

Introduction to LS-PrePost

Quanqing Yan, Philip Ho, LSTC

2018

Table of Contents

SECTION	WORKSHOPS
Overview	-
General Operations	Workshop 1
Mesh Generation	Workshop 2
Pre-Processing	Workshop 3-4-5
Post-Processing	Workshop 6
Configuration Files	-

Overview

