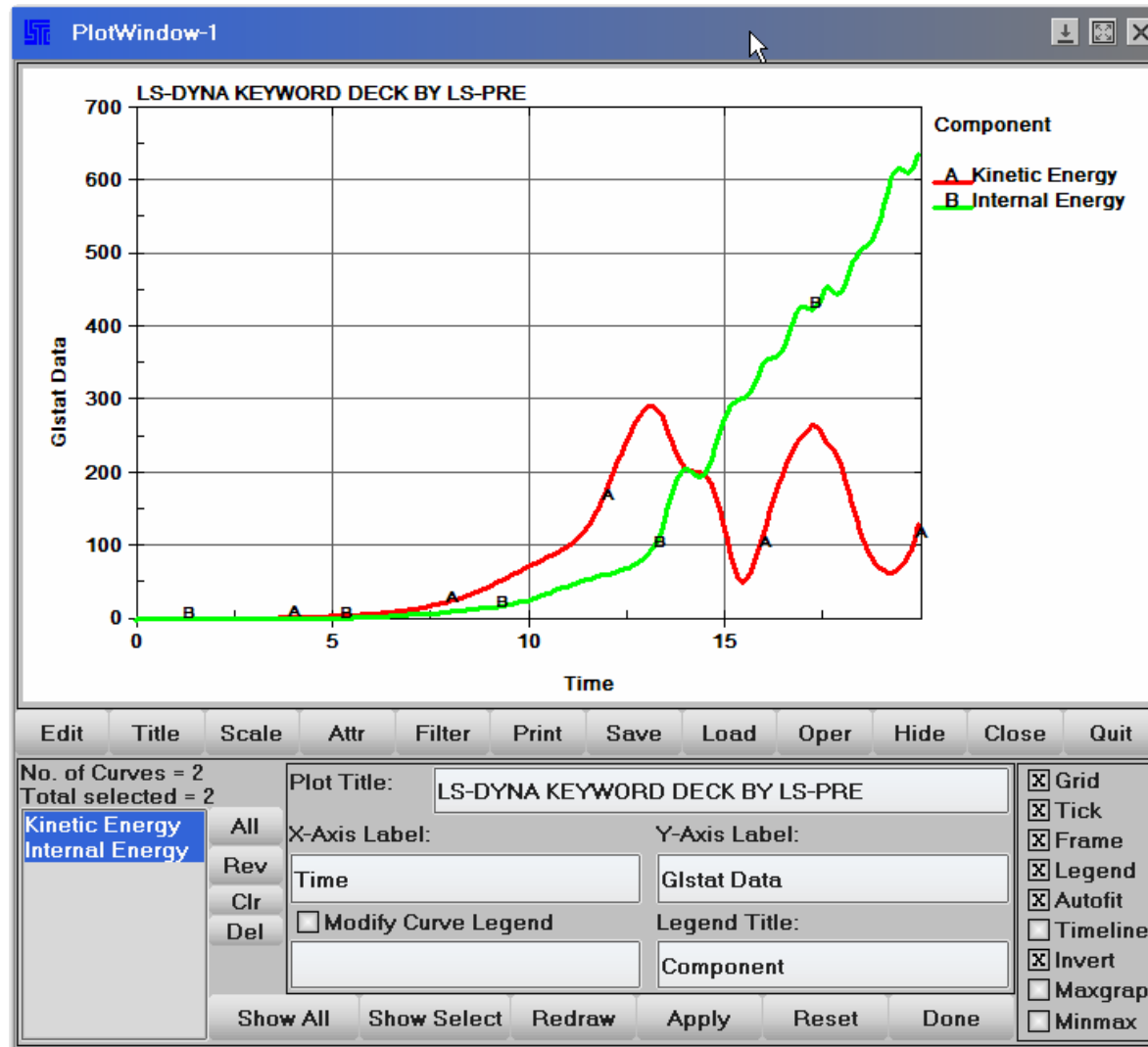
Formatted Output

Purpose: To output post-processing data into various formatted file

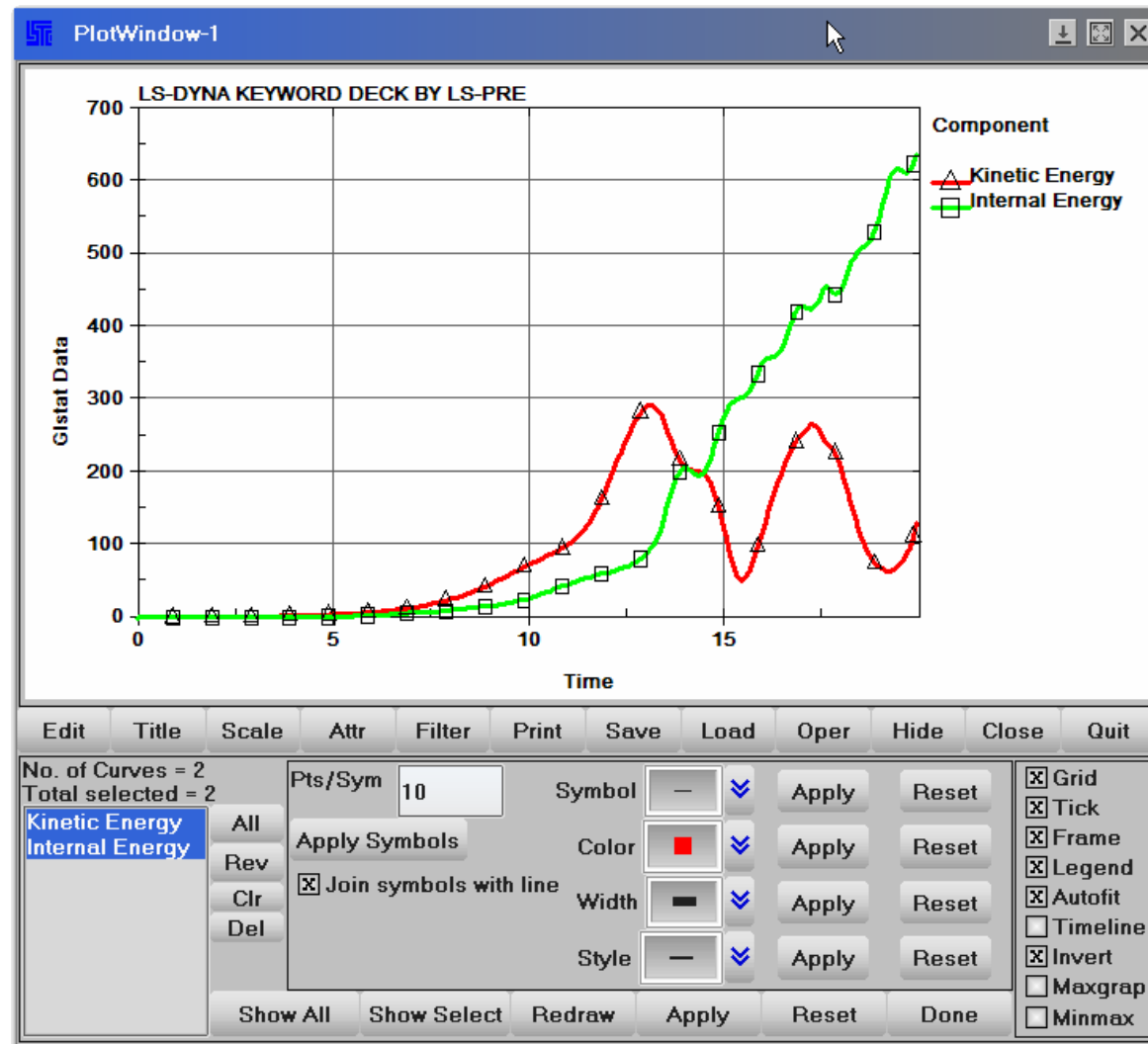
- ☐ Option for whole model or active parts only
- ☐ Output in user Ids or internal Ids
- ☐ Output model itself or with nodal/elemental results
- ☐ State no selection
- ☐ Format selection:
 - ❖ Keyword
 - ❖ Nastran
 - ❖ Dynain
 - ❖ STL ascii or binary



XYPlot Window



XYPLOT Window



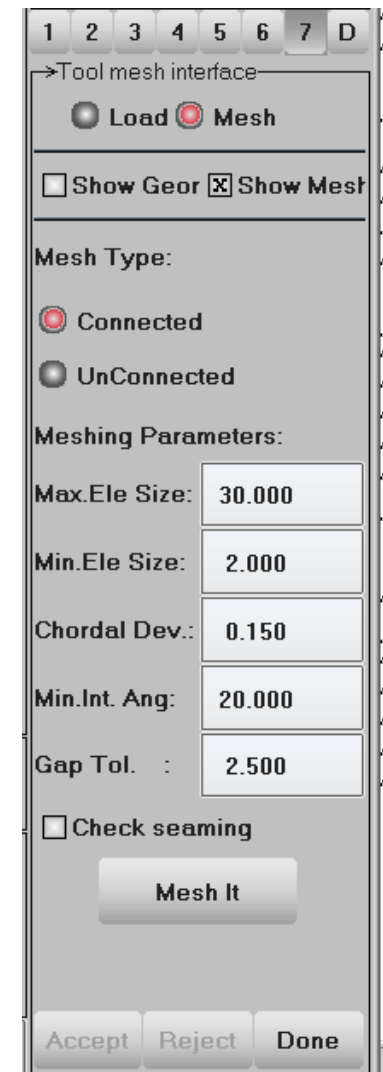
LS-PREPOST

Pre-Processing

Meshing – TMesh and AMesh

Tmesh and Amesh are 3rd party meshers that require addition license and cost

- ☐ Tmesh – tool meshing for metal stamping
- ☐ Amesh – topology meshing for regular car model
- ☐ Read Iges or Vda geometry file
- ☐ Select surfaces to be meshed
- ☐ Enter meshing parameters
- ☐ Mesh it, accept or reject mesh



The screenshot shows the 'Tool mesh interface' window. At the top, there are tabs numbered 1 through 7 and a 'D' button. Below the tabs, there are two radio buttons: 'Load' (unselected) and 'Mesh' (selected). Below these are two checkboxes: 'Show Geor' (unselected) and 'Show Mesh' (selected). The 'Mesh Type' section has two radio buttons: 'Connected' (selected) and 'UnConnected' (unselected). The 'Meshing Parameters' section contains several input fields: 'Max.Ele Size' (30.000), 'Min.Ele Size' (2.000), 'Chordal Dev.' (0.150), 'Min.Int. Ang' (20.000), and 'Gap Tol.' (2.500). Below these fields is a checkbox for 'Check seaming' (unselected). At the bottom of the parameter section is a 'Mesh It' button. At the very bottom of the window are three buttons: 'Accept', 'Reject', and 'Done'.

Meshing – 2d Mesher with sketch board

2dmesh – is the magic mesher from
Maze

- ☐ 2d sketch board to create points, lines, arcs and curves
- ☐ Use curves to form outline of mesh
- ☐ Define mesh density on the outlines of the geometry
- ☐ Mesh it and accept or reject

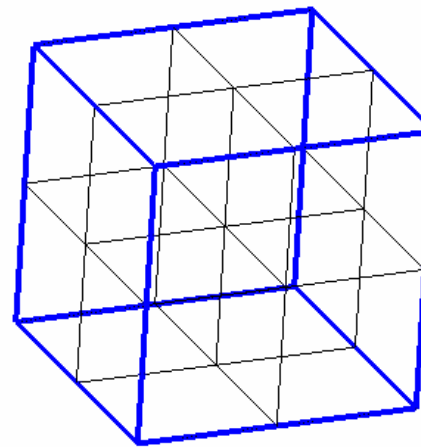
Meshing – Simple mesh creation

Purpose: To create simple geometry

Create Solid by giving box value.

Create Shell by give four positions.

Create Cylinder shell by give Radius, Length and direction.



Meshing

Entity:

☐ Region

	PMin	PMax
X	0.0	100.0
Y	0.0	100.0
Z	0.0	100.0

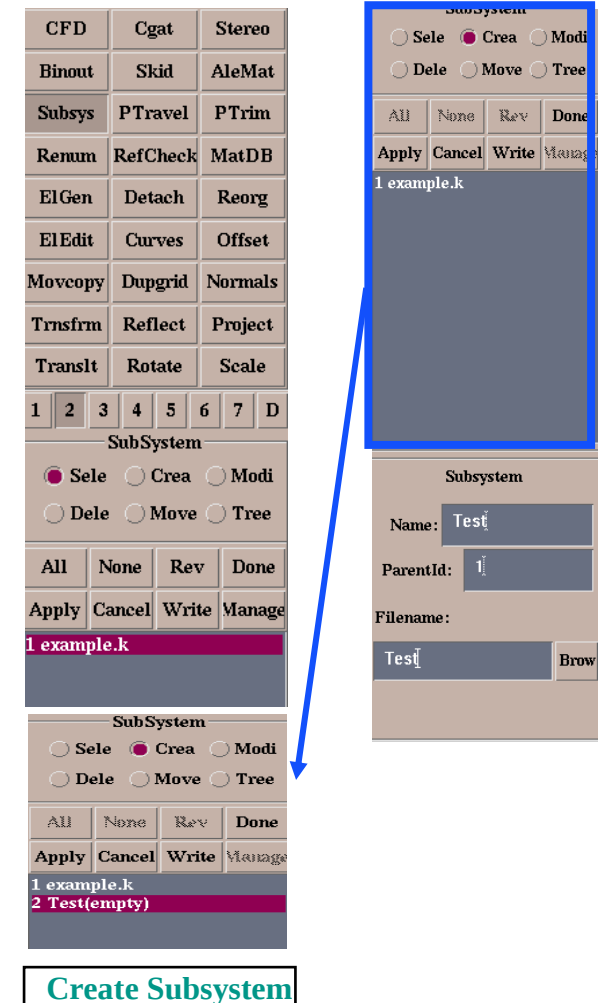
Mesh Density:

Nx:	2		1
Ny:	2		1
Nz:	2		1

Subsystem

SUBSYSTEM Creation and modification

- ☐ Files can be splitted to smaller subsystems
- ☐ Subsystems can be modified or deleted
- ☐ Managing of subsystems can be easily carried out
- ☐ Create Subsystem
 - ❖ Read in Keyfile “example.k” and go to page 2
 - ❖ Select “Subsys”, this will show the single file “example.k” as the only existing subsystem
 - ❖ Select “Crea”
 - ❖ Type in name of subsystem
 - ❖ Parent ID is the ID of “example.k” → 1
 - ❖ Type in optional File name and “Apply”



SubSystem

☐ Sele ☒ Crea ☐ Modi
☐ Dele ☐ Move ☐ Tree

All None Rev Done

Apply Cancel Write Manage

1 example.k

SubSystem

☐ Sele ☒ Crea ☐ Modi
☐ Dele ☐ Move ☐ Tree

All None Rev Done

Apply Cancel Write Manage

1 example.k
 2 Test(empty)

Create Subsystem

Subsystem

Sele Crea Modi
 Dele Move Tree

All None Rev Done

Apply Cancel Write Manage

Name: Test

ParentId: 1

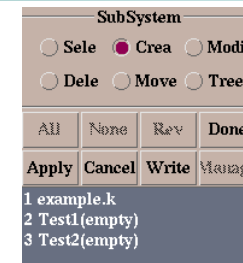
Filename: Test

Brow

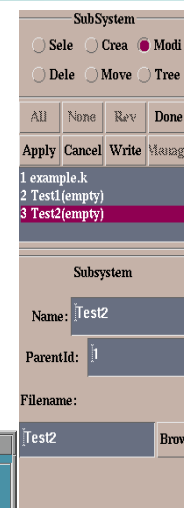
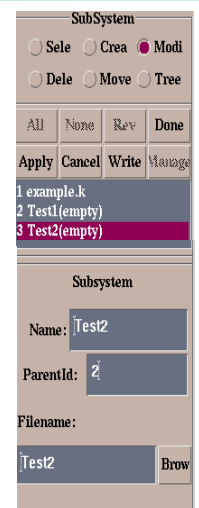
Subsystem

❑ Modifying a Subsystem

- ❖ The parent ID and the name of the subsystem can be modified using this feature
- ❖ Create a new subsystem “Test2”
- ❖ After Creation select “Modi” and choose “Test2”
- ❖ Change Parent ID from 1 to 2



Create
Test2

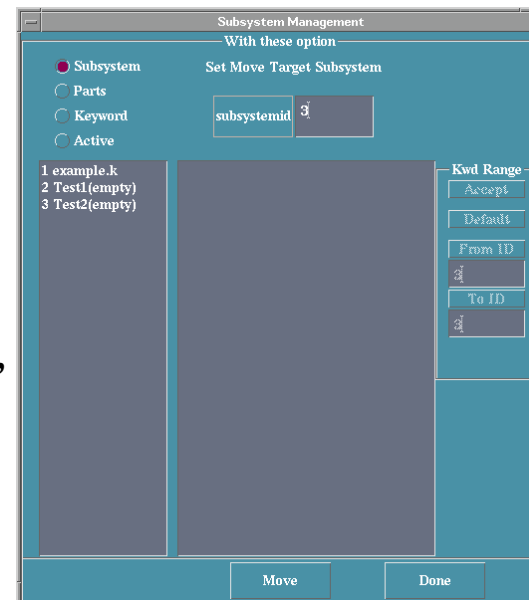
Modify Parent ID

❑ Delete Subsystem

- ❖ Used to delete subsystems

❑ Move – Subsystem Management

- ❖ This feature can be used to move parts, keywords, active parts (parts selected on screen using “selpar” on page1) to alternate subsystems



Subsystem
management

Subsystem

With these option

☐ Subsystem
☒ Parts
☐ Keyword
☐ Active

Set Move Target Subsystem

subsystemid 3

(sub 1)1 material ty
(sub 1)2 material ty
(sub 1)3 material ty
(sub 1)4 material ty
(sub 1)5 material ty
(sub 1)6 material ty

With these option

☐ Subsystem
☐ Parts
☒ Keyword
☐ Active

Set Move Target Subsystem

subsystemid 3

CONTACT(2)
CONTROL(1)
DATABASE(10)
ELEMENT(1174)
HOURLASS(6)
INITIAL(576)
MAT(6)
NODE(1236)
PART(6)
RIGIDWALL(2)
SECTION(6)
SET(3)

With these option

☐ Subsystem
☐ Parts
☐ Keyword
☒ Active

Set Move Target Subsystem

subsystemid 3

(sub 1)1 material ty
(sub 1)2 material ty
(sub 1)3 material ty

With these option

☐ Subsystem
☐ Parts
☐ Keyword
☒ Active

Set Move Target Subsystem

subsystemid 3

(sub 3)1 material ty
(sub 3)2 material ty
(sub 3)3 material ty

By part

By keyword

By Active Part

Moving part 1,2 & 3
to subsystem 3

Move – Subsystem Management

- ❖ Select move in subsystem management interface
- ❖ Select move by active parts and move parts 1, 2 & 3 to subsystem 3 and parts 4, 5 & 6 to subsystem 2.
- ❖ The change is reflected in the Subsystem menu

SubSystem

☐ Sele ☐ Crea ☐ Modi
☐ Dele ☒ Move ☐ Tree

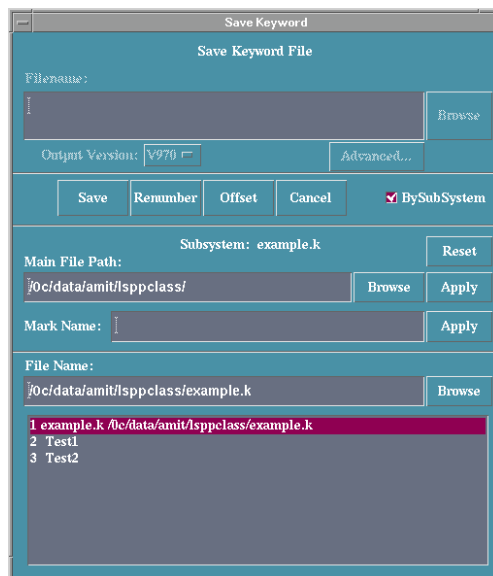
All None Rev Done

Apply Cancel Write Manage

1 example.k
2 Test1
3 Test2

All subsystems now have
some parts in them

Subsystem



Save Keyword File

Filename: Browse

Output Version: Advanced...

Save Renumber Offset Cancel ☒ BySubSystem

Subsystem: example.k Reset

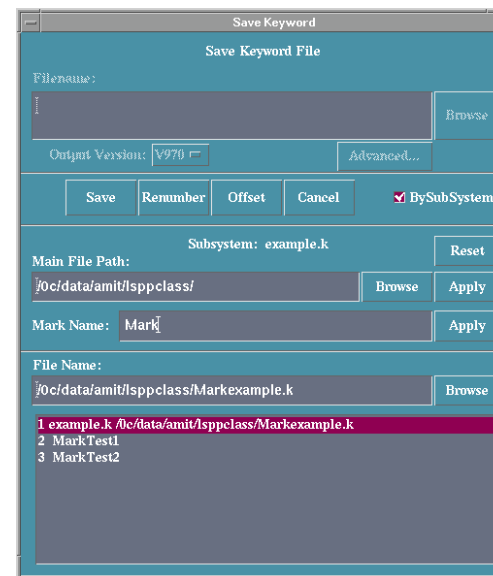
Main File Path: Browse Apply

Mark Name: Apply

File Name: Browse

1 example.k /0c/data/amt/lspclass/example.k
2 Test1
3 Test2

Saving subsystems



Save Keyword File

Filename: Browse

Output Version: Advanced...

Save Renumber Offset Cancel ☒ BySubSystem

Subsystem: example.k Reset

Main File Path: Browse Apply

Mark Name: Apply

File Name: Browse

1 example.k /0c/data/amt/lspclass/Markexample.k
2 MarkTest1
3 MarkTest2

Save subsystems by marks

❑ Save file by selecting “by Subsystem”

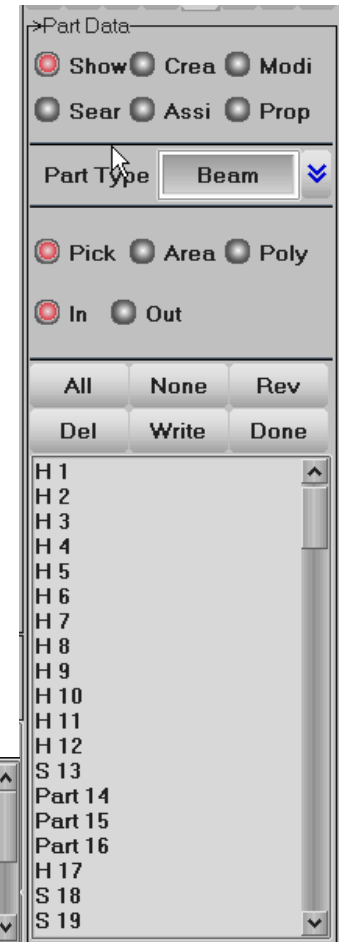
- ❖ Files can be saved by Subsystems using this feature available in the File Menu
- ❖ If subsystems already exist and you do not want to disturb the existing files then a “mark” can be added to the file names and then saved as shown above

Part Data

Part data interface allows user to show, create and modify part data

- ☐ Show – show part data information
 - ❖ No. of Nodes and elements are included
 - ❖ Use extended selection on the part list
 - ❖ Use mouse single pick, area select or polygon select
 - ❖ Delete parts will delete all elements in the part
 - ❖ Write part data will include *MAT, *SECTION, *EOS, *HOURLASS and any *DEFINE_CURVE, *DEFINE_COORDINATE data

```
Solid PID=2 SID=2 MID=2 EOSID=0 HOURSID=0 #Elms=16 #Nodes=48 MinEID=2 MaxEID=17 MinNID=8 MaxNID=55
Solid PID=3 SID=3 MID=3 EOSID=0 HOURSID=0 #Elms=16 #Nodes=48 MinEID=18 MaxEID=33 MinNID=57 MaxNID=104
Solid PID=5 SID=5 MID=5 EOSID=0 HOURSID=0 #Elms=16 #Nodes=48 MinEID=242 MaxEID=257 MinNID=207 MaxNID=416
Solid PID=7 SID=7 MID=7 EOSID=0 HOURSID=0 #Elms=1 #Nodes=8 MinEID=274 MaxEID=274 MinNID=539 MaxNID=546
Solid PID=8 SID=8 MID=8 EOSID=0 HOURSID=0 #Elms=1 #Nodes=8 MinEID=275 MaxEID=275 MinNID=547 MaxNID=554
Solid PID=10 SID=10 MID=10 EOSID=0 HOURSID=0 #Elms=8 #Nodes=27 MinEID=1611 MaxEID=1618 MinNID=1 MaxNID=4895
```



Part Data

☒ Show ☐ Crea ☐ Modi

☐ Sear ☐ Assi ☐ Prop

Part Type: Beam

☒ Pick ☐ Area ☐ Poly

☒ In ☐ Out

All None Rev

Del Write Done

H 1
H 2
H 3
H 4
H 5
H 6
H 7
H 8
H 9
H 10
H 11
H 12
S 13
Part 14
Part 15
Part 16
H 17
S 18
S 19

Part Data

- ❑ Create – create new part data according to the part type: Beam, Shell, Solid, Tshell, Mass, Discrete, Seatbelt, Inerta, Rsurf, Sphnode, Fluid
 - ❖ NewID – take old part data and create a new one with a new ID
 - ❖ Add – create a new part with all data clear
 - ❖ Accept – accept the data creation
 - ❖ Using bottom working area input part data or get them from link dialog
- ❑ Modi – modify the existing part data
- ❑ Only single selection on the part list
- ❑ Use mouse to select part in the graphics area

Part Title						PSHELL : 1 CQUAD4:SPINE	
SECID	MID	OSID	HGID	TMID	GRAV	0	
13	13	0	0	0	ADPOPT	0	

Part Data

☐ Show
 ☒ Crea
 ☐ Modi

☐ Sear
 ☐ Assi
 ☐ Prop

Part Type: Solid

Part ID: 1524

New ID
 Add
 Accept

All
 None
 Rev

Del
 Write
 Done

Part 16
H 17
 S 18
 S 19
 H 20
 S 21
 S 22
 H 23
 S 24
 H 25
 S 26
 H 27
 S 28
 Part 29
 S 30
 Part 31
 S 32
 Part 33
 S 34

Part Data

- ❑ Sear – search parts with specified data
 - ❖ Data specified will be used as filter
 - ❖ Multiple fields can be set
 - ❖ Load Refby – can load data from a referenced by buffer
- ❑ Assi – assign new data to a set of parts
 - ❖ Only fields being set will be assigned

SECID	MID	EOSID	HGID	TMID	GRAV	ADPOPT
13					NA	NA

Part Data

☐ Show
 ☐ Crea
 ☐ Modi
 ☒ Sear
 ☐ Assi
 ☐ Prop

Part Type: Solid

Search/Assign New data

- Part 83
- Part 84
- Part 85
- S 86
- H 87
- S 88
- Part 89
- Part 90
- H 91
- H 92
- H 93
- Part 301
- S 303
- S 304
- S 305
- S 306
- S 307
- S 308
- S 309

Part Data

- ❑ Prop – show properties of a selected part
 - ❖ Single selection on the part list
 - ❖ Each individual property can be examined
 - ❖ Modify - property data can be modified by going to the keyword data form
 - ❖ Refby – popup window to show all parts that use this property
 - ❖ AuMd – automatically popup keyword data form when the property is being selected

Material Data for Part Id : 65

*MAT_RIGID

65	6.0000E-7	0.005000	0.300000	0.0	0.0	0.0
0.0						
0.0	0.0	0.0	0.0	0.0	0.0	0.0

☐ Section
 ☐ Eos
 ☐ Gravity
 ☐ AuMd

☒ Material
 ☐ Hourglas
 ☐ Adaptivity

Part Data

☐ Show
 ☐ Crea
 ☐ Modi
 ☐ Sear
 ☐ Assi
 ☒ Prop

Part Type: Shell

☐ Pick
 ☐ Area
 ☐ Poly

☐ In
 ☐ Out

All
None
Rev

Del
Write
Done

S 30

Part 31

S 32

Part 33

S 34

Part 35

H 36

H 37

S 38

H 39

H 40

S 41

H 42

H 43

S 44

S 45

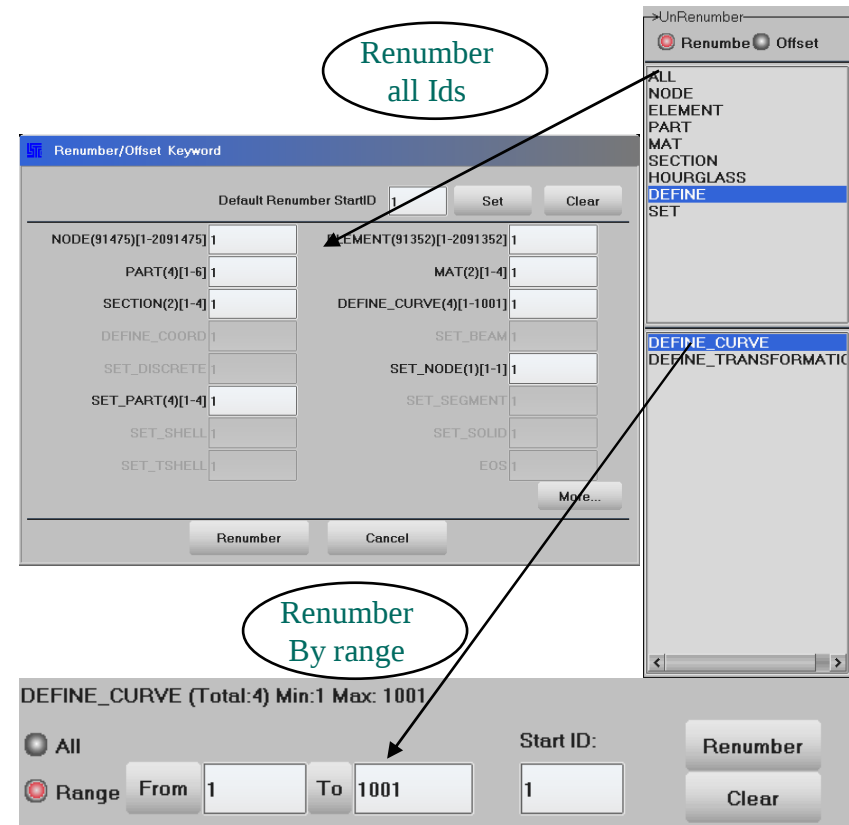
H 46

H 47

H 48

Renumber

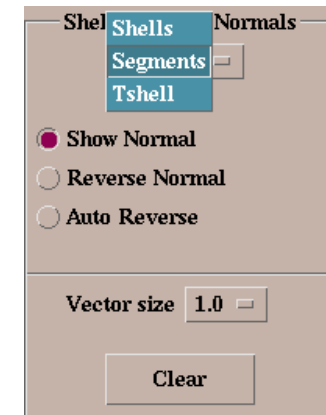
- ❑ allows the user to renumber and add offsets to model entities
- ❑ Renumber – Renumber selected entities using associated interface
- ❑ Offset – Add a numerical offset to selected entities using associated interface
- ❑ Renumber/Offset All, one entity or one entity by range.



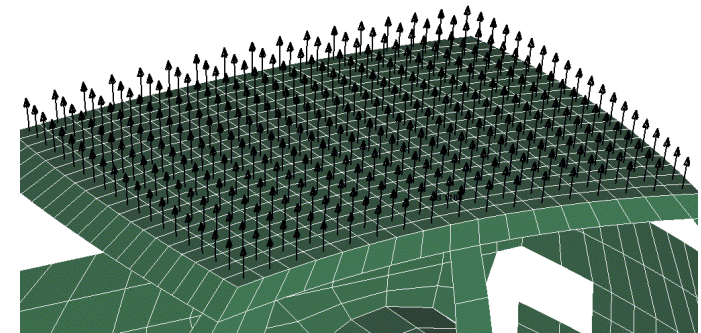
Normals

Element normals can be visualized and modified using this interface

- ☐ Load keyword pole.k
- ☐ Select part #3
- ☐ Select shells
- ☐ Select element + Propagate
- ☐ The element normals will be shown
- ☐ The vector normals size can be changed by selecting the appropriate size (vector size menu)



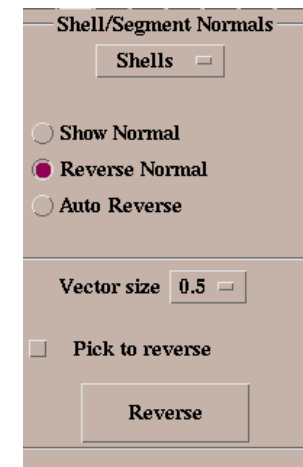
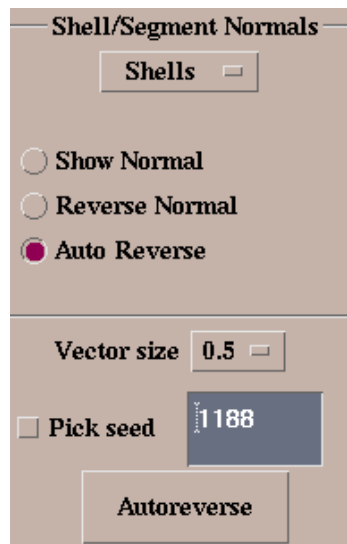
Keyin	☐ Label	Selecting Shells (0)		
LabOn	Save	Whole	ByNode	BySet
LabOff	Load	All Vis	ByElem	Propagate
Clear	Desel	Rev	ByPart	3Dsurf



Normals

To Reverse the normals of the selected elements

- ☐ Select Reverse Normals
- ☐ Click on reverse
- ☐ Individual element normals can be reverse by checking “pick to reverse” box



- ☐ To automatically reverse element normals by specifying an specific element direction, select auto reverse
- ☐ Check “Pick to reverse” box
- ☐ Select the correct element normal as the seed element
- ☐ Then the chosen elements (elements + propagate selected earlier) will reverse all the normals to the direction specified by the seed element

Duplicate grids

Dupgrid interface allows the user to show and merge duplicate grids

- ☐ Input a tolerance value
- ☐ Select “show” dup grid
- ☐ The duplicate grids are highlighted and the command is shifted to “merge” button
- ☐ The “clear” function releases the selected duplicate nodes
- ☐ Merge duplicate grids
- ☐ The interface can even show free edges in parts by selecting “show free edges”



Dupgrid Show/Merge

Tolerance:

☒ Keep smaller NID
☐ Keep Larger NID

☐ Show Free Edges

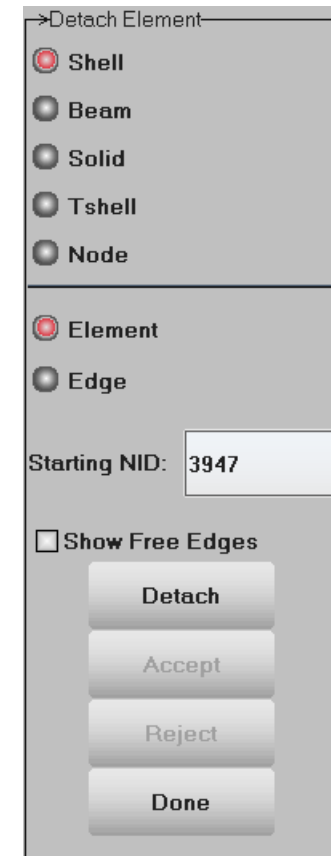
Line Width

Line Color

Detach

Detach: This interface allows the user to detach elements.

- ☐ Shell, Beam, Solid, Tshell – Separate chosen element type
- ☐ Node – Separate all elements at a node
- ☐ Element – Select group of elements to be detached
- ☐ Edge – Select edges of elements to be detached
- ☐ Starting NID – Enter starting node ID
- ☐ Show Free Edges – Displays free edges in white. Displays free edges in blue upon clicking [Detach]



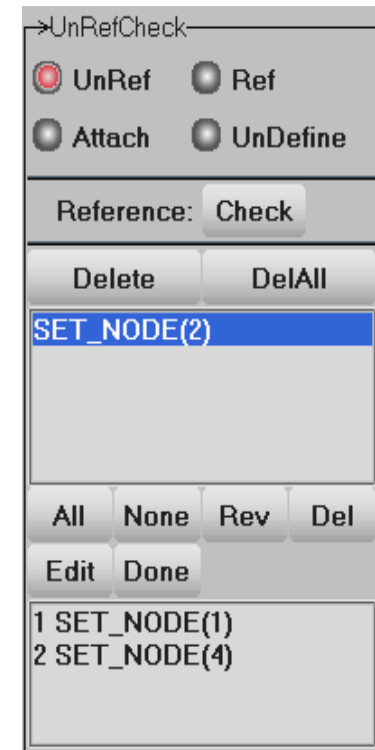
The dialog box is titled "Detach Element". It contains two sections of radio buttons. The first section has five options: Shell (selected), Beam, Solid, Tshell, and Node. The second section has two options: Element (selected) and Edge. Below these sections is a text field labeled "Starting NID:" with the value "3947". Below the text field is a checkbox labeled "Show Free Edges". At the bottom of the dialog are four buttons: "Detach", "Accept", "Reject", and "Done".

Reference Check - Unreferenced and Undefined

❑ Use this to check and delete all unreferenced entities

- ❖ Check for unreferenced entities
- ❖ Delete selected unreferenced data
- ❖ Delete all unreferenced data
- ❖ List of unreferenced entity types found
- ❖ Select, deselect, reverse unreferenced data of selected entity type
- ❖ Delete selected unreferenced data of selected entity type
- ❖ Edit selected unreferenced data of selected entity type

❑ Undefined entities can be checked/edited/deleted with this interface

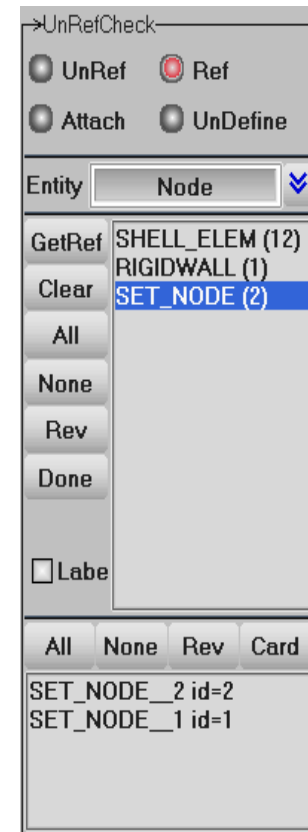


The image shows a software dialog box titled "UnRefCheck". It contains several controls: four radio buttons labeled "UnRef", "Ref", "Attach", and "UnDefine"; a "Reference:" label followed by a "Check" button; two buttons labeled "Delete" and "DelAll"; a list box containing the text "SET_NODE(2)" which is currently selected; a row of four buttons labeled "All", "None", "Rev", and "Del"; a row of two buttons labeled "Edit" and "Done"; and a text area at the bottom containing the text "1 SET_NODE(1)" and "2 SET_NODE(4)".

Reference Check – Reference

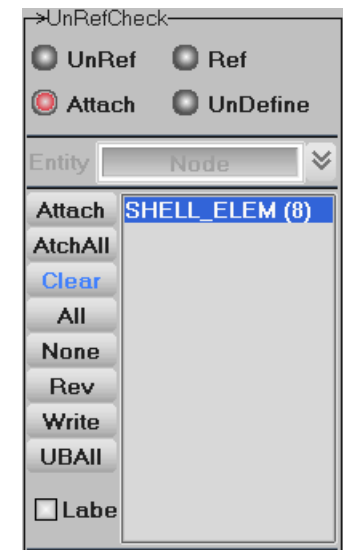
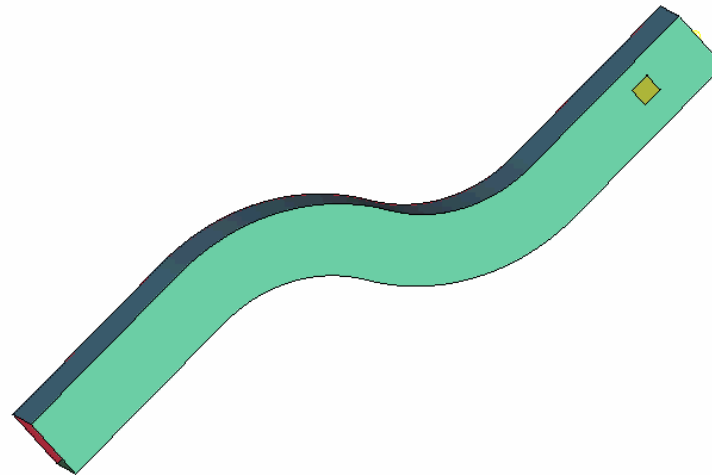
- ☐ Use this to display all referenced entities for the entity selection
- ☐ Select the entity type used to get reference
- ☐ Get all the referenced entities with the selection
- ☐ Clear entity selection
- ☐ All entities displayed
- ☐ Deselect all referenced data of selected entity type
- ☐ Reverse selection

- ☐ Selection of entities can be made with different options available. Entity selection/deselection can be made by picking nodes/elements/parts and also by area.



Reference Check - Attach

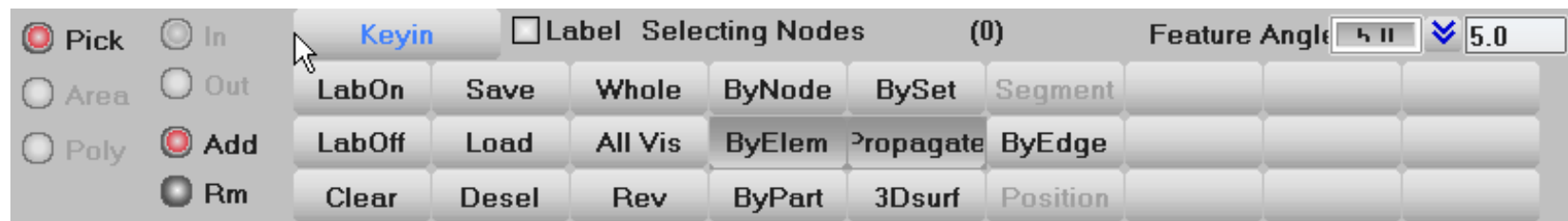
- ❑ Use this to find all the attached entities to the entity selection by using bottom area general selection to get nodes
 - ❖ Get all entities that are attached to the entities picked step by step or get all.
 - ❖ Write the displayed entities into a file
 - ❖ Unblank all the parts in display for restore model after attach
 - ❖ Attach for pick node by general selection.



General Selection

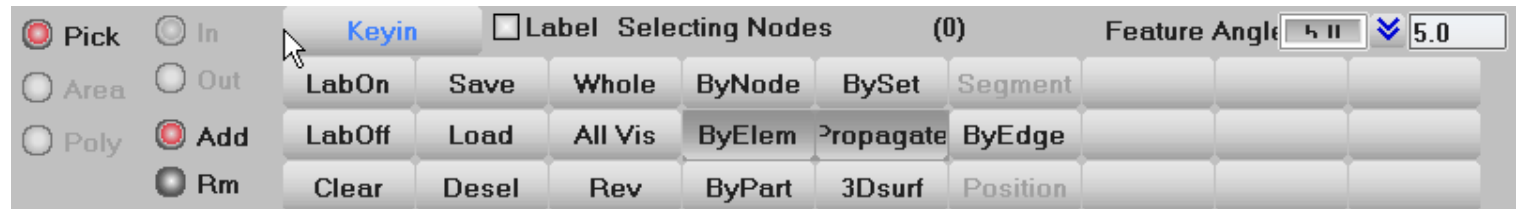
A general interface to select nodes/elements/parts/segment

- ☐ Many interfaces use General Selection
- ☐ Single pick, area select, or polygon select
- ☐ Add/Rm – add to list or remove from list
- ☐ Keyin – key in the entity Ids for selection
- ☐ Label – turn label on/off for new selection
- ☐ Labon – turn on labeling
- ☐ Laboff – turn off labeling
- ☐ Clear – clear the selection buffer



General Selection

- ☐ Save – save current selection into memory buffer
- ☐ Load – load selection from buffer and make it current selection
- ☐ Desel – deselect the last selected items
- ☐ Whole – set the whole model to be selected
- ☐ All Vis – set all display items to be selected
- ☐ Rev – reverse the selected and unselected items
- ☐ ByNode – select by nodes (for node selection only)
- ☐ ByElem – select by element (for node and element)
 - ❖ Propagate with feature angle (default to 2.0 degree)
- ☐ ByPart – select by part (for node, element or part)
- ☐ BySet – select the entity by set
- ☐ ByEdge – select by edges (for node or edge only)
- ☐ 3Dsurf – select only visible surface for solid element



Translate Mesh

Translate a group of nodes in a specified direction

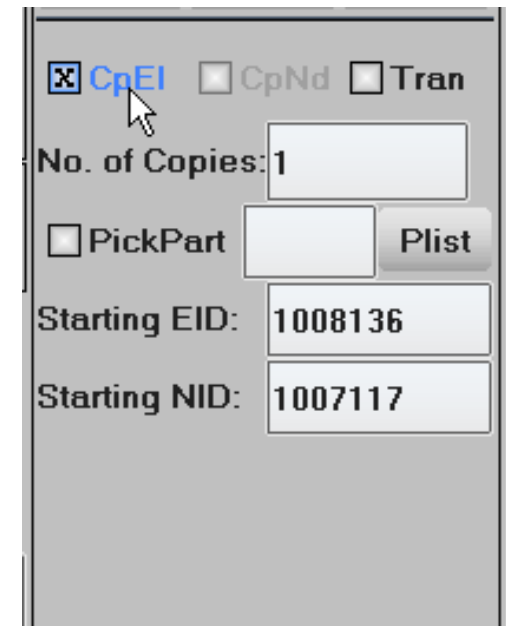
- ☐ New nodes and/or elements can be created with the copy option
- ☐ Translate Distance – distance to be translated
- ☐ X,Y,Z directions can be in global or local system
- ☐ N1-N2 – translate in the direction from node1 to node2
- ☐ N1-N2-N3 – translate in the normal direction formed by 3 nodes
- ☐ Pt-to-Pt – translate in the direction from point to point
- ☐ Sh-Normal – pick a shell element and use its normal as the direction of the translation



Translate Mesh - Copy Option

Nodes or Element can be created with CpEl or CpNd options

- ☐ Multiple copies can be created
- ☐ New elements can be assigned with new part Id
- ☐ User can set starting new element Id
- ☐ User can set starting new node Id

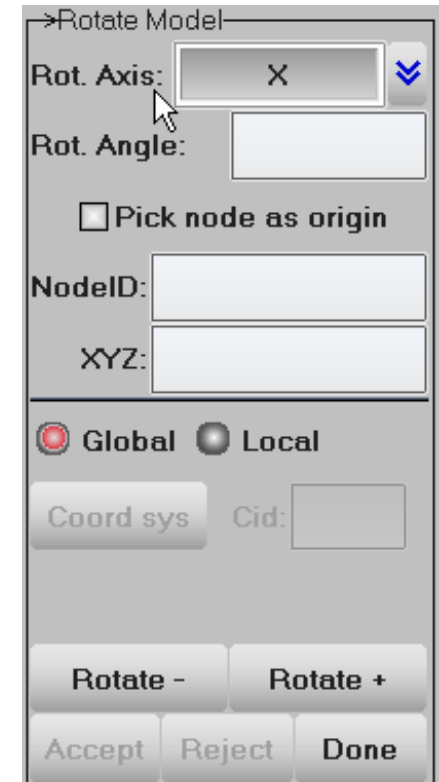


The screenshot shows a dialog box titled "Translate Mesh - Copy Option". It contains three radio buttons: "CpEl" (selected), "CpNd", and "Tran". Below the radio buttons is a text field labeled "No. of Copies:" with the value "1". There is a "PickPart" button and a "Plist" button. Below these are two text fields: "Starting EID:" with the value "1008136" and "Starting NID:" with the value "1007117".

Rotate Mesh

Rotate the selected mesh about an axis

- ☐ Axis of rotation in X, Y, Z direction can be in global or local system
- ☐ Pick a node as origin or
- ☐ Enter xyz position as origin of rotation
- ☐ N1-N2 – Node1 to Node2 form axis of rotation
- ☐ N1-N2-N3 – the normal of the plane formed by 3 nodes is the axis of rotation
- ☐ New elements or nodes can be created with copy options

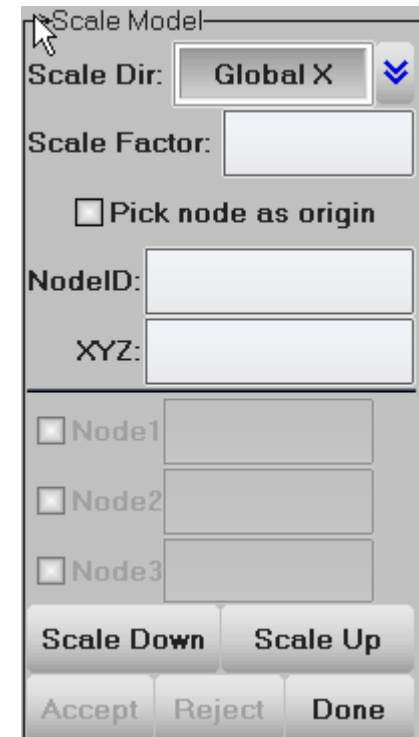


The screenshot shows the 'Rotate Model' dialog box. It has a title bar '>Rotate Model'. Inside, there is a 'Rot. Axis:' label with a dropdown menu showing 'X' and a blue arrow icon. Below it is a 'Rot. Angle:' label with an empty text box. A checkbox labeled 'Pick node as origin' is present. Below that are 'NodeID:' and 'XYZ:' labels, each with an empty text box. There are two radio buttons: 'Global' (selected) and 'Local'. Below these are 'Coord sys' and 'Cid:' labels with an empty text box. At the bottom, there are three buttons: 'Rotate -', 'Rotate +', and 'Accept'. There are also 'Reject' and 'Done' buttons.

Scale Mesh

Scale the selected mesh

- ☐ Scale in global X,Y,Z directions
- ☐ N1-N2 – define the direction of scaling
- ☐ N1-N2-N3 – scale in the direction of the normal of a plane formed by 3 nodes
- ☐ Scale factor greater than 1.0 to enlarge, smaller than 1.0 to shrink
- ☐ NodeID – to define a point of origin for scaling
- ☐ XYZ – or to define a position of origin for scaling
- ☐ Scale Up and down have the same effect as scale factor
- ☐ New nodes and elements can be created with copy option

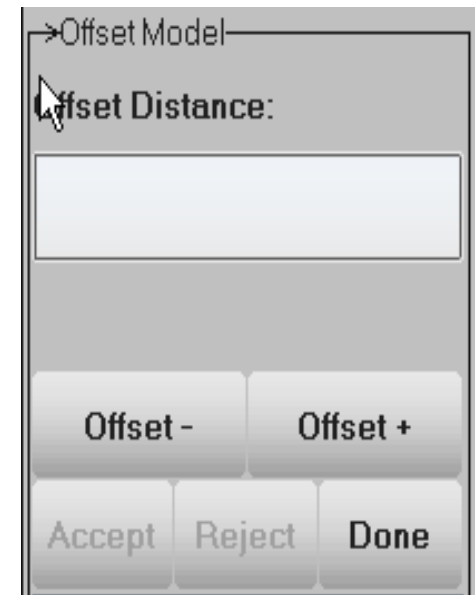


The image shows a 'Scale Model' dialog box. It has a title bar 'Scale Model'. Inside, there is a 'Scale Dir:' dropdown menu currently set to 'Global X'. Below it is a 'Scale Factor:' text input field. There is a checkbox labeled 'Pick node as origin'. Below that are 'NodeID:' and 'XYZ:' text input fields. Further down are three checkboxes labeled 'Node1', 'Node2', and 'Node3', each followed by a text input field. At the bottom, there are three buttons: 'Scale Down', 'Scale Up', and 'Done'. There are also 'Accept' and 'Reject' buttons at the very bottom.

Shell offset

Offset shell mesh in the direction of its normal

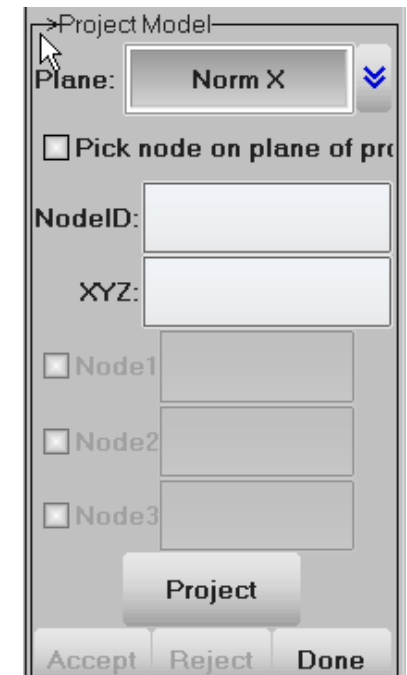
- ☐ Must make sure the normal of all shell element are consistent
- ☐ Must pay attention to small radius area if offset distance too big
- ☐ Offset+ offset in the positive normal direction
- ☐ Offset- offset in the negative normal direction
- ☐ New nodes and elements can be created with copy option



Project Mesh

Project mesh onto a plane

- ☐ Global planes normal to global X, Y, and Z axes
- ☐ N1-N2 – node1 to node2 forms the normal of the plane being projected to
- ☐ N1-N2-N3 – plane formed by 3 nodes
- ☐ NodeId or XYZ: a point that is on the plane of projection
- ☐ New nodes and elements can be created with the copy option



The image shows a software dialog box titled "Project Model". It contains the following elements:

- A "Plane:" label followed by a dropdown menu currently showing "Norm X" with a blue arrow icon to its right.
- A checkbox labeled "Pick node on plane of projection" which is currently unchecked.
- A "NodeID:" label followed by a text input field.
- An "XYZ:" label followed by a text input field.
- Three checkboxes labeled "Node1", "Node2", and "Node3", each followed by a text input field. All three checkboxes are currently unchecked.
- A "Project" button.
- At the bottom, three buttons: "Accept", "Reject", and "Done".

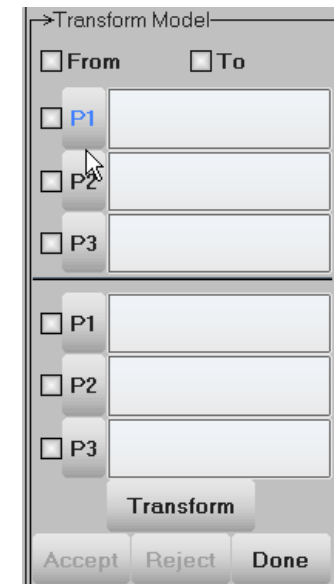
Transform Mesh

Transform mesh from one coordinate system to another coordinate system

- ☐ From – pick 3 nodes or positions to form the “From” coordinate system
- ☐ To – pick 3 nodes or positions to form the “to” coordinate system
- ☐ P1,P2,P3 – use the popup position interface

Popup position interface:

- ☐ Can be in global or local system
- ☐ Pick node or position (any where on element)
- ☐ C_Element – use center of element
- ☐ C_Edge – use center of an edge
- ☐ C_Area – use center of selection (nodes or elements)
- ☐ 3Ptcircle – center of a circle formed by 3 points
- ☐ Done will transfer the xyz position data to calling area



Transform Model

☐ From ☐ To

☐ P1

☐ P2

☐ P3

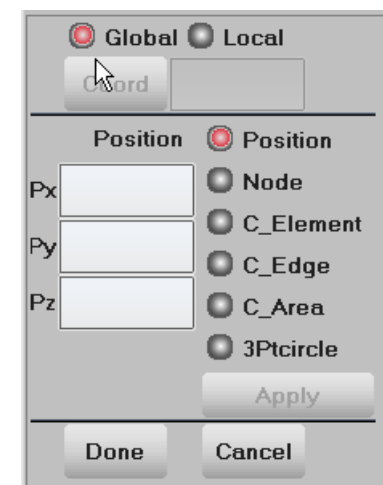
☐ P1

☐ P2

☐ P3

Transform

Accept Reject Done



☒ Global ☐ Local

Position ☒ Position

Px ☐ Node

Py ☐ C_Element

Pz ☐ C_Edge

☐ C_Area

☐ 3Ptcircle

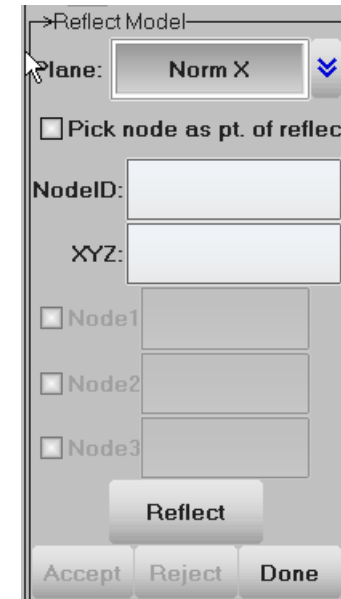
Apply

Done Cancel

Reflect Mesh

Reflect mesh about a symmetry plane

- ☐ Norm X – reflect about YZ plane normal to global X axis
- ☐ Norm Y – reflect about XZ plane normal to global Y axis
- ☐ Norm Z – reflect about XY plane normal to global Z axis
- ☐ N1-N2 – node1 to node2 form the normal of plane for reflection
- ☐ N1-N2-N3 – Plane of reflection formed by 3 nodes
- ☐ New nodes and elements can be created with copy option
- ☐ Other data like *INITIAL_STRESS_SHELL or element thickness will also be copied



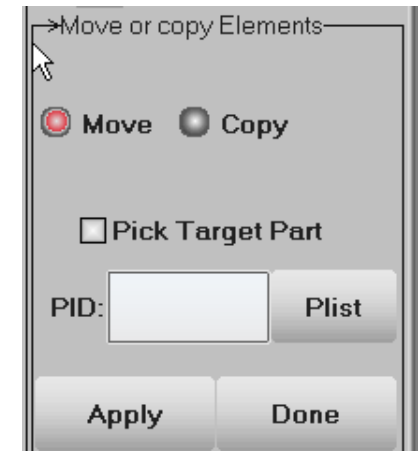
The dialog box titled "Reflect Model" contains the following elements:

- A "Plane:" dropdown menu with "Norm X" selected and a blue downward arrow.
- A checkbox labeled "Pick node as pt. of reflec" which is currently unchecked.
- A "NodeID:" text input field.
- An "XYZ:" text input field.
- Three checkboxes labeled "Node1", "Node2", and "Node3", all of which are unchecked.
- A "Reflect" button.
- Three buttons at the bottom: "Accept", "Reject", and "Done".

Move and Copy

Move or copy elements from one part to another part

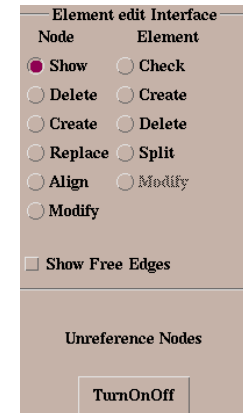
- ☐ Use general selection to select elements
- ☐ Target part can be a non-existing part
- ☐ Move – will only change part id for the selected elements
- ☐ Copy – will create new elements with the new part id



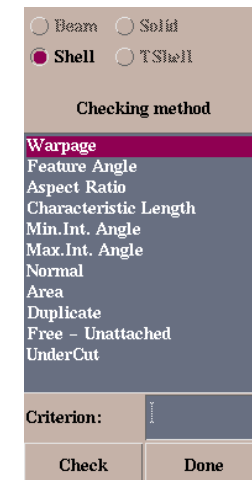
Element Edit

Node and Element creation and editing interface

- ❑ The interface can be used create and manipulate nodes and elements
 - ❖ The show feature can turn unreferenced nodes on/off
 - ❖ Both nodes and elements can be created
 - ❖ Nodes can be aligned and locations modified by moving the nodes simultaneously checking element quality
 - ❖ Shell element quality can be checked
 - ❖ Shell elements can be split



ELEDT interface



Element quality check

Element Edit - Node modification and creation

☐ NodePick	X		Node Id: 6182 NewID	☒ ByOne	Undo
☐ PtonElem	Y			☐ ByTwo	Create
Position	Z			☐ Translation	Reject
					Accept

Node creation by either specifying coordinates, selecting arbitrary point or positioning the point using the window shown

☒ NodePick	☒ PT1		Begin	End	Node Id: 6182 NewID	☐ ByOne	Undo
☐ PtonElem	☐ PT2		X			☒ ByTwo	Create
☐ TwoNode	Num	1	Y			☐ Translation	Reject
			Z			Accept	

Create nodes between two points

Create Position

☒ Global
 ☐ Local

☒ Position

☐ Node
☐ C_Element
☐ C_Edge
☐ C_Area
☐ 3Ptcircle

Positioning nodes

☒ Node
 ☐ PtOnElm

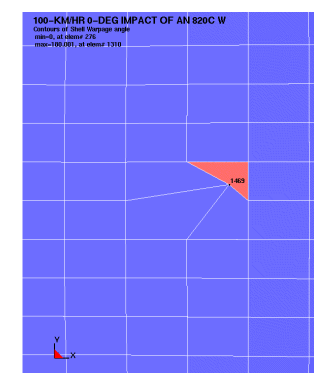
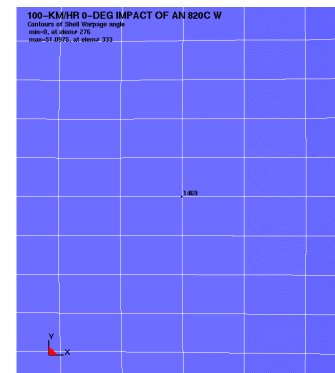
☒ Point1
 ☐ Point2

Align nodes between two points

☒ Pick Node

☒ Quality Check

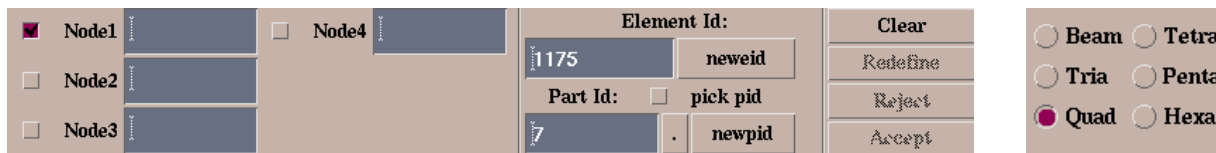
Modify position of nodes



Modifying nodes with element quality check

Element Edit - Element modification and creation

- ❖ Beams, shell and solid elements can be created using the interface shown above
 - Pick a new or existing PID and select the nodes to create an element
- ❖ Elements can be deleted easily using the interface
- ❖ Modifying of elements like splitting can be done using the different methods of splitting as shown

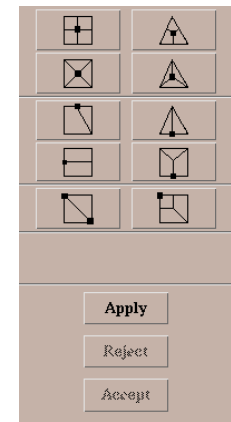


Node1 ☒ Node2 ☐ Node3 ☐ Node4 ☐

Element Id: 1175 neweid
Part Id: 7 pick pid newpid

Clear Redefine Reject Accept

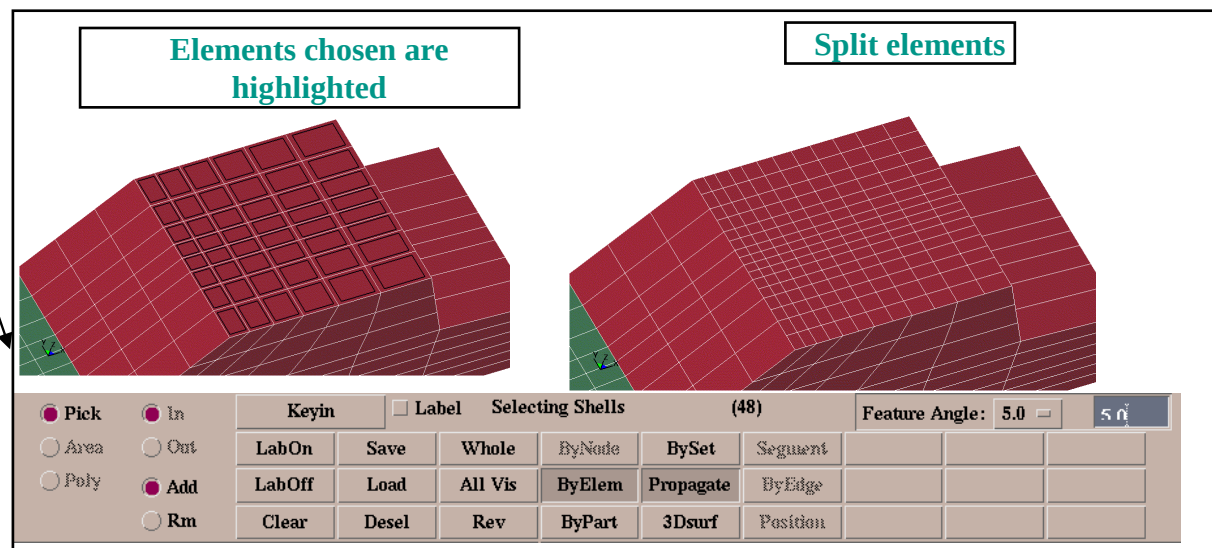
Beam Tetra
Tria Penta
Quad Hexa



Apply
Reject
Accept

Split elements
methods

1. Select the method you want to use to split the elements
2. Elements can be selected using the interface shown



Pre Processing Tools

Entity Operation Interface

This interface is used with many of the interfaces on Page 5

- ☐ Right working area allows the user to show, create, modify and delete entity by using list and bottom area. To different entities right area is used to input parameter and data.
- ☐ The bottom area is used to select entities from the drawing area
 - ❖ Show all, none, reverse or pick by area
 - ❖ Identity entity for modify and delete
 - ❖ Read entity keyword data using page3 and page4 form format by Kcard option

>Accelerometers Data
☒ Show ☐ Create
☐ Modify ☐ Delete

Label None

SBACID NewID

☐ Pick Ent ☒ In ☐ Out ☐ Partial ☐ Whole

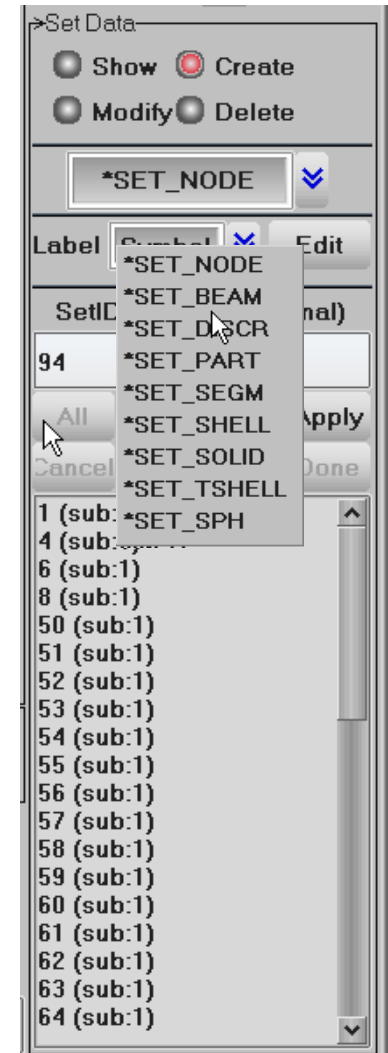
Entity Type: SBACceleratorometer

Operation: ☒ Showing ☐ Blank ☐ Ident ☐ KCard

LSDYNA data - SetD

Set data interface is used to show/create/modify/delete
LSDYNA *SET_ data

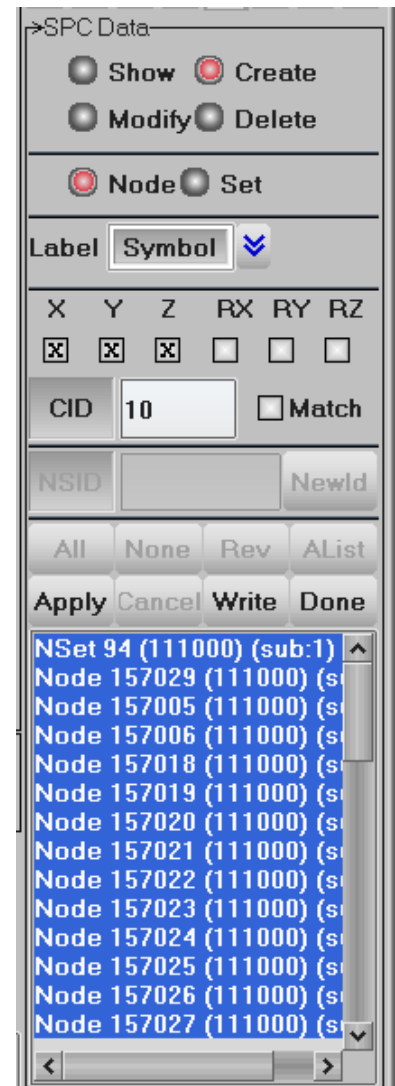
- ☐ Set data types – nodes, beam, discrete, part, segment, shell, solid, tshell, sph
- ☐ Show – show one or more sets graphically
- ☐ Create – create set data using general selection
- ☐ Modify – modify a set data
- ☐ Delete – delete a set data
- ☐ Edit – allow the attribute data to be edited
- ☐ Write – write *SET_ keyword data to file



LSDYNA data - SPC

SPC – to show/create/modify/delete *BOUNDARY_SPC data

- ☐ Show – show existing SPC data,
 - ❖ if local coordinate systems will also be shown if being used
 - ❖ Label – set to no label, symbol or details
 - ❖ Match – match the exact constrained DOF for showing
- ☐ Create – use general selection to select nodes
 - ❖ Node or Node Set
 - ❖ Set the constrained DOF
 - ❖ Use local coordinate system
- ☐ Modify and delete SPC data



SPC Data

☐ Show ☒ Create
☐ Modify ☐ Delete

☒ Node ☐ Set

Label: ▼

X	Y	Z	RX	RY	RZ
☒	☒	☒	☐	☐	☐

CID: ☐ Match

NSID: NewId:

All None Rev AList

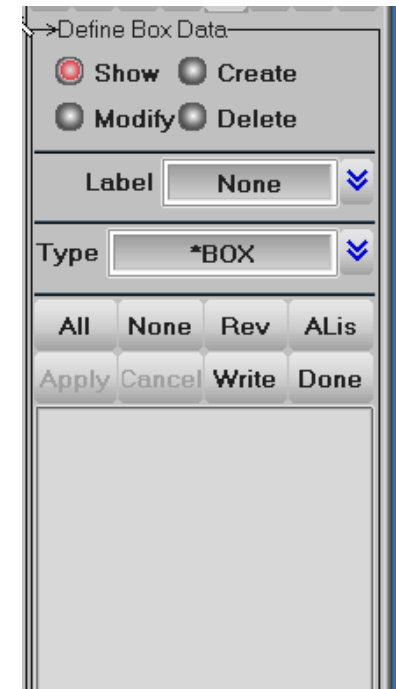
Apply Cancel Write Done

NSet 94 (111000) (sub:1) ▲
 Node 157029 (111000) (s)
 Node 157005 (111000) (s)
 Node 157006 (111000) (s)
 Node 157018 (111000) (s)
 Node 157019 (111000) (s)
 Node 157020 (111000) (s)
 Node 157021 (111000) (s)
 Node 157022 (111000) (s)
 Node 157023 (111000) (s)
 Node 157024 (111000) (s)
 Node 157025 (111000) (s)
 Node 157026 (111000) (s)
 Node 157027 (111000) (s) ▼

LSDYNA data - Box

Box – to show/create/modify/delete *DEFINE_BOX data

- ☐ Show – show existing boxes data
- ☐ Create – create new box
 - ❖ By min. and max.
 - ❖ By nodes in area or polygon
 - ❖ Can be translated or scaled
- ☐ Modify – change existing box
- ☐ Delete – delete existing boxes
- ☐ Write – write *Define_box data to file



Define Box Data

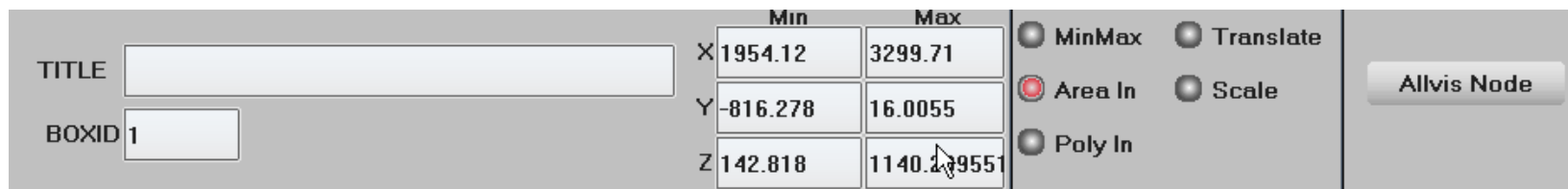
☒ Show ☐ Create
☐ Modify ☐ Delete

Label

Type

All None Rev ALis

Apply Cancel Write Done



TITLE

BOXID

	Min	Max
X	1954.12	3299.71
Y	-816.278	16.0055
Z	142.818	1140.29551

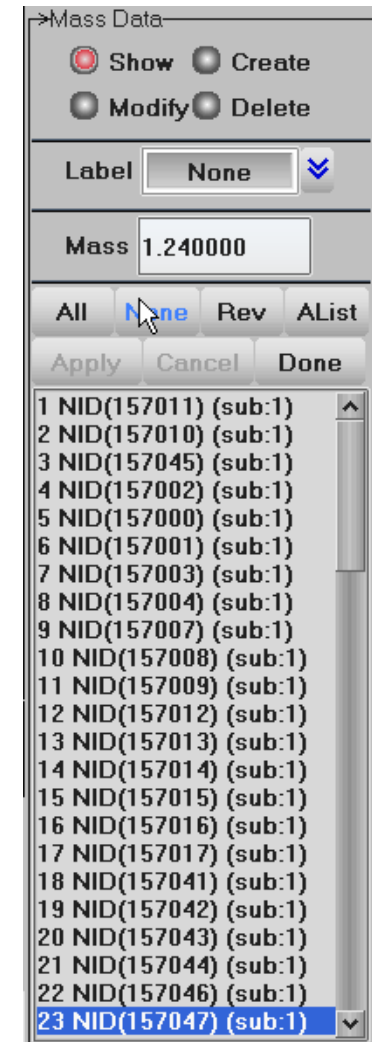
☐ MinMax ☐ Translate
☒ Area In ☐ Scale
☐ Poly In

Allvis Node

LSDYNA data – Mass Element

MassD – to show/create/modify/delete mass element data

- ☐ Show – show existing mass element with its value
- ☐ Create – create new mass elements
 - ❖ Select nodes in General selection
 - ❖ Assign mass value
- ☐ Modify – change existing mass element
- ☐ Delete – delete existing mass elements



The image shows a 'Mass Data' dialog box from the LS-PREPOST software. It has a title bar 'Mass Data' and four radio buttons: 'Show' (selected), 'Create', 'Modify', and 'Delete'. Below these are two input fields: 'Label' with a dropdown menu showing 'None' and a 'Mass' field with the value '1.240000'. There are three buttons: 'All', 'None' (highlighted with a mouse cursor), and 'Rev', followed by 'AList'. At the bottom are 'Apply', 'Cancel', and 'Done' buttons. A list box contains 23 entries, each representing a node and its mass: '1 NID(157011) (sub:1)', '2 NID(157010) (sub:1)', '3 NID(157045) (sub:1)', '4 NID(157002) (sub:1)', '5 NID(157000) (sub:1)', '6 NID(157001) (sub:1)', '7 NID(157003) (sub:1)', '8 NID(157004) (sub:1)', '9 NID(157007) (sub:1)', '10 NID(157008) (sub:1)', '11 NID(157009) (sub:1)', '12 NID(157012) (sub:1)', '13 NID(157013) (sub:1)', '14 NID(157014) (sub:1)', '15 NID(157015) (sub:1)', '16 NID(157016) (sub:1)', '17 NID(157017) (sub:1)', '18 NID(157041) (sub:1)', '19 NID(157042) (sub:1)', '20 NID(157043) (sub:1)', '21 NID(157044) (sub:1)', '22 NID(157046) (sub:1)', and '23 NID(157047) (sub:1)' (which is highlighted).

LSDYNA data - CNRB

CNRB – to show/create/modify/delete

*constrained_nodal_rigid_body

- ☐ Show – show existing nodal_rigid_body data
- ☐ Create – create new nodal_rigid_body data
 - ❖ Select nodes in General selection
 - ❖ Assign new part Id and Set_node Id
- ☐ Modify – change existing nodal_rigid_body data
- ☐ Delete – delete existing nodal_rigid_body data

→CNRB Data

☒ Show
 ☐ Create
 ☐ Modify
 ☐ Delete

Label ▾

PID	CID	NSID
2		96

☐ Pick PNODE

PNODE

IPRT ▾

DRFLAG ▾

RRFLAG ▾

part(1).NSET95 (sub:1)

LSDYNA data – Initial Velocity

Inivel – to show/create/modify/delete

*Initial_velocity data

- ☐ Show – show existing initial velocity data
- ☐ Create – create new initial_velocity data
 - ❖ Select nodes using general selection
- ☐ Modify – change existing initial_velocity data
- ☐ Delete – delete existing initial_velocity data

Initial Velocity Data

☒ Show ☐ Create
☐ Modify ☐ Delete

Label ▼

Vx	Vy	Vz
2.9		
Vrx	Vry	Vrz

All None Rev
AList Apply Done

160000 (sub:1)
160001 (sub:1)
160002 (sub:1)
160003 (sub:1)
160004 (sub:1)
160005 (sub:1)
160006 (sub:1)
160007 (sub:1)
160008 (sub:1)
160009 (sub:1)
160010 (sub:1)
160011 (sub:1)
160012 (sub:1)
160013 (sub:1)
160014 (sub:1)
160015 (sub:1)
160016 (sub:1)
160017 (sub:1)

LSDYNA data – Spotweld/Rivet

Spweld/Rivet – to show/create/modify/delete

*Constrained_Spotweld and *Constrained_Rivet data

- ☐ Show – show existing spotweld/rivet data
- ☐ Create – create new spotweld/rivet data
 - ❖ Autocreate on will create without click apply
 - ❖ Assign mass value
- ☐ Modify – change existing spotweld/rivet data
- ☐ Delete – delete existing spotweld/rivet data

Rivet Data

☐ Show
 ☒ Create
 ☐ Modify
 ☐ Delete

Label
☒ Autocreate

RID ☒ N1 ☐ N2

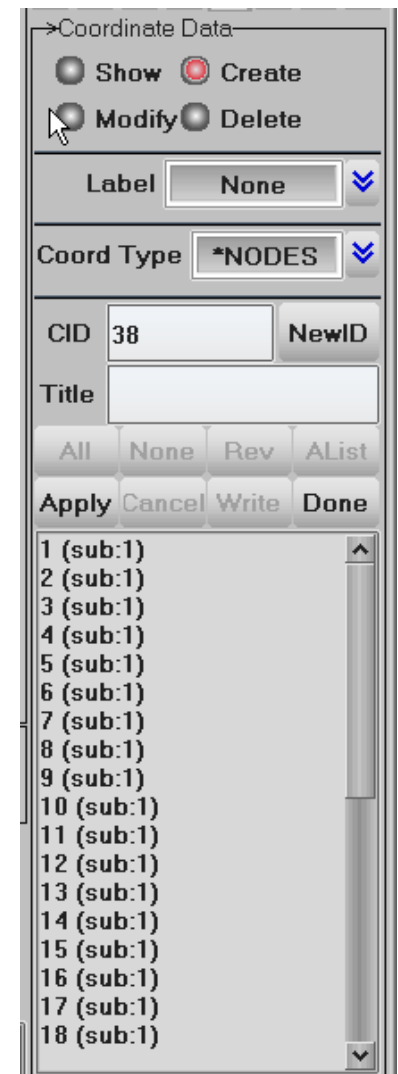
TF

157034, 157036 (sub:1)
 157039, 157031 (sub:1)
 157015, 157005 (sub:1)

LSDYNA Data – Coordinate System

Coord – to show/create/modify/delete *Define_coordinate data

- ☐ Show – show existing define_coordinate data
- ☐ Create – create new define_coordinate data
 - ❖ NODES - 3 Nodes form a system
 - ❖ SYSTEM – 3 position form a coordinate system
 - ❖ VECTOR – a direction
 - ❖ Title – a title can be add to the data
- ☐ Modify – change existing define_coordinate data
- ☐ Delete – delete existing define_coordinate data



Coordinate Data

☐ Show ☒ Create
☒ Modify ☐ Delete

Label

Coord Type

CID NewID

Title

All None Rev AList

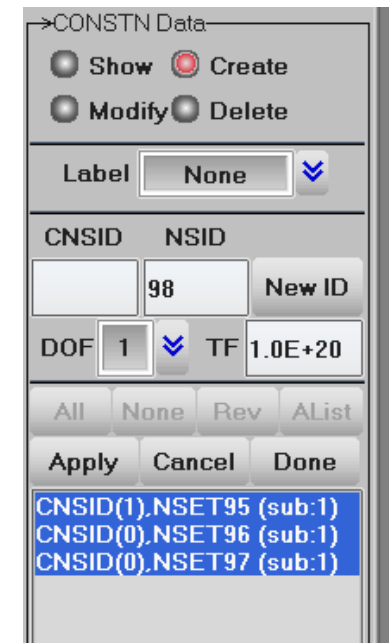
Apply Cancel Write Done

1 (sub:1)
2 (sub:1)
3 (sub:1)
4 (sub:1)
5 (sub:1)
6 (sub:1)
7 (sub:1)
8 (sub:1)
9 (sub:1)
10 (sub:1)
11 (sub:1)
12 (sub:1)
13 (sub:1)
14 (sub:1)
15 (sub:1)
16 (sub:1)
17 (sub:1)
18 (sub:1)

LSDYNA Data – Constrained_node_set

Constn – to show/create/modify/delete *Constrained_node_set data

- ☐ Show – show existing Constrained_node_set data
- ☐ Create – create new Constrained_node_set data
 - ❖ Select nodes by general selection
 - ❖ Provide Set_node ID
 - ❖ Set DOF with 1 to 7
 - ❖ Set TF – Failure time
- ☐ Modify – change existing Constrained_node_set data
- ☐ Delete – delete existing Constrained_node_set data



CNSID	NSID	DOF	TF
	98	1	1.0E+20

CNSID(1).NSET95 (sub:1)
 CNSID(0).NSET96 (sub:1)
 CNSID(0).NSET97 (sub:1)

LSDYNA Data – Generalize_weld

Gweld – to show/create/modify/delete

*Constrained_generalize_weld_spot data

- ☐ Show – show existing Constrained_node_set data
- ☐ Create – create new Constrained_node_set data
 - ❖ Select nodes by general selection
 - ❖ Provide Set_node ID
 - ❖ Set DOF with 1 to 7
 - ❖ Set TF – Failure time
- ☐ Modify – change existing Constrained_node_set data
- ☐ Delete – delete existing Constrained_node_set data

Genweld Data

☒ Show
 ☐ Create
 ☐ Modify
 ☐ Delete

Label ▾

BackList

WID	NSID	NewID
	98	

CID	
FILTER	
WINDOW	0
NPR	
NPRT	0 ▾
TFAIL	1.0E+20
EPSF	
SN	
SS	
N	
M	

LSDYNA Data - Seatbelt Accelerometer

- ❑ Show, create, modify and delete Seatbelt Accelerometer
 - ❖ Create seatbelt accelerometer by input or pick node from graphic
 - ❖ Check node is rigid node
 - ❖ Modify and delete by selecting seatbelt accelerometer from list or pick from graphic by bottom entity interface
 - ❖ Display seatbelt accelerometer by coordinate system.

☒	NID1	0	IGRAV	0	
☐	NID2	0	INTOPT	0	
☐	NID3	0			

Accelerometers Data

☐ Show
 ☒ Create
☐ Modify
 ☐ Delete

Label None ▼

SBACID 17 NewID

1 (sub:1)

2 (sub:1)

3 (sub:1)

4 (sub:1)

5 (sub:1)

6 (sub:1)

7 (sub:1)

8 (sub:1)

9 (sub:1)

10 (sub:1)

11 (sub:1)

12 (sub:1)

13 (sub:1)

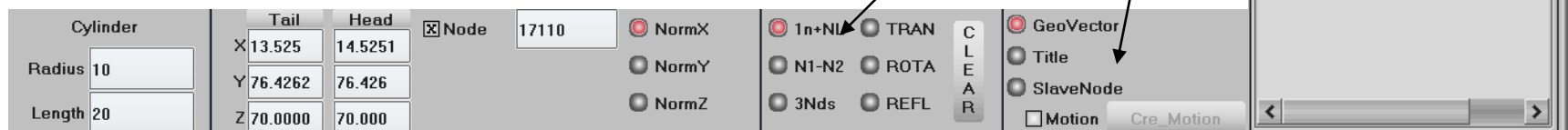
14 (sub:1)

15 (sub:1)

16 (sub:1)

LSDYNA Data - RigidWall

- ❑ Show, create, modify and delete Rigid Wall
 - ❖ There are Geometry and Planar
 - ❖ Geometry include cylinder, flat, prism and sphere
 - ❖ Planar have finite, forces, moving and ortho options
 - ❖ Creation and modify can be preview and the wall can be translation, rotation and reflection.
 - ❖ By change bottom working area to create slave node and input more parameters



The screenshot shows the 'RigidWall Data' dialog box. It has tabs for 'Show', 'Create', 'Modify', and 'Delete'. The 'Create' tab is active, showing options for 'geometry' (selected) and 'planar'. Under 'geometry', 'Cylind' is selected. The 'Label' is set to 'None'. Below are buttons for 'All', 'None', 'Rev', and 'AList', and 'Apply', 'Cancel', and 'Done'. A list of wall types is shown: 1 GEOMETRIC_CYLINDER, 2 GEOMETRIC_CYLINDER, 3 GEOMETRIC_FLAT (sub:1), 4 GEOMETRIC_PRISM (sub:1), 5 GEOMETRIC_SPHERE (sub:1), 6 PLANAR (sub:1), 7 PLANAR_FINITE (sub:1), 8 PLANAR_ORTHO_FINITE.

Annotations with arrows point to specific fields:

- 'Change bottom working area' points to the 'Node' field (17110).
- 'Modify preview Rigid wall' points to the 'GeoVector' field.

Below the dialog box, a 'Cylinder' configuration panel is visible with fields for Radius (10), Length (20), Tail (X: 13.525, Y: 76.4262, Z: 70.0000), Head (X: 14.5251, Y: 76.426, Z: 70.000), and NormX, NormY, NormZ. There are also checkboxes for 'Node' and 'Motion', and a 'Cre_Motion' button.

Database Cross Section

- ❑ Show, create, modify and delete Database Cross Section
 - ❖ Show database cross section
 - ❖ Create database cross section using bottom working area and preview
 - ❖ Modify by translation, rotation and reflection database cross section and preview
 - ❖ By change bottom working area to create slave node and input more parameters
 - ❖ Delete database cross section by selecting from list and pick from graphic.

XXSection Data

☐ Show ☒ Create
☐ Modify ☐ Delete

☒ Plane ☐ Set

Label ▾

1 - 1 (sub:1)
 2 - 2 (sub:1)
 3 - 3 (sub:1)
 4 - 4 (sub:1)

CSID ITYPE ▾

Title

PSID Cre_PSet

XV YV ZV
 LENL LENM
 0.0 0.0

Tail Head
 X
 Y
 Z

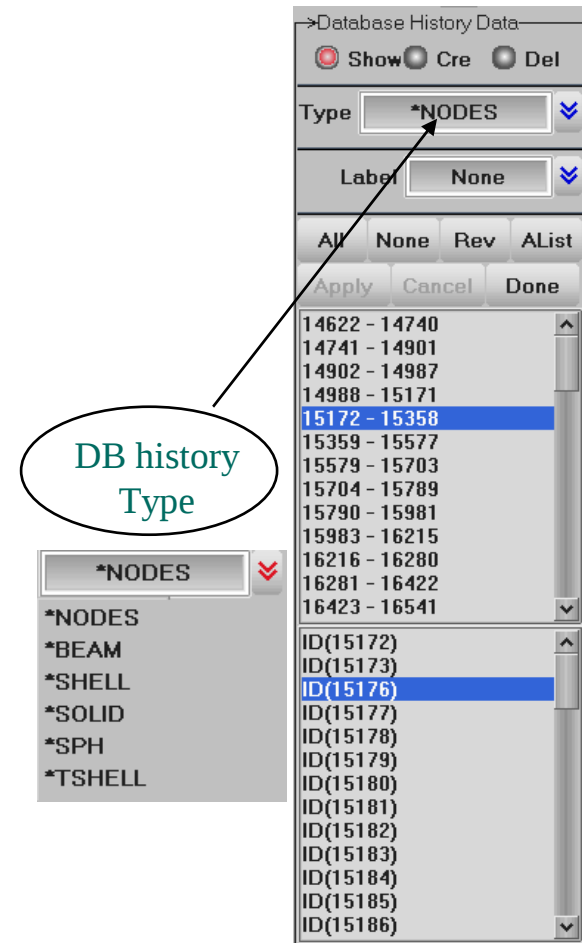
☒ Node

☒ NormX ☐ 1n+NL ☐ TRAN
☐ NormY ☐ N1-N2 ☐ ROTA
☐ NormZ ☐ 3Nds ☐ REFL

Database History

❑ Show, create and delete Database History

- ❖ There are nodes, beam, shell, solid, sph and tshell database history
- ❖ Create database history using bottom working area pick nodes or same type elements
- ❖ Delete database history by selecting from list



Tied Nodes Failure

- ❑ Show, create, modify and delete Tied Nodes Failure
 - ❖ Show Constrained Tied Nodes.
 - ❖ Create Constrained tied nodes by pick node from graphic using general selection
 - ❖ Modify and delete constrained tied nodes by selecting from list or picking from graphics

→TiedNF Data

☐ Show ☒ Create
☐ Modify ☐ Delete

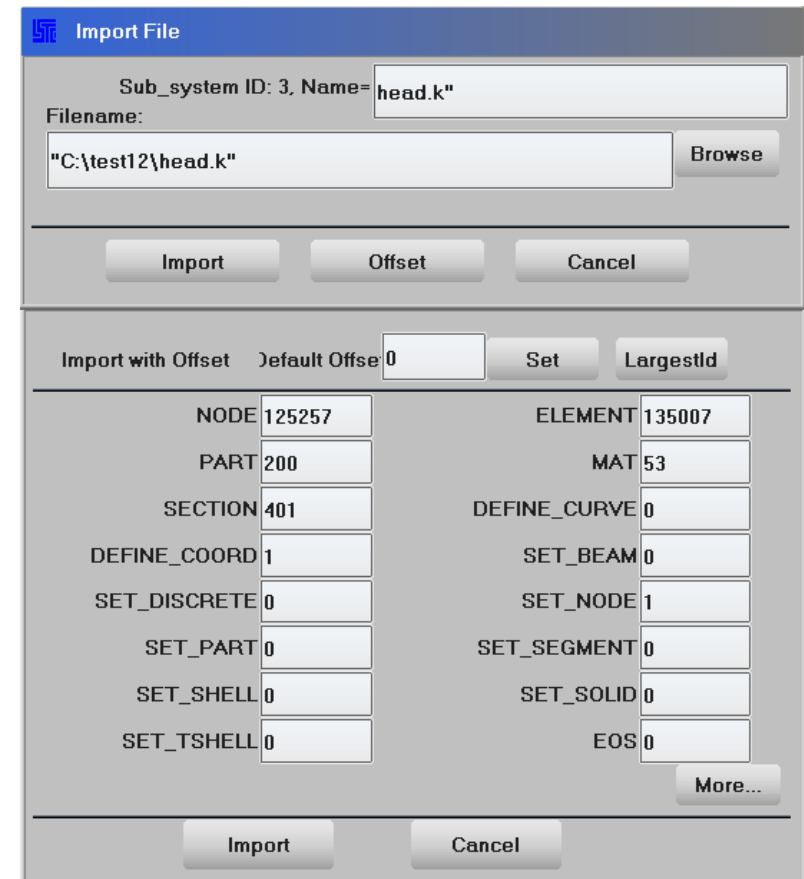
Label ▾

NID NewId
 NSID NewId
 EPPF

1, NSET1(4) 0
 2, NSET2(4) 0
 3, NSET3(4) 0
 4, NSET4(4) 0
 5, NSET5(4) 0
 6, NSET6(2) 0
 7, NSET7(6) 0
 8, NSET8(4) 0
 9, NSET9(4) 0
 10, NSET10(4) 0
 11, NSET11(4) 0
 12, NSET12(4) 0
 13, NSET13(4) 0
 14, NSET14(4) 0
 15, NSET15(4) 0
 16, NSET16(4) 0
 17, NSET17(4) 0
 18, NSET18(4) 0
 19, NSET19(2) 0

Input keyword

- ☐ Input keyword
- ☐ Import more keyword file
define each import
keyword for one
subsystem.
- ☐ Offset import file
keyword in. Set offset to
largest id or input value
by each keyword



Import File

Sub_system ID: 3, Name= head.k"

Filename: "C:\test12\head.k" Browse

Import
Offset
Cancel

☒ Import with Offset
 Default Offset: 0
Set LargestId

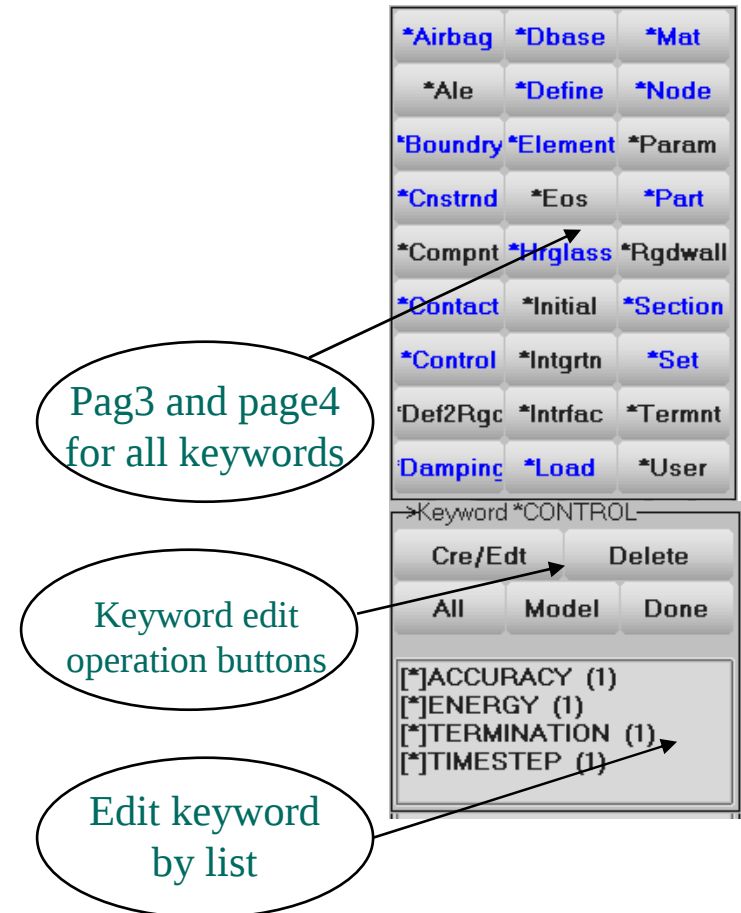
NODE 125257 PART 200 SECTION 401 DEFINE_COORD 1 SET_DISCRETE 0 SET_PART 0 SET_SHELL 0 SET_TSHELL 0	ELEMENT 135007 MAT 53 DEFINE_CURVE 0 SET_BEAM 0 SET_NODE 1 SET_SEGMENT 0 SET_SOLID 0 EOS 0
--	--

More...

Import
Cancel

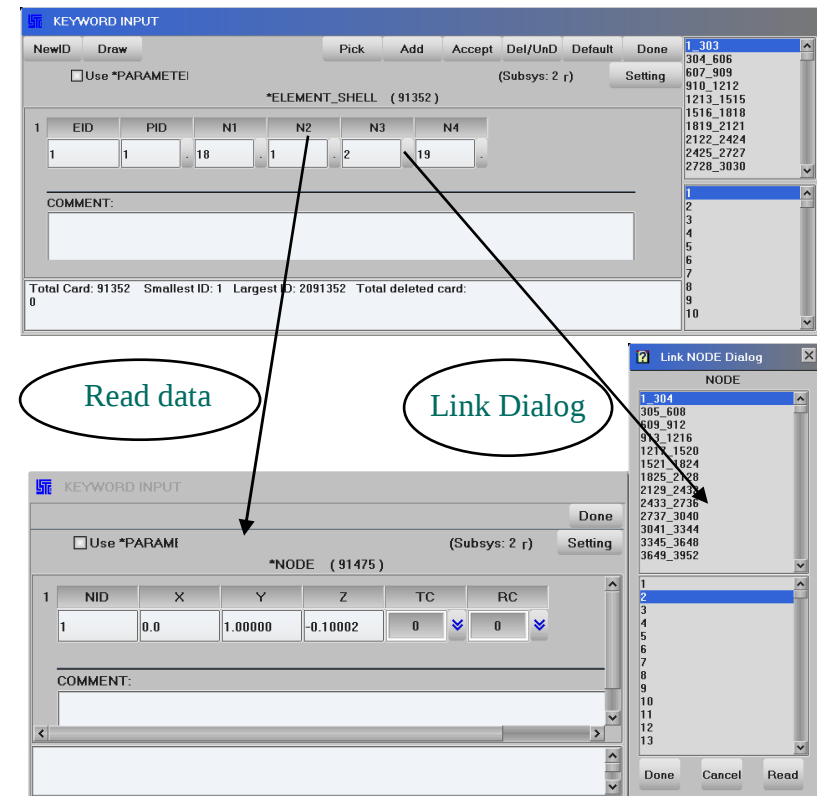
LSDYNA Keyword data

- ❑ Each keyword has its own form for input and editing
- ❑ Keyword data that is present in the model will be highlighted with blue color
- ❑ There are over 800 keyword entities
- ❑ Comment cards, title card and Parameters are available for each keyword input
- ❑ All keyword can be edit by name



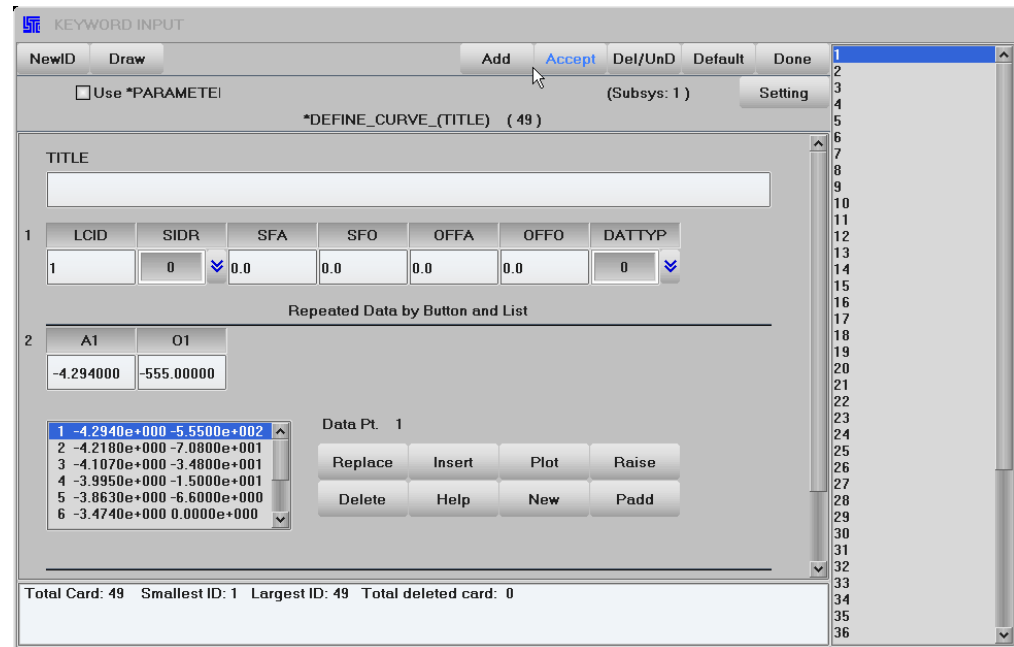
Keyword Edit - Feature

- ❑ Keyword Selection Menu
- ❑ CONTROL Extra Keyword Options
- ❑ Ref By Interface and save ref to buffer
- ❑ Keyword Input Form
- ❑ Link Keyword Dialog
- ❑ Pick interface
- ❑ Dynamic Keyword Input Forms
- ❑ Entity Draw Option
- ❑ Input data by table
- ❑ Transfer Material data from Material Database
- ❑ Plot curve and sort section data



Keyword Edit - Form

- ❑ Keyword input forms match LS-DYNA manual
- ❑ Each data field is identified by its name
- ❑ The description of the field is shown with a simple click in the field or the field name



KEYWORD INPUT

NewID Draw Add Accept Del/UnD Default Done

☐ Use *PARAMETER (Subsys: 1) Setting

*DEFINE_CURVE_(TITLE) (49)

TITLE

LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP
1	0	0.0	0.0	0.0	0.0	0

Repeated Data by Button and List

A1	O1
-4.294000	-555.00000

1 -4.2940e+000 -5.5500e+002
 2 -4.2180e+000 -7.0800e+001
 3 -4.1070e+000 -3.4800e+001
 4 -3.9950e+000 -1.5000e+001
 5 -3.8630e+000 -6.6000e+000
 6 -3.4740e+000 0.0000e+000

Data Pt. 1

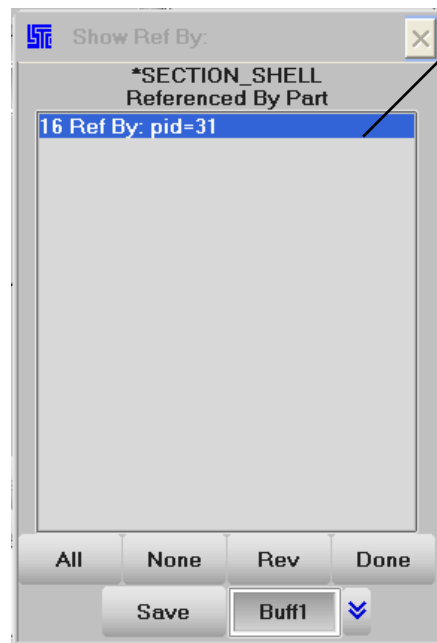
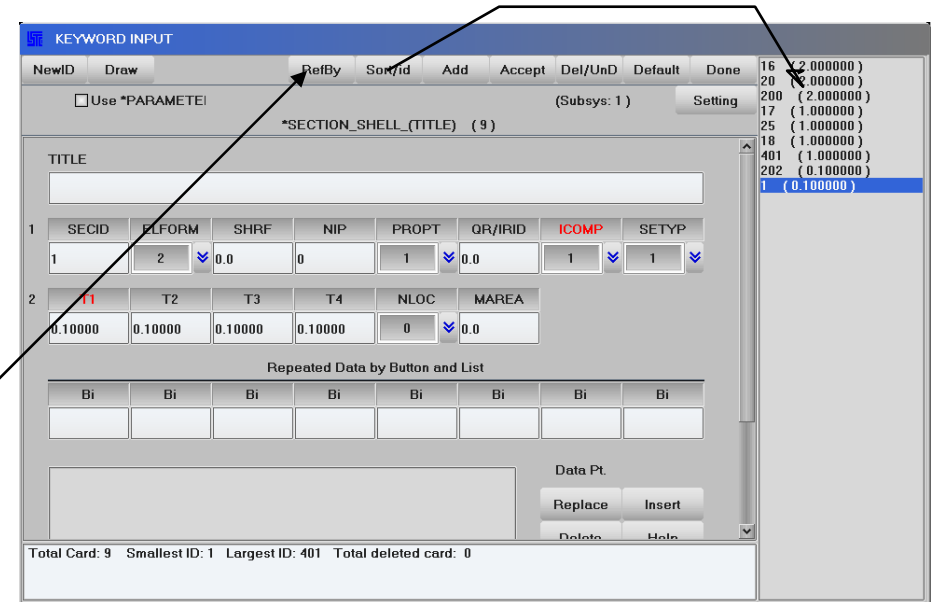
Replace Insert Plot Raise
Delete Help New Padd

Total Card: 49 Smallest ID: 1 Largest ID: 49 Total deleted card: 0

- ❑ Simple selection button is used for the data field with pre-defined values
- ❑ A popup table can be used as an aid to transfer data to the selected field
- ❑ Link data can be viewed with a click on the name

Keyword Edit - Sort and Reference

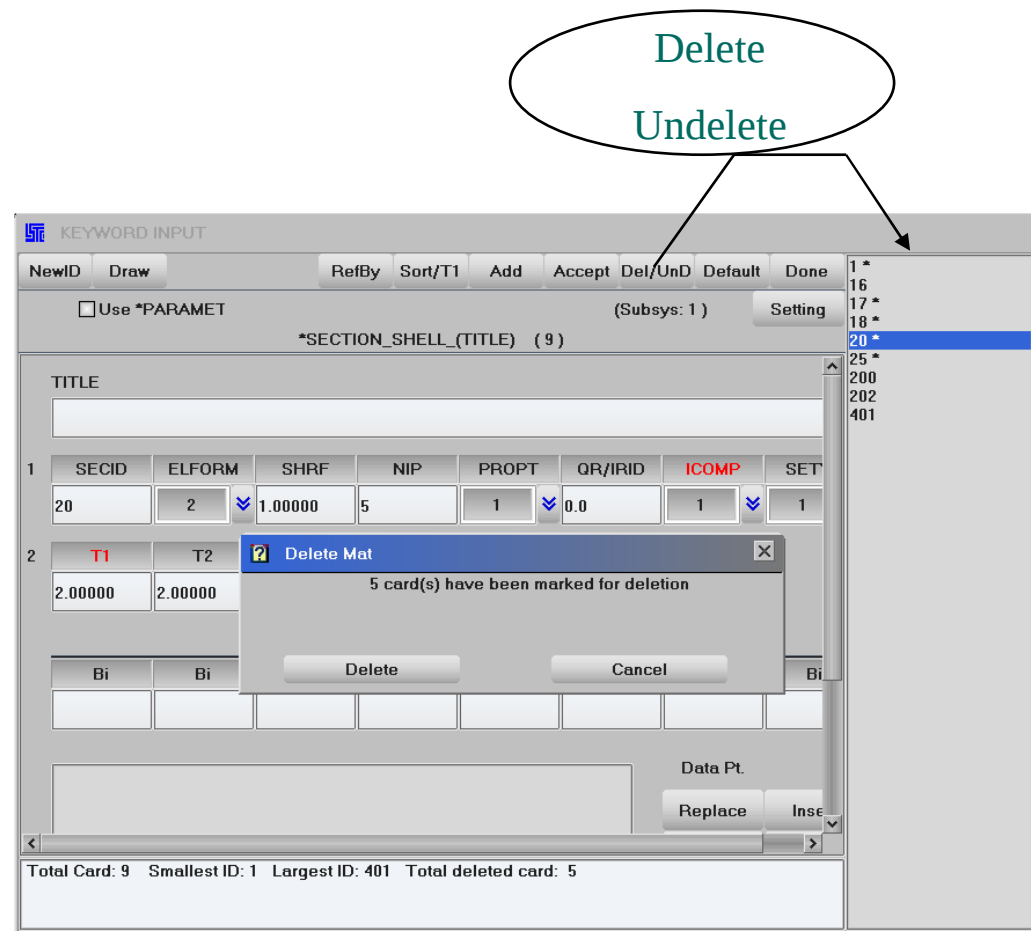
- ❑ Keyword input forms handle all special case
- ❑ Highlight key field name to change form when the value be changed

- ❑ Keyword input forms handle all special case
- ❑ Reference check, list reference by part and save to buff

Keyword Edit – Delete

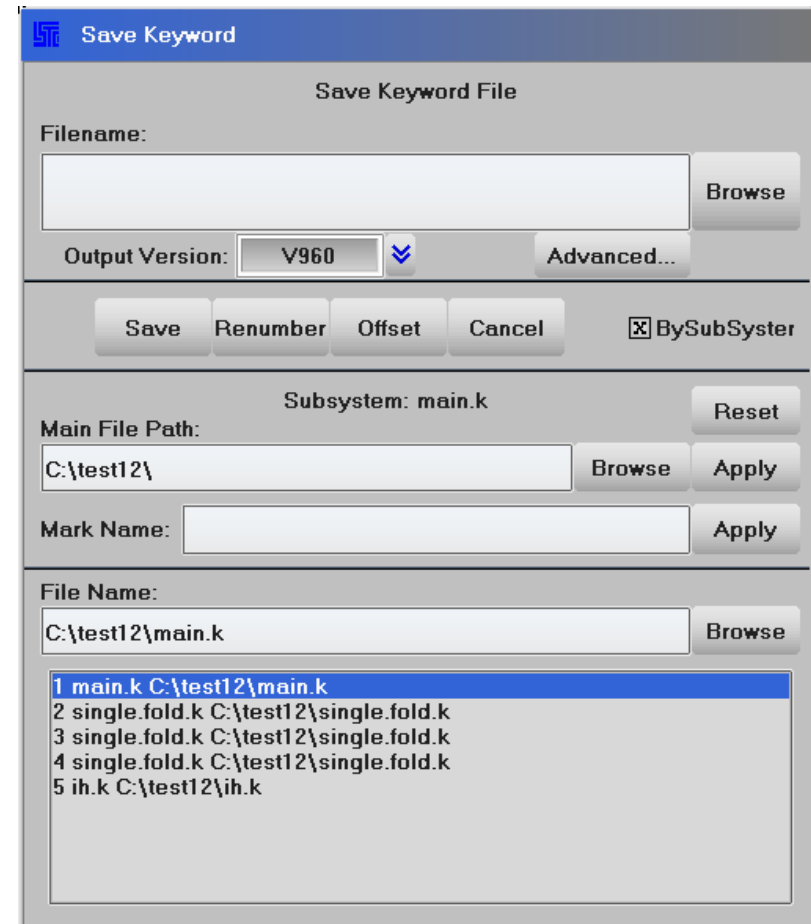
- ❑ Delete keyword by make mark from list.
One more time check when delete them
- ❑ Undelete keyword by remove mark fro list
- ❑ One more time check when delete them



Save Keyword File

Save keyword data into file

- ☐ Save in 960, 970, and 971 format
- ☐ Advanced options
 - ❖ Output keyword data in users' own sequence
 - ❖ Output keyword data selectively
 - ❖ Output keyword with title
 - ❖ Output parameter names instead of actual data
 - ❖ Output field names
 - ❖ Output comments
- ☐ Renumber data Ids
- ☐ Offset data Ids
- ☐ Output subsystems in different files



The dialog box is titled "Save Keyword" and "Save Keyword File". It contains the following fields and controls:

- Filename:** A text input field with a "Browse" button to its right.
- Output Version:** A dropdown menu currently showing "V960" and a downward arrow. To its right is an "Advanced..." button.
- Buttons:** "Save", "Renumber", "Offset", and "Cancel". To the right of these is a checkbox labeled "BySubSystem" which is checked.
- Subsystem:** A label "Subsystem: main.k" with a "Reset" button to its right.
- Main File Path:** A text input field showing "C:\test12\" with a "Browse" button to its right and an "Apply" button to the right of the "Browse" button.
- Mark Name:** A text input field with an "Apply" button to its right.
- File Name:** A text input field showing "C:\test12\main.k" with a "Browse" button to its right.
- List:** A list box containing the following items:
 - 1 main.k C:\test12\main.k
 - 2 single.fold.k C:\test12\single.fold.k
 - 3 single.fold.k C:\test12\single.fold.k
 - 4 single.fold.k C:\test12\single.fold.k
 - 5 ih.k C:\test12\ih.k

Save Active Keyword

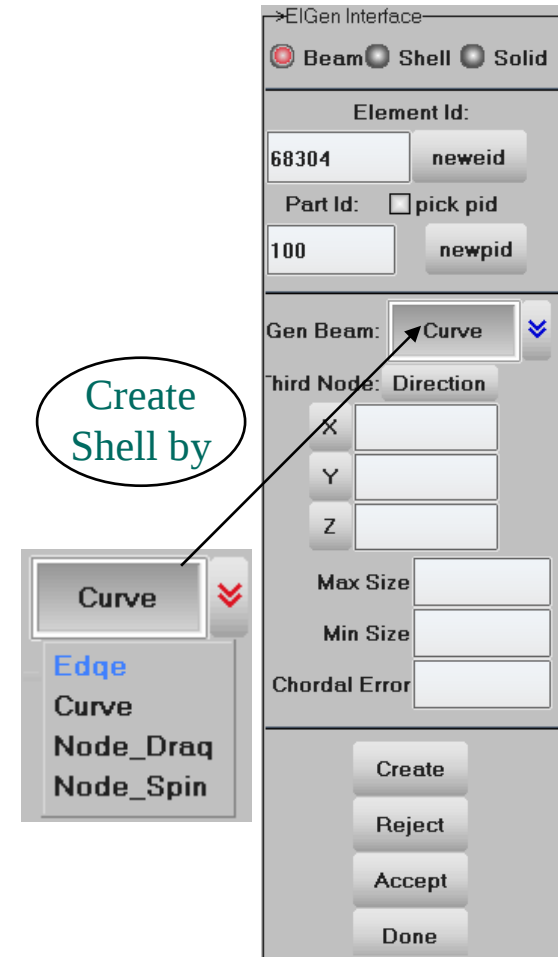
Save Active keyword data into file

- ☐ Save all active nodes, element and parts
- ☐ Save all parts section, material, eos and hourglass datas.
- ☐ Save all control, database datas
- ☐ Save all active parts set
- ☐ Save all constrained rigid body and airbag which include active part and part set
- ☐ Save above data reference data
- ☐ Save more reference data
 - ❖ Define curves reference data
 - ❖ Initial strain data
 - ❖ Initial stress beam data
 - ❖ Initial stress shell data
 - ❖ Initial stress solid
- ☐ Move active keyword to target subsystem

Element Generation - Beam

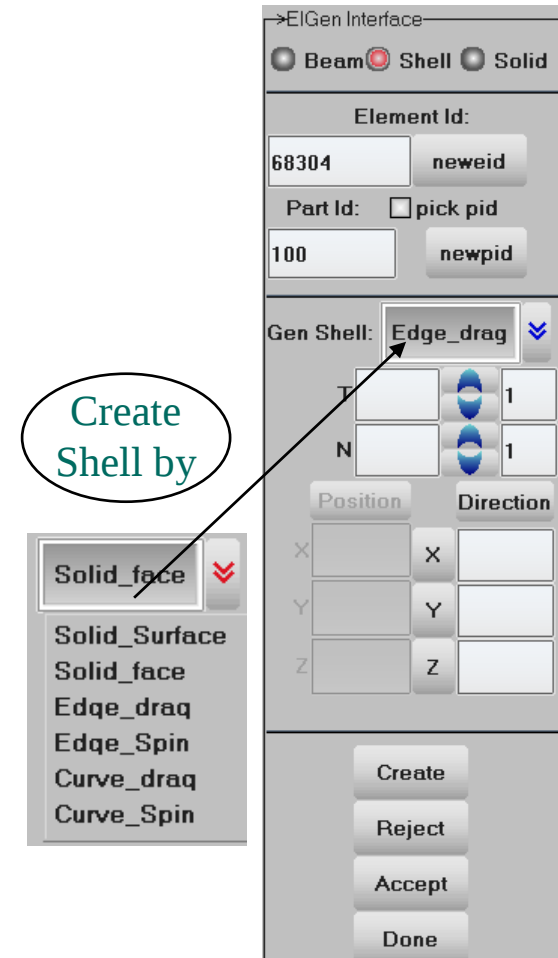
Element generation is an interface for creating beams, shells and solids

- ❑ Beam elements can be created by selecting an edge, a curve or create nodes using either drag or spin feature
 - ❖ Select an existing node or create a node using the ELEDIT feature
 - ❖ T- Element length, N- number of elements in given length
 - ❖ Select the direction in either global or local system and create beams
 - ❖ For beam creation using node spin an angle and the number of segments on the angle needs to be given



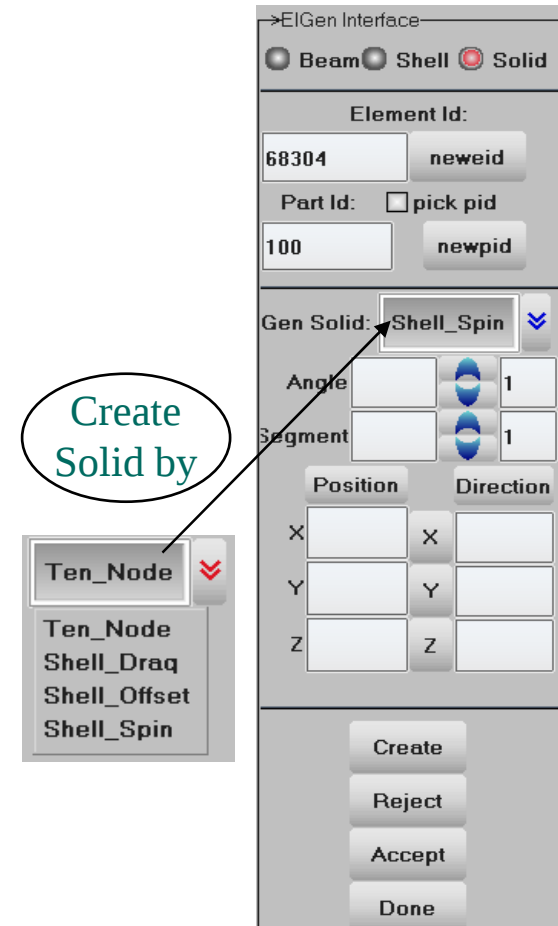
Element Generation - Shell

- ❑ Shell elements can be created by selecting an edge, a curve, solid element faces or surfaces
 - ❖ By selecting solid surfaces the entire solid surface can be chosen and shell elements created
 - ❖ By selecting solid faces, only a single face of the solid entity is selected
 - ❖ By selecting an edge an entire edge is selected. The edge selection can be enhanced by selecting an appropriate feature angle to include all features of the edge.
 - ❖ Curves can be created and used by either dragging or spinning to create the shell element surface



Element Generation - Solid

- ❑ Solid elements can be created by ten node, shell drag, offset and spin
 - ❖ Create ten node solid by selecting 4 node solid.
 - ❖ By selecting a shell surface the elements can be dragged in a particular direction locally or in global systems
 - ❖ Solids can also be created using offset in shells
 - ❖ Solids can be created using a spin axis and direction



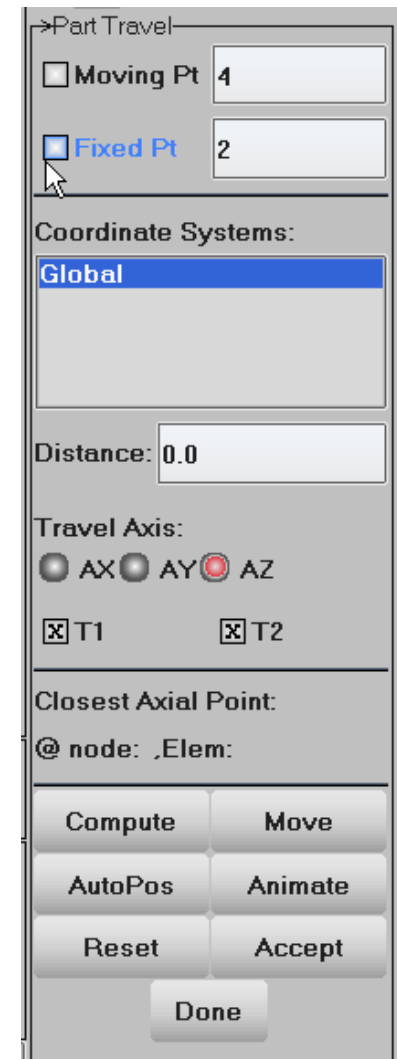
LS-PREPOST

Applications

Metal Stamping – Part Travel

Purpose: To measure the distance
between two parts

- ☐ Distance can be measured in global or local direction
- ☐ Thickness from each part can be ignored or considered
- ☐ After distance is computed, the moving part can be automatically positioned



Part Travel

☐ Moving Pt 4

☒ Fixed Pt 2

Coordinate Systems:

Global

Distance: 0.0

Travel Axis:

☐ AX ☐ AY ☒ AZ

☒ T1 ☒ T2

Closest Axial Point:

@ node: ,Elem:

Compute Move

AutoPos Animate

Reset Accept

Done

Metal Stamping - PTRIM

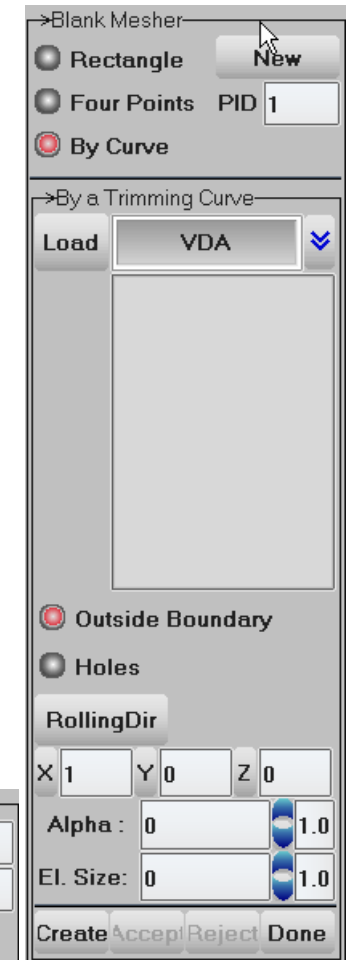
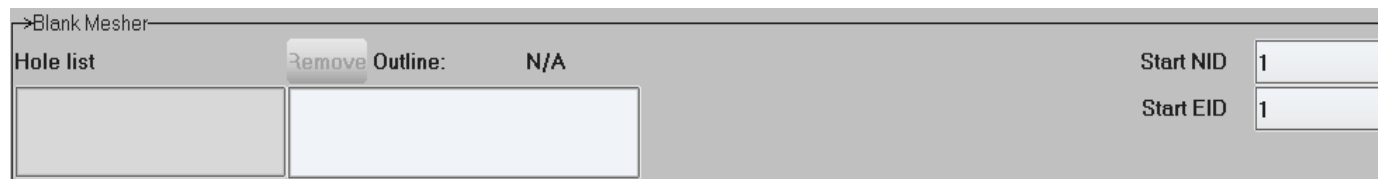
• Part Trimming

- ☐ Trimming a given part with open curves.
 - ❖ Trimming algorithms taking care of open curves.
 - ❖ Several open curves are allowed to trim at the same time.
 - ❖ Seed nodes can be provided to keep/remove the needed region.
- ☐ Trimming algorithm will take care of the adapted elements as well as element thickness and initial stress/strains during the trimming process.



Metal Stamping - BMesh

- Blank mesher : Creating a blank mesh for metal forming simulations by using part trim
- Three available methods for creating blanks
 - ☐ Rectangle : create rectangle blanks by giving dimensions and desired element size or number of elements
 - ☐ Four points : create blanks by giving 4 points in the space and the desired element size
 - ☐ By Curve : create blanks by giving outlines of the blank and desired element size



Wall Creation for Binder

Purpose:

Selects the edges of the binder to create wall.

Giving: Angle, Length, Element segment and Element normal or Direction

Binder Wall Data

☐ Pick

angle

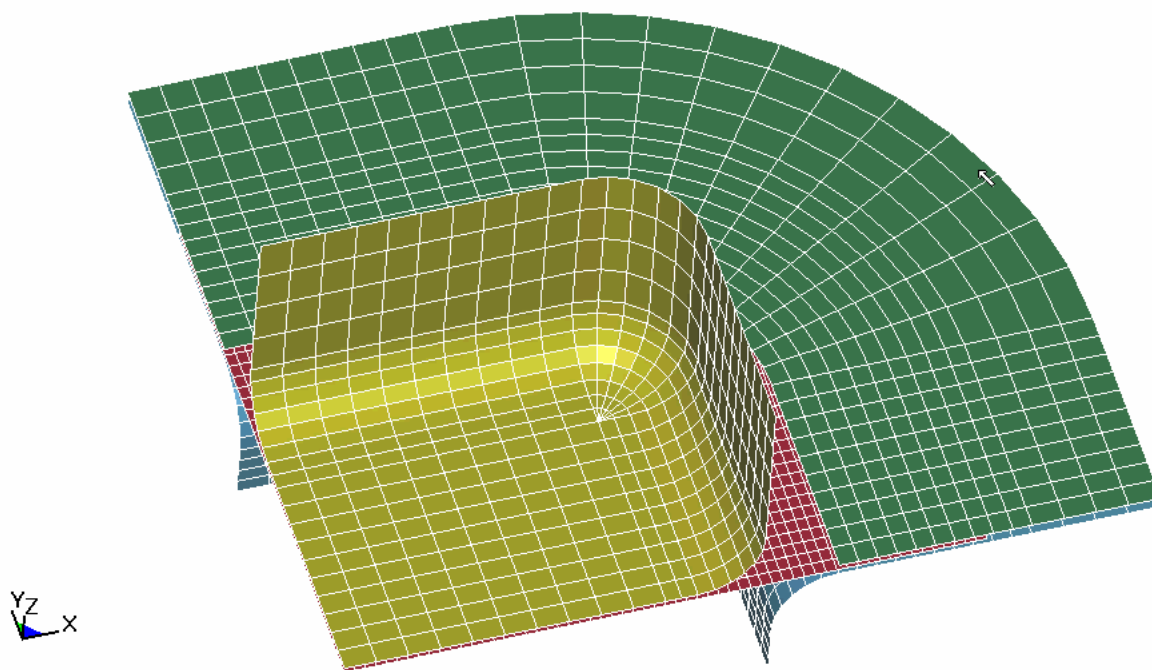
☒ EI_normal ☐ Direction

length

Segment

Wall Creation for Binder

Rectangular Cup Drawing



Guide Building

Purpose:

Selects a direction and two positions to create Guide.

Giving angle, length and below percent.

→Guide Building Data

☐ Pick Partid

6 newpid

40417 Neweid

Pos1 Pos2

X 89 114

Y 20.4842 20.4842

Z 2 2

Direction

dirx 0

diry 0

dirz 1

Angle 15 5

Length 40 5

below 10 5

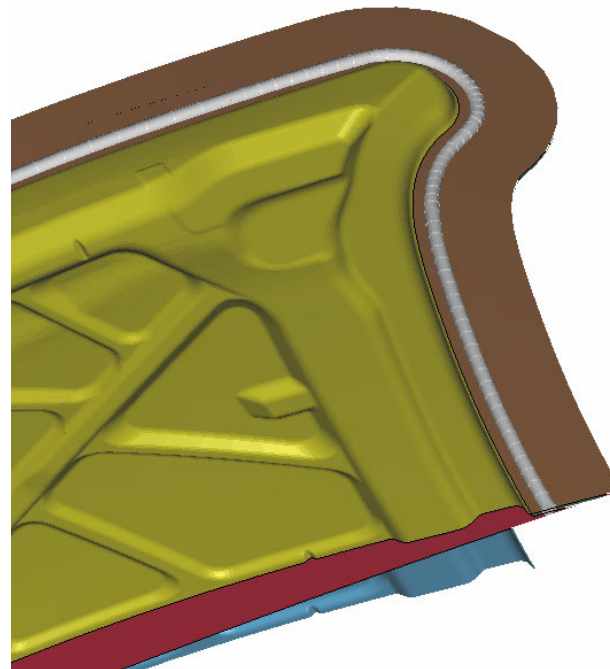
Build Reject

Accept Done

Metal Stamping – Draw Bead

To show/create/modify/delete drawbead data

- ☐ Create by beam part, or by node set, or by curves
- ☐ Automatically create all necessary keyword data cards
- ☐ Drawbead is shown as pipe with actual depth



>Drawbead Data:

☒ Show ☐ Create
☐ Modify ☐ Delete

SSTYPE BeamPart

☒ Pick PBeam
☐ Pick MSID
☐ Pick Dield
☐ Pick Boxid 9

☐ Creat Apply

Depth LCIDRF LCIDNF
LCEPS2 OFFSET LCEPS

Show All

All None Rev AList

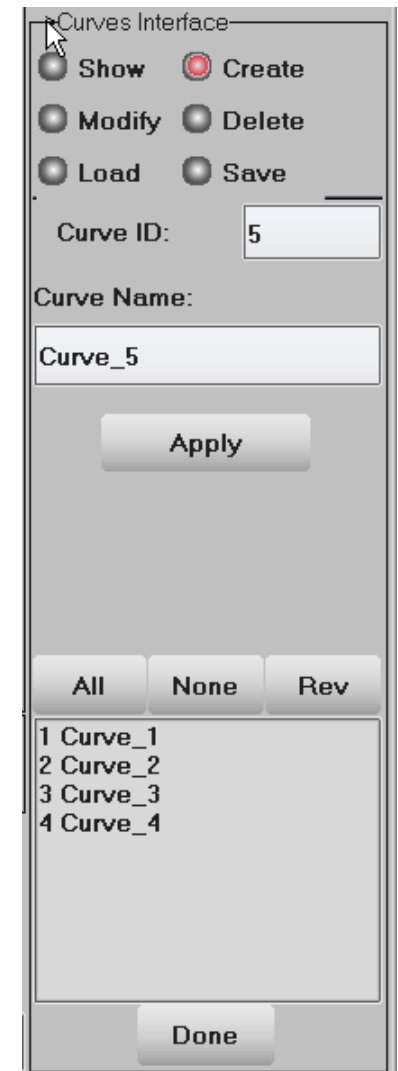
Apply Card Done

1 - 5 (sub:1)
2 - 6 (sub:1)
3 - 7 (sub:1)
4 - 8 (sub:1)
5 - 9 (sub:1)
6 - 10 (sub:1)
7 - 11 (sub:1)

Metal Stamping – Curves

To show/create/modify/delete curves data

- ☐ A curve is a series of points connect together
- ☐ Points can be created by general selection
- ☐ Save – save curves data to file with IGES or VDA or simple format
- ☐ Load – load curves data from file
- ☐ Modify
 - ❖ Break curve
 - ❖ Joint curve
 - ❖ Translate
 - ❖ Insert and delete points

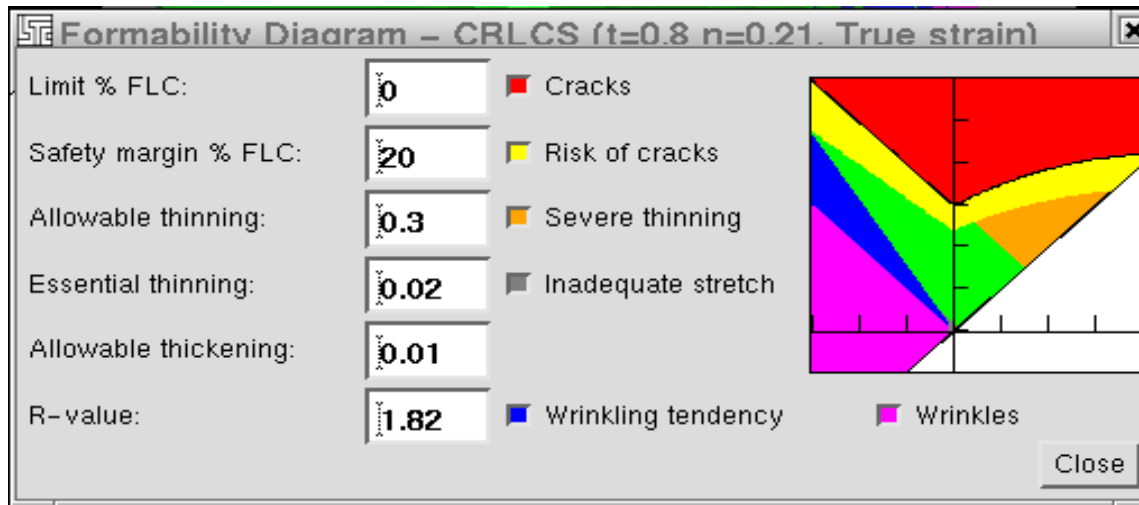


The image shows a software dialog box titled "Curves Interface". It contains several radio buttons for actions: "Show", "Create" (which is selected), "Modify", "Delete", "Load", and "Save". Below these is a text field for "Curve ID:" with the value "5". Another text field for "Curve Name:" contains "Curve_5". An "Apply" button is located below the name field. At the bottom, there are three buttons: "All", "None", and "Rev". Below these is a list box containing the following items: "1 Curve_1", "2 Curve_2", "3 Curve_3", and "4 Curve_4". A "Done" button is at the very bottom of the dialog.

Metal Stamping – FLD

Forming Limit Diagram

- ❑ Metal forming results for state of strain in formed part is analysed by reference to the forming limit curve for the material
- ❑ The biaxial strains for each element can be plotted on the FL Diagram to decide if the state of strain is safe, ie material is in a serviceable condition.
- ❑ The FLD is split into regions which are collectively known as Formability.



Forming Limit Results

Calculated FLD Curve:

t= 0.8 n= 0.21 Set

t_n File Browse

Point Tracer

FLD E*Strain

Material Clr Info

Element Formability

Area

Polygon In Out

El#=, Node=, , , ,
T=, %R=
Major Strain:
U=, L=
Minor Strain:
U=, L=

Position: Middle

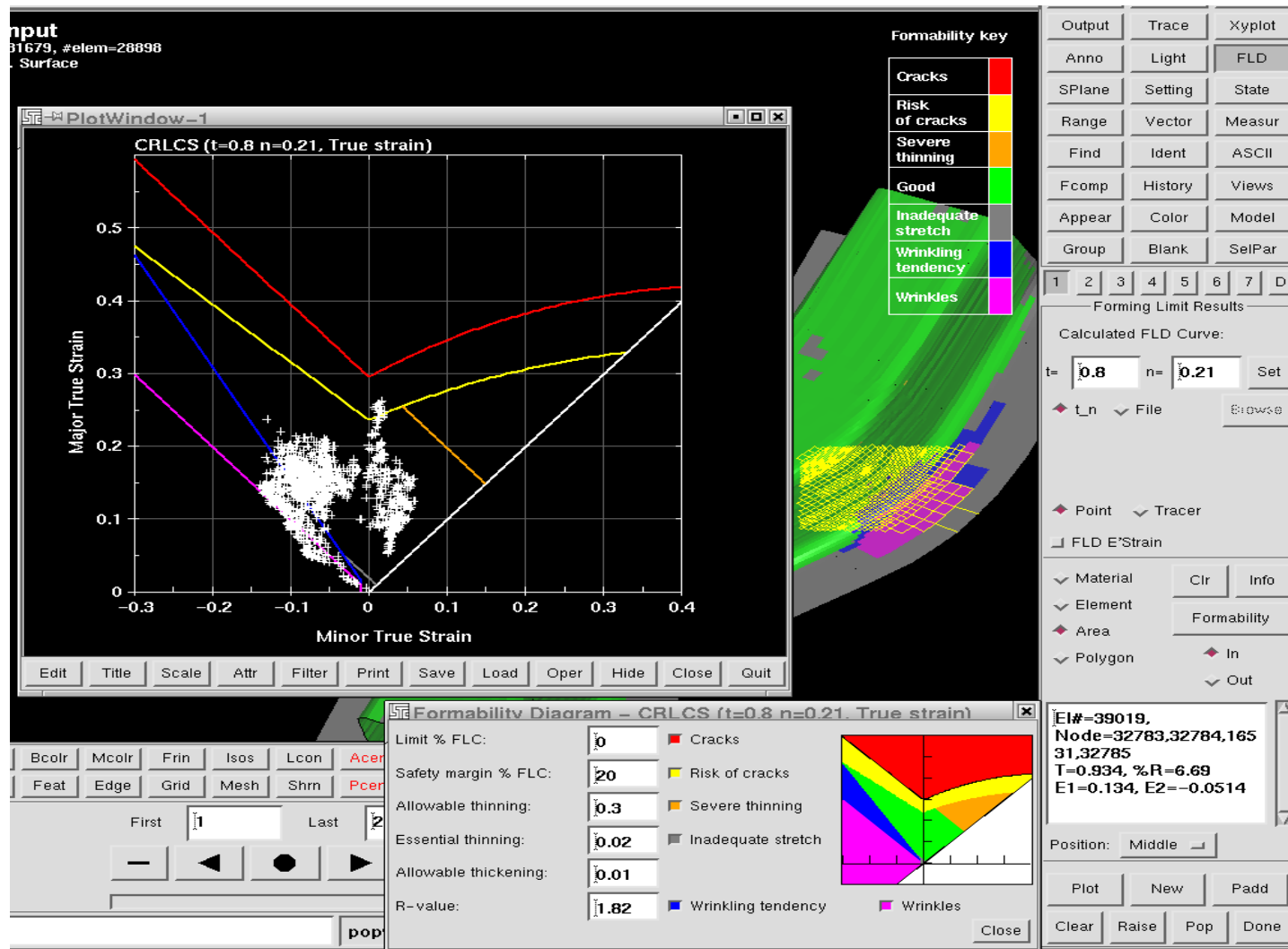
Plot New Padd

Clear Raise Pop Done



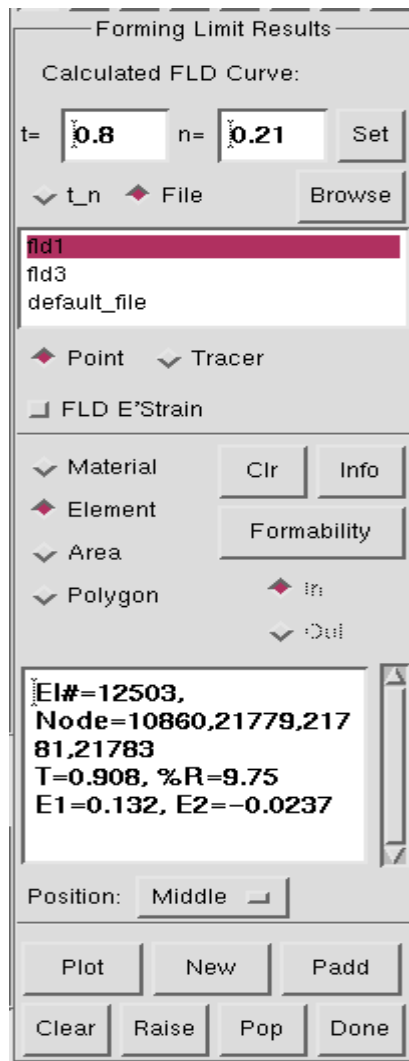
Metal Stamping – FLD

Forming Limit Diagram



Metal Stamping – FLD

Forming Limit Diagram



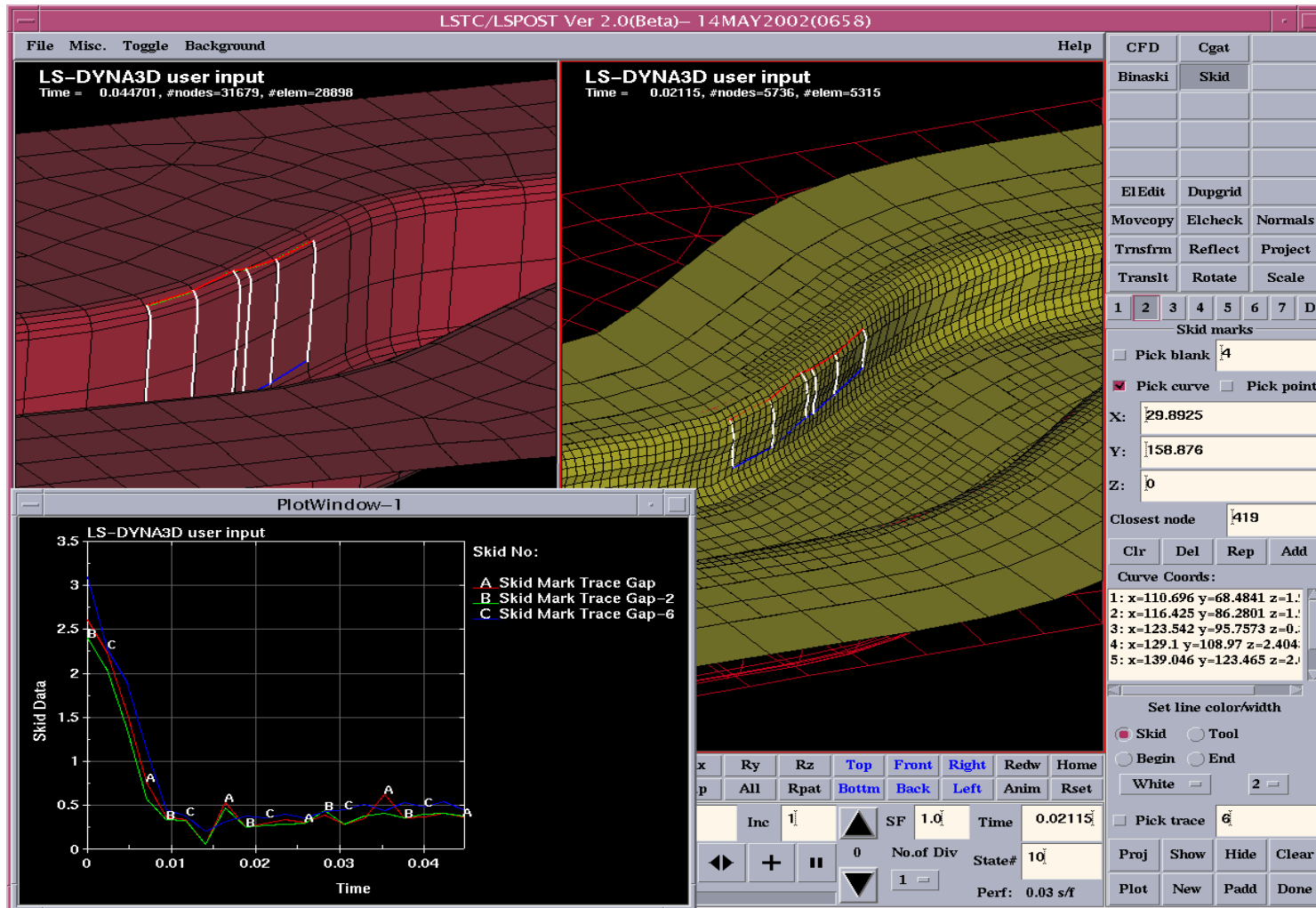
- ☐ Forming limit curve can be read in via the File menu and select from or Browse button, after selection press Set button to apply
- ☐ There is a built in curve for cold rolled carbon steel based on thickness and a factor n
- ☐ Elements can be selected by material, area and item by picking the model. FLD will be displayed by pressing the Plot button
- ☐ Either points or tracers can be displayed on the fld. The points are for the current state and tracer draws the points from state 1 to the current state
- ☐ The last picked element strain values are shown in the text list
- ☐ Strains are either at the outer shell integration points or the mean value given as the middle position

Metal Forming Skid Mark Traces

- ☐ **The motion of the blank over critical areas of the die or punch are tracked for the forming process.**
- ☐ **A set of line segments are picked on the die at the critical region. Where this line crosses the edges of the mesh representing the die gives points taken as the tracking positions on the die.**
- ☐ **The die tracking points are projected onto the blank at each plot data state.**
- ☐ **The points found on the blank as it proceeds over the die are connected up to appear as line traces. Each of these points is also tracked over the proceeding time.**



View of Skid Traces

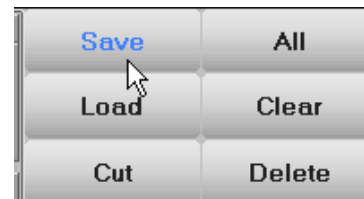
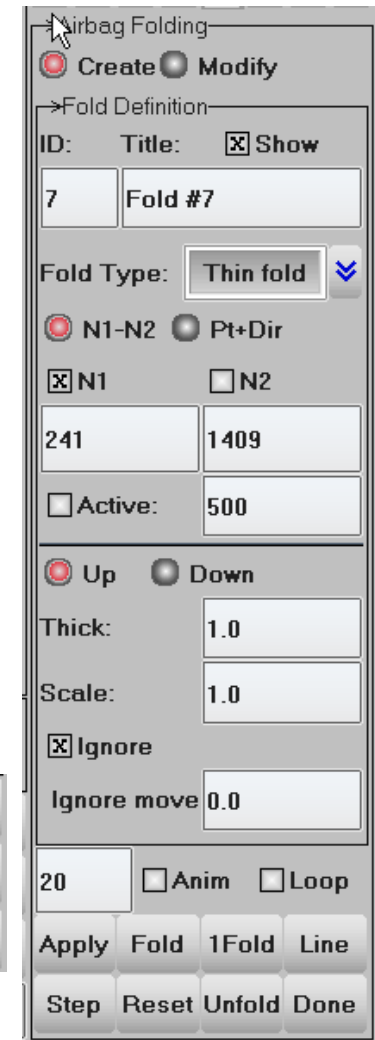


Airbag Folding

To create fold definition and fold an airbag

- ☐ Fold types
 - ❖ Thin, Thick, Tuck, Spiral
- ☐ Fold line does not have to be on element edges
- ☐ Not limit to XY plane
- ☐ Step by step folding
- ☐ Save and load fold definitions
- ☐ Fold definition can be modified
- ☐ Animate fold process to give better visual effect

```
1 thin 2783 3668 4402 +ve Up 1 1 0 0
2 thin 585 468 2411 +ve Up 1 1 0 0
3 thin 1414 496 1511 +ve Up 1 1 0 0
4 thin 3844 2786 4605 +ve Up 1 1 0 0
5 spiral 602 611 1661 +ve Up 1 10 0
6 spiral 2885 2894 3873 -ve Down 1 10 0
```

Airbag Folding

☒ Create ☐ Modify

>Fold Definition

ID: 7 Title: Fold #7 ☒ Show

Fold Type: Thin fold

☒ N1-N2 ☐ Pt+Dir

☒ N1 ☐ N2

241 1409

☐ Active: 500

☒ Up ☐ Down

Thick: 1.0

Scale: 1.0

☒ Ignore

Ignore move 0.0

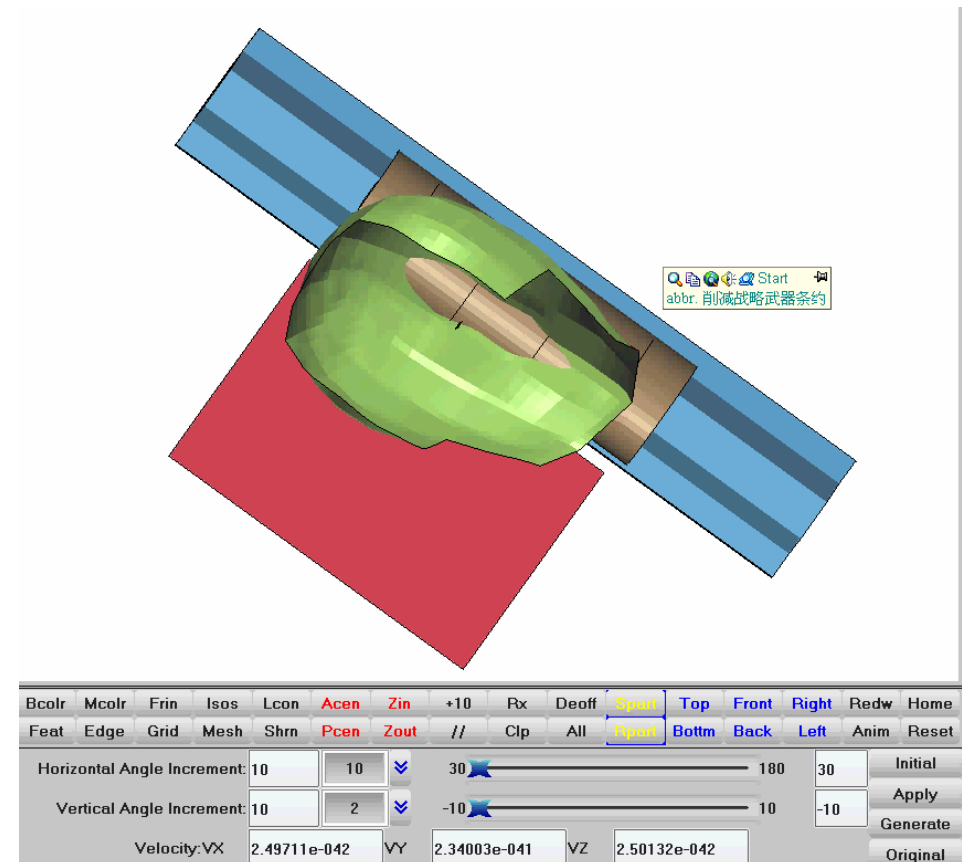
20 ☐ Anim ☐ Loop

Apply Fold 1Fold Line

Step Reset Unfold Done

Head Impact Positioning

- ☐ Read in head and locate position
- ☐ Multiple heads can be positioned in the same model
- ☐ Head can be tilted vertically or rotated horizontally interactively
- ☐ Configuration file can be setup to have head model loaded automatically
- ☐ Multiple LS-DYNA keyword files will output for each head positions



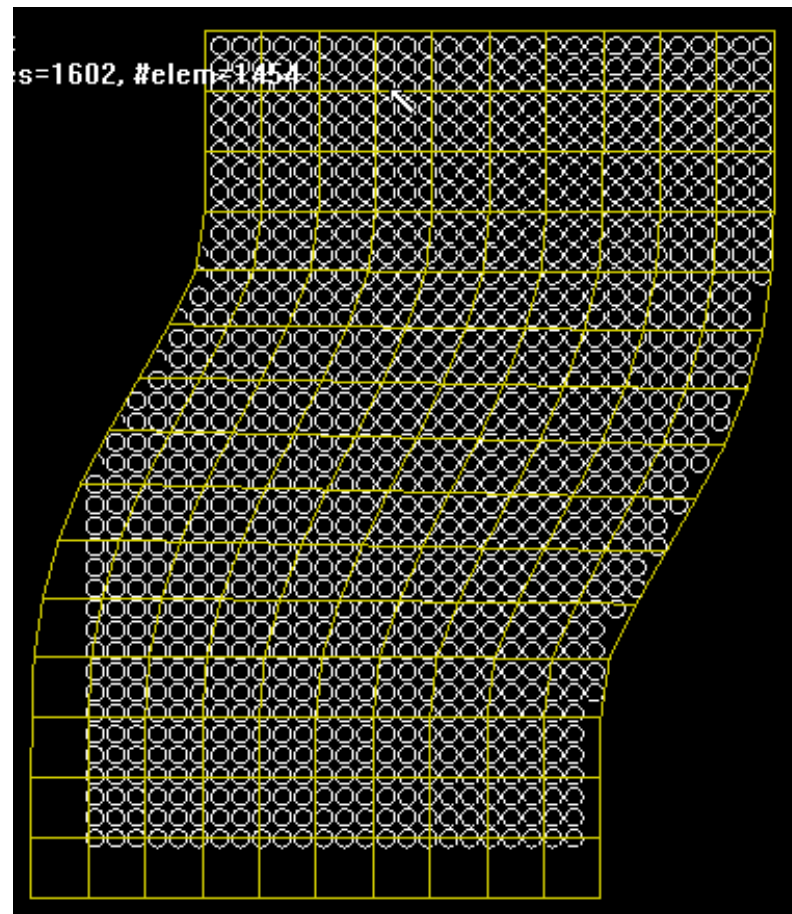
Occupant – DmyPos



- Occupant Positioning
 - ❑ Occupant positioning provides an interface to manipulate the occupant which can be integrated into other simulation.
 - ❑ Users can rotate, translate or reflect the dummy along/about specified directions.
 - ❑ Each subordinate of the occupant can be rotated about its joint defined in the corresponding tree file.
 - ❑ All constrains and physical properties will also be transformed along with the geometry data.
 - ❑ Multiple occupants control and importing are also supported.

Circular Grid Generation Technique

- ❑ A method for measuring the biaxial strain on a formed part.
- ❑ The menu allows this procedure to be simulated by tracing parametric points on the mesh through the forming simulation

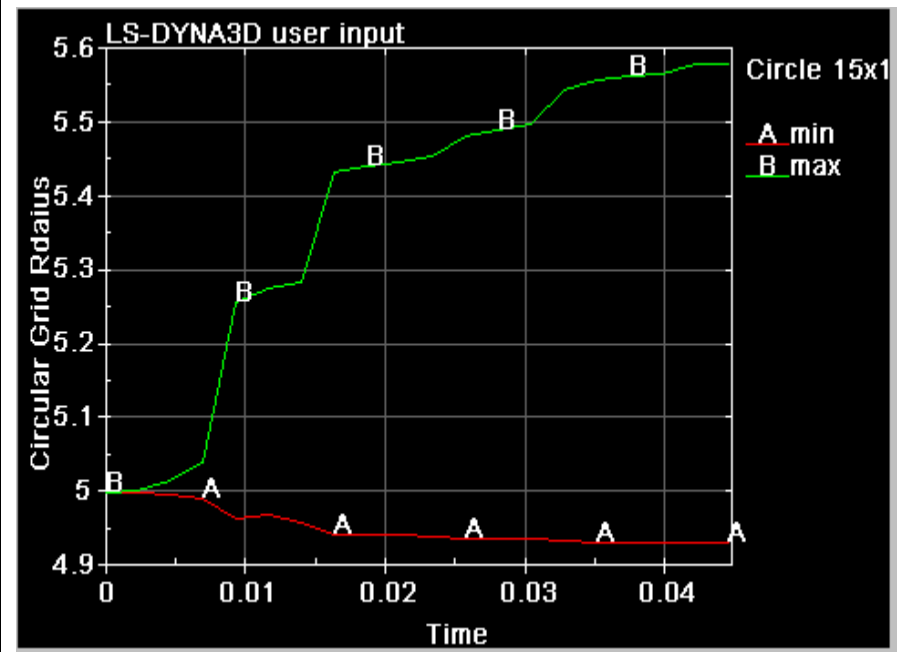
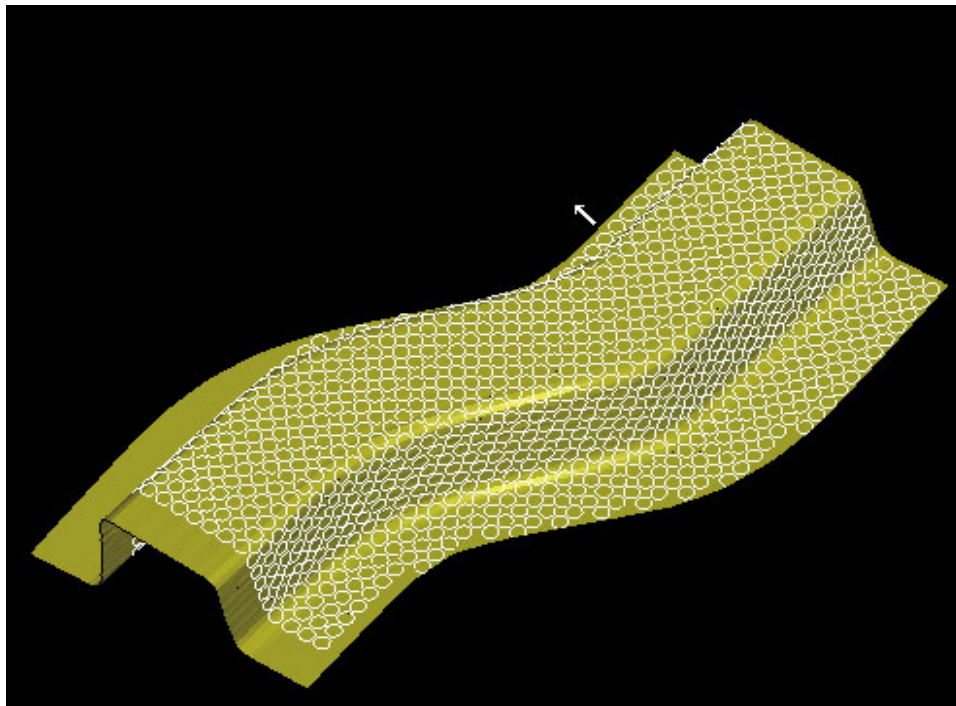


Cgat

Circles	X	Y
50	50	
Diameter		Spacing
5		6
☐ Origin	☒ Xpoint	
18.3656		134.046
17.391		17.3911
-1		-1
☐ Y-node		2909
Part No.		4
Color	White	
Linew	1	
☐ Pick	15x1	
Show	Hide	Clear
Plot	New	Padd
		Done

Circular Grid Generation Technique

- Here the part is completely formed with a refined (adapted) mesh to comply with the shape
- The change in the radii of any circle indicates the biaxial strain and can be compared to the real part scribed with the same grid, $\epsilon_1 = \ln(d_1/d_0)$ $\epsilon_2 = \ln(d_2/d_0)$



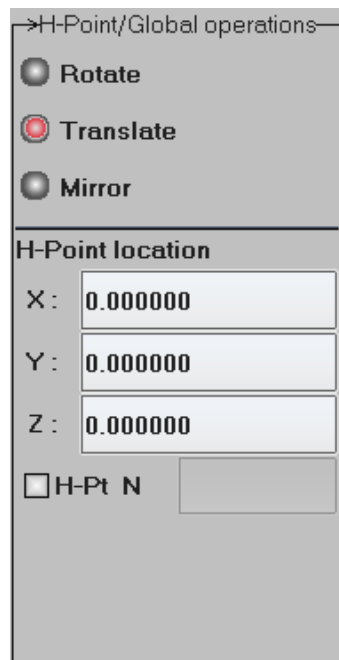
Occupant – DmyPos Basic Operations

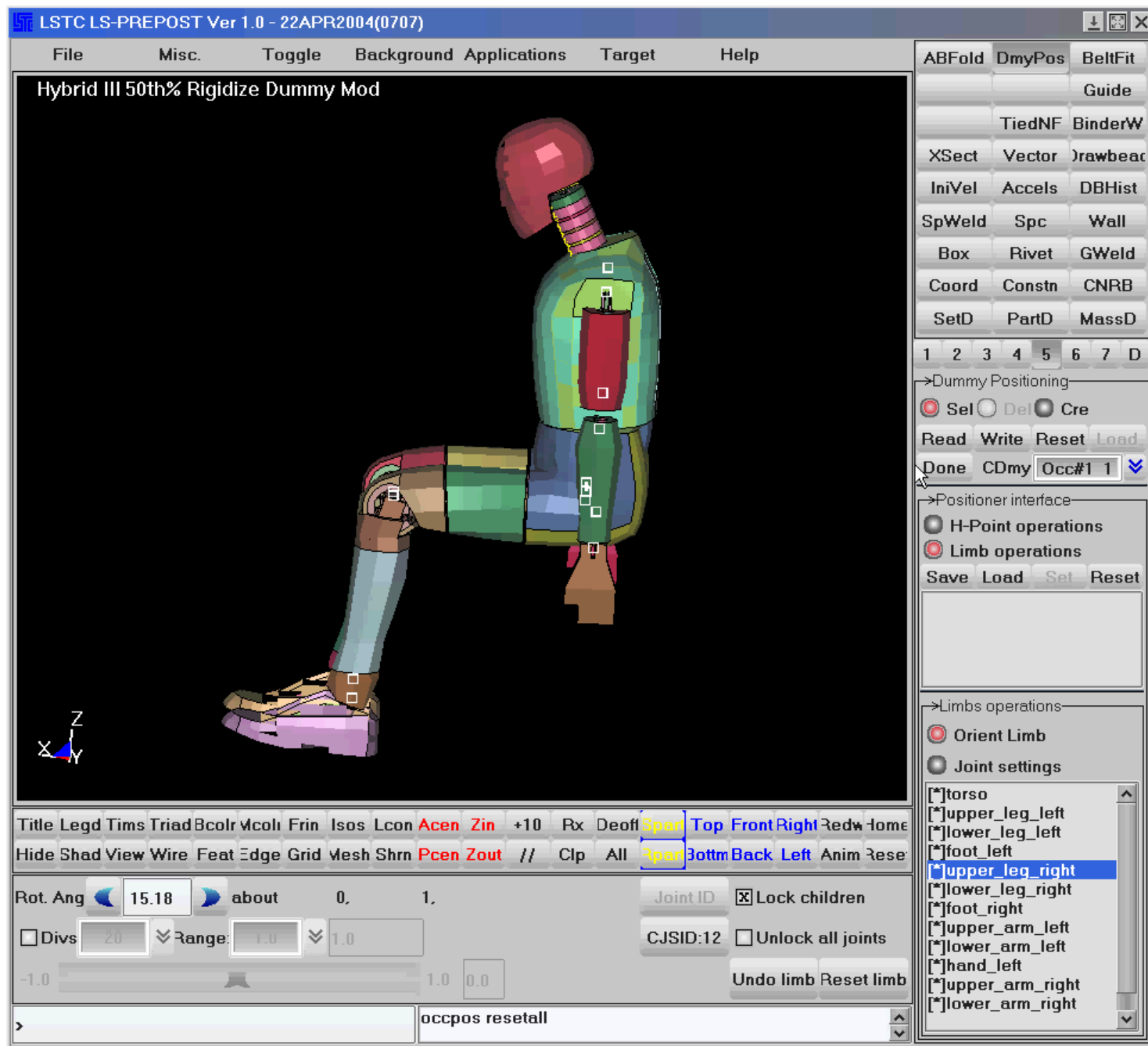
•Importing Occupants

- ☐ By central lists
- ☐ By keyword and tree files

•H-Point/Limb Operations

- ☐ H-Point: Operations that control the entire occupant.
 - ❖ Translation
 - ❖ Rotation
 - ❖ Reflection
- ☐ Limb: Operations that control a single subordinate of an occupant.
 - ❖ Orientation
 - ❖ Joint setting





Occupant – DmyPos Advance Topics

•Multiple Occupants

[File]-[Import]-[Occ. Model]

You can import as many as occupants into a model and position them into the right place.

•Tree files

- ☐ Version 2 : LS-Ingrid style
- ☐ Version 3: LS-PREPOST style

•Angle Files

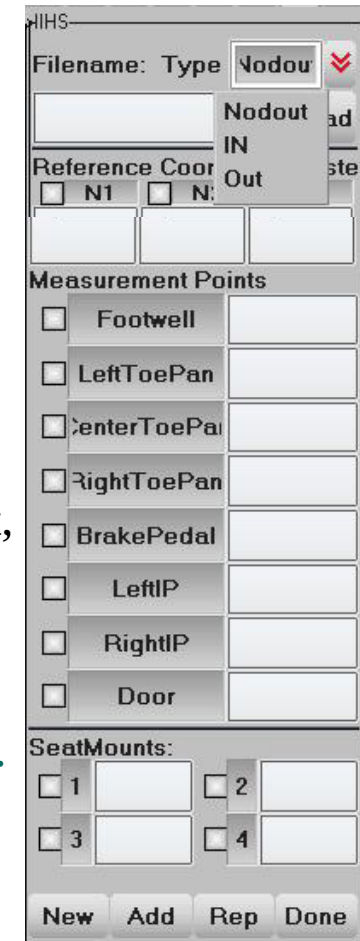
- ☐ Inherited from LS-Ingrid, that LS-PREPOST can also take the angle files and position a subordinate into an exact orientation.

•Stop Angles

- ☐ *CONSTRAINED_JOINT_STIFFNESS_{*option*} card in the keyword file of an occupant can potentially define stop angles for a subordinate, if stop angles were reached, LSPREPOST will not allow further orientation of the subordinate.

IIHS

- ❑ Use this interface to get IIHS (Insurance Institute of Highway Safety) intrusion plot
- ❑ Filename: Type Nodout/IN/OUT
- ❑ Read **Nodout** file which consists of the nodes required to plot intrusions for the model as measured by IIHS
- ❑ Read **IN** file which contains the node numbers assigned for each parameter required for IIHS intrusion plot. If **IN** file does not exist, then the node numbers should be entered through key input or picked from graphical interface.
- ❑ Read **OUT** file to re-plot the IIHS intrusion plot, previously saved.

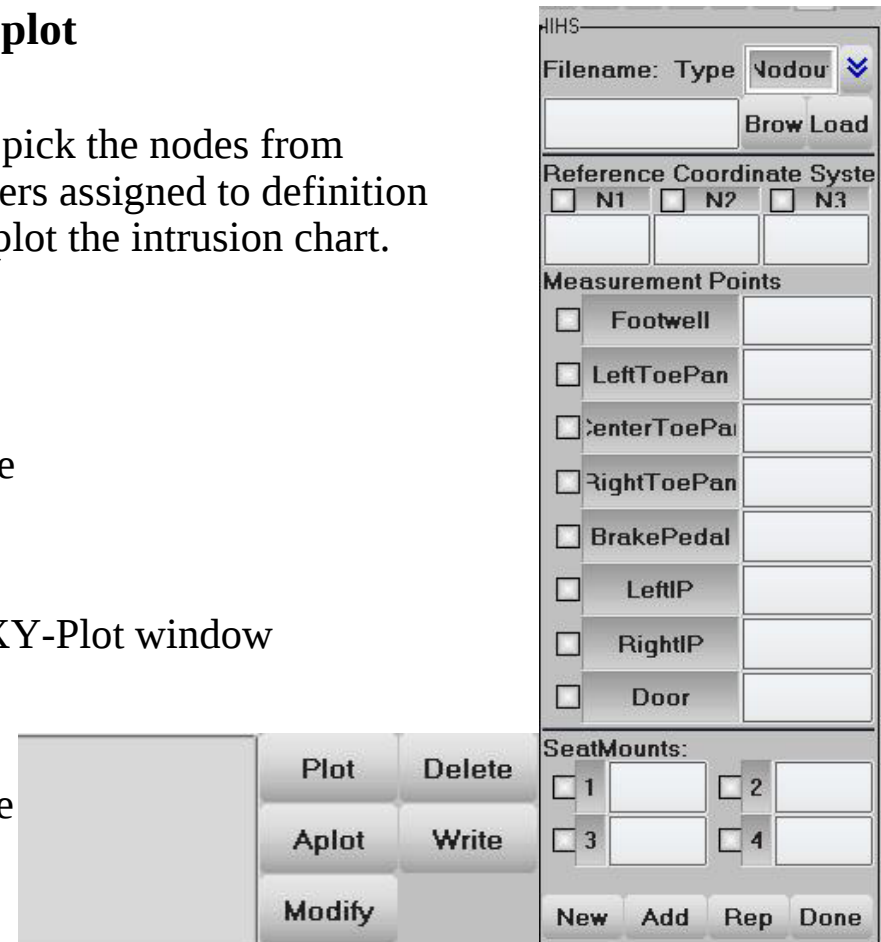


The screenshot shows the IIHS software interface. At the top, it says 'IIHS'. Below that is a 'Filename: Type' dropdown menu currently set to 'Nodout'. To the right of this is a small window showing 'Nodout', 'IN', and 'OUT' options. Below the filename field are checkboxes for 'Reference Coord' and 'N1', and a field for 'N2'. The main section is titled 'Measurement Points' and contains a list of vehicle components with checkboxes and input fields: Footwell, LeftToePan, CenterToePan, RightToePan, BrakePedal, LeftIP, RightIP, and Door. At the bottom is a 'SeatMounts' section with checkboxes and input fields for 1, 2, 3, and 4. At the very bottom are four buttons: 'New', 'Add', 'Rep', and 'Done'.

IIHS

- ☐ **Use this interface to get IIHS intrusion plot**
- ☐ Filename: Type Nodout/IN/OUT
- ☐ Once the appropriate file is read in, you can pick the nodes from graphical interface or Key in the node numbers assigned to definition of Reference plane and points to be used to plot the intrusion chart.
- ☐ [NEW] New IIHS plot
- ☐ [ADD] Add an IIHS plot
- ☐ [REP] Repeat?
- ☐ [Done] Done with IIHS plotting interface

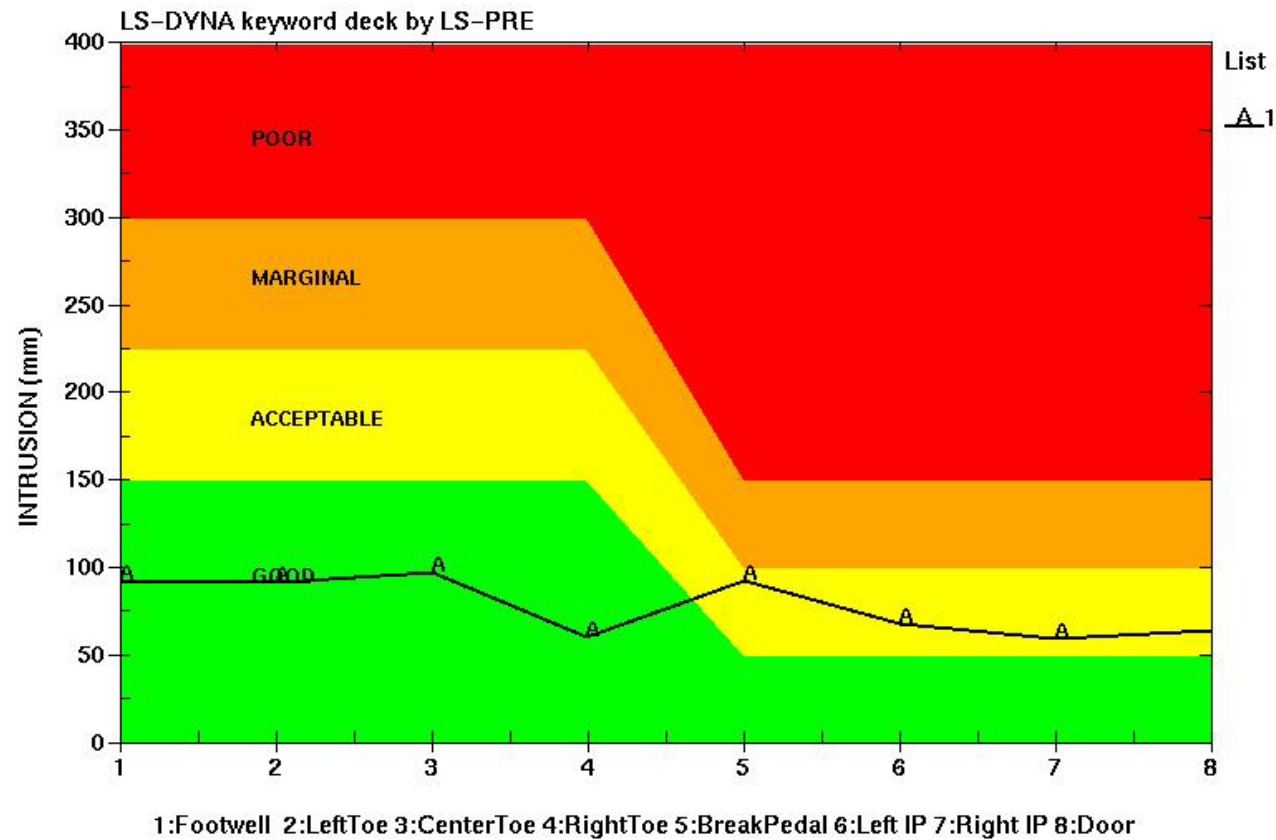
- ☐ Plot Plot the selected IIHS plot
- ☐ APlot Add intrusion plot in the current XY-Plot window
- ☐ Modify Modify intrusion data
- ☐ Delete Delete IIHS plot from the list
- ☐ Write Write IIHS intrusion data into a file



The screenshot shows the IIHS plotting interface. It includes a 'Filename: Type' dropdown set to 'Nodou', a 'Browse Load' button, and a 'Reference Coordinate System' section with checkboxes for N1, N2, and N3. Below this is a 'Measurement Points' section with checkboxes for Footwell, LeftToePan, CenterToePan, RightToePan, BrakePedal, LeftIP, RightIP, and Door, each followed by an input field. At the bottom is a 'SeatMounts' section with checkboxes for 1, 2, 3, and 4, each followed by an input field. A control panel on the left contains buttons for Plot, Delete, APlot, Write, and Modify. At the very bottom are buttons for New, Add, Rep, and Done.



IIHS



IIHS

Filename: Type

Reference Coordinate System
☐ N1 ☐ N2 ☐ N3
123 345 567

Measurement Points

☐ Footwell	34546
☐ LeftToePan	65465
☐ CenterToePan	56788
☐ RightToePan	343425
☐ BrakePedal	454657
☐ LeftIP	5655
☐ RightIP	45466
☐ Door	656754

SeatMounts:

☐ 1	5656	☐ 2	6546
☐ 3	3456	☐ 4	5764

New Add Rep Done

Plot Delete
Aplot Write
Modify

SPH - SphGen

Generating SPH particles in simple geometries.
Current available geometries are

- ☐ Box
- ☐ Sphere
- ☐ Cylinder

This user interface is still under construction,
and will have more capability in the future.

Particle generation interface:

New Load Done

☒ Gen ☐ Wrk

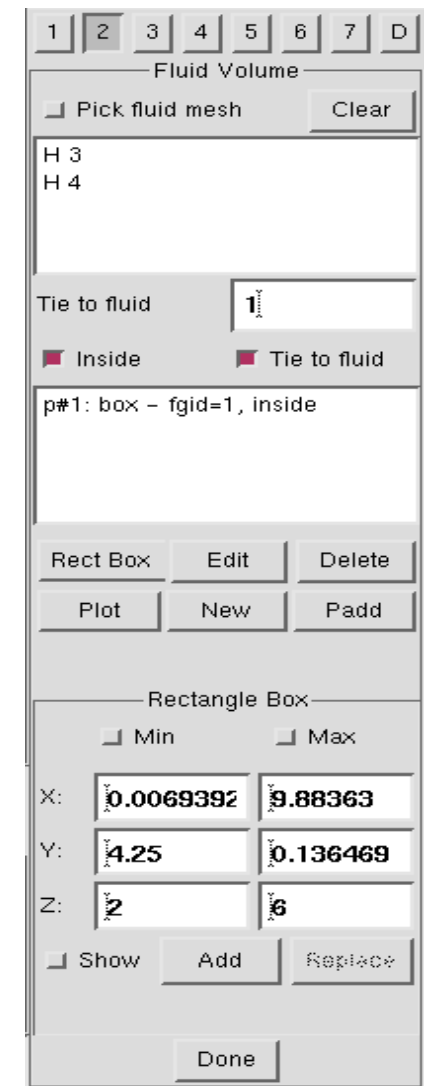
Box Sphere Cylinder

Sketch

X1:	0.000000	X2:	1.000000	Nx:	10	Box name BOX01 ☐ Interferred remove Part ID 0 Mat ID 1 >>	Apply Done
Y1:	0.000000	Y2:	1.000000	Ny:	10		
Z1:	0.000000	Z2:	1.000000	Nz:	10		

ALE Mat Menu

- ❑ Use to measure the volume of fluid occupying a Control Volume.
- ❑ A control volume is defined from primitives and shell parts. Primitives are: plane, cylinder/cone, rectangle box and a sphere
- ❑ Combined together complicated shapes can be achieved.
- ❑ Before using the menu the fluids in the d3plot file must be attained by using the SelPar menu and Fluid check button



1 2 3 4 5 6 7 D

Fluid Volume

☐ Pick fluid mesh

H 3
H 4

Tie to fluid

☒ Inside ☐ Tie to fluid

p#1: box - fgid=1, inside

Rectangle Box

☒ Min ☐ Max

X:

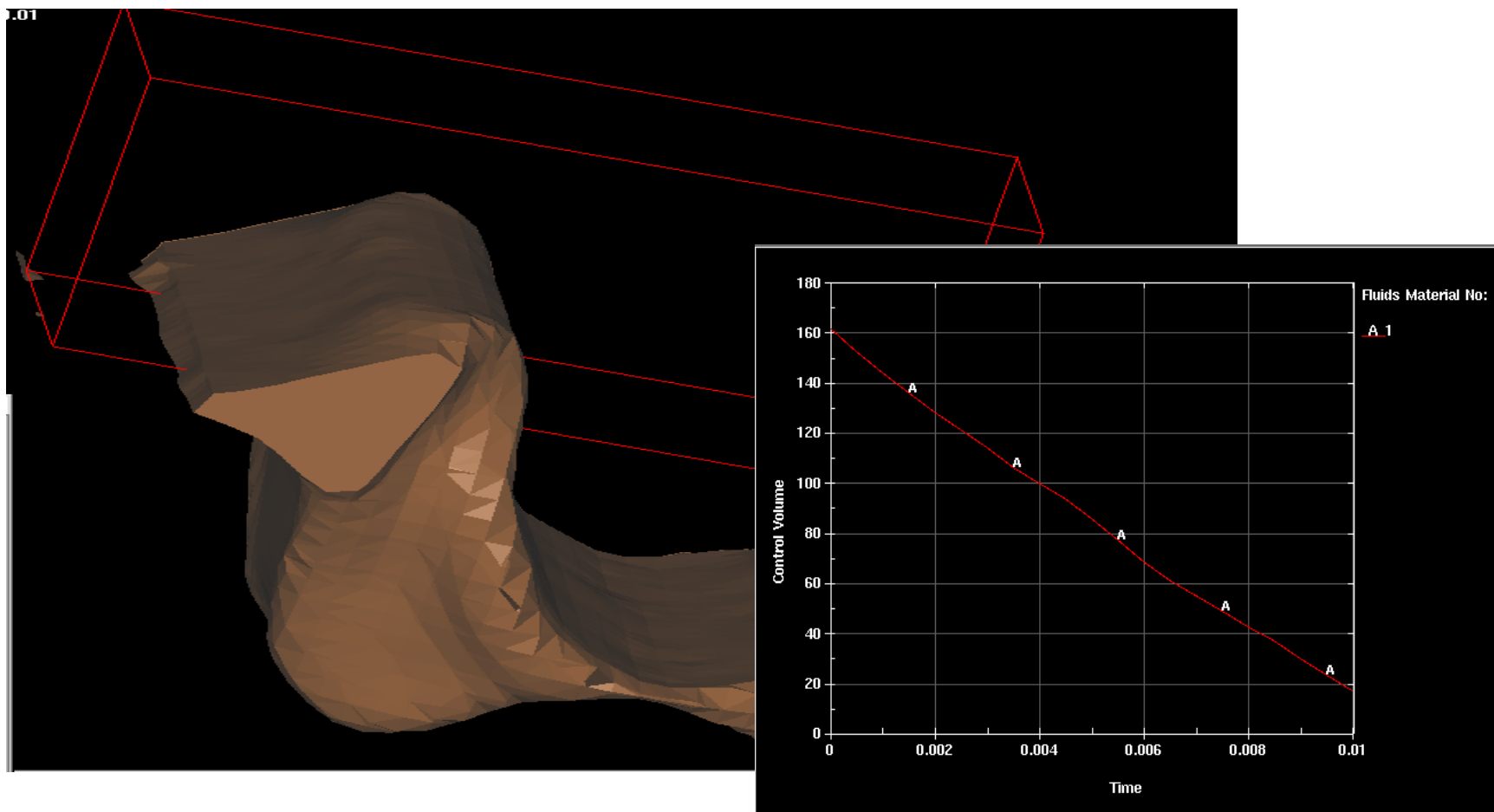
Y:

Z:

☐ Show

ALE Mat Menu

A rectangular box with fluid flowing out

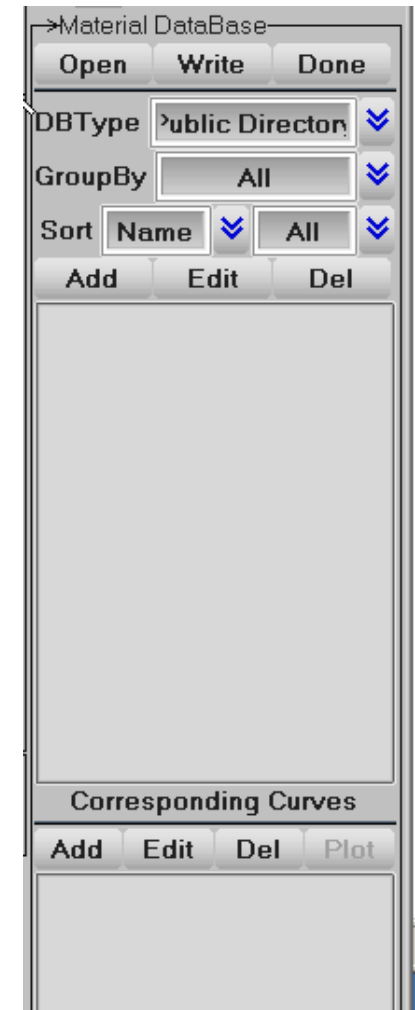


ALE Mat Menu

- ☐ The volume of Fluid is calculated based on finding the volume of each Euler element with the control volume times the volume fraction of the fluid in the element.
- ☐ Where the control volume cuts an Euler element the volume of the element contained time the volume fraction is used.
- ☐ Over time the fluid flows through the Euler mesh and that volume of it within the bounds of the control volume is plotted against time.
- ☐ The creation of the control volume is stored as keyword ***INITIAL_VOLUME_FRACTION_GEOMETRY** and can be used to create complex volume definition for fluids simulations

Material Database

- ☐ Manage material data as separate entity
- ☐ Put material data in files and directories.
- ☐ Define material data by public and private ownership.
- ☐ Material data including corresponding curves and coordinate systems
- ☐ Material data file/directory name can be save in configuration file
- ☐ Allow user to write to material database
- ☐ Material data can be easily transferred between model and database



Configuration File

File name - .lspostrc

- ☐ Ls-PrePost searches this file in the directories in the order:
 - ❖ LSTC_FILE directory (normally /usr/local/lstc)
 - ❖ User \$HOME directory
 - ❖ Working directory
- ☐ The last encountered parameter will be used
- ☐ Only used parameters need to be stored in .lspostrc file
- ☐ .lspostrc is saved with the current setting in File -> Save Config pull down menu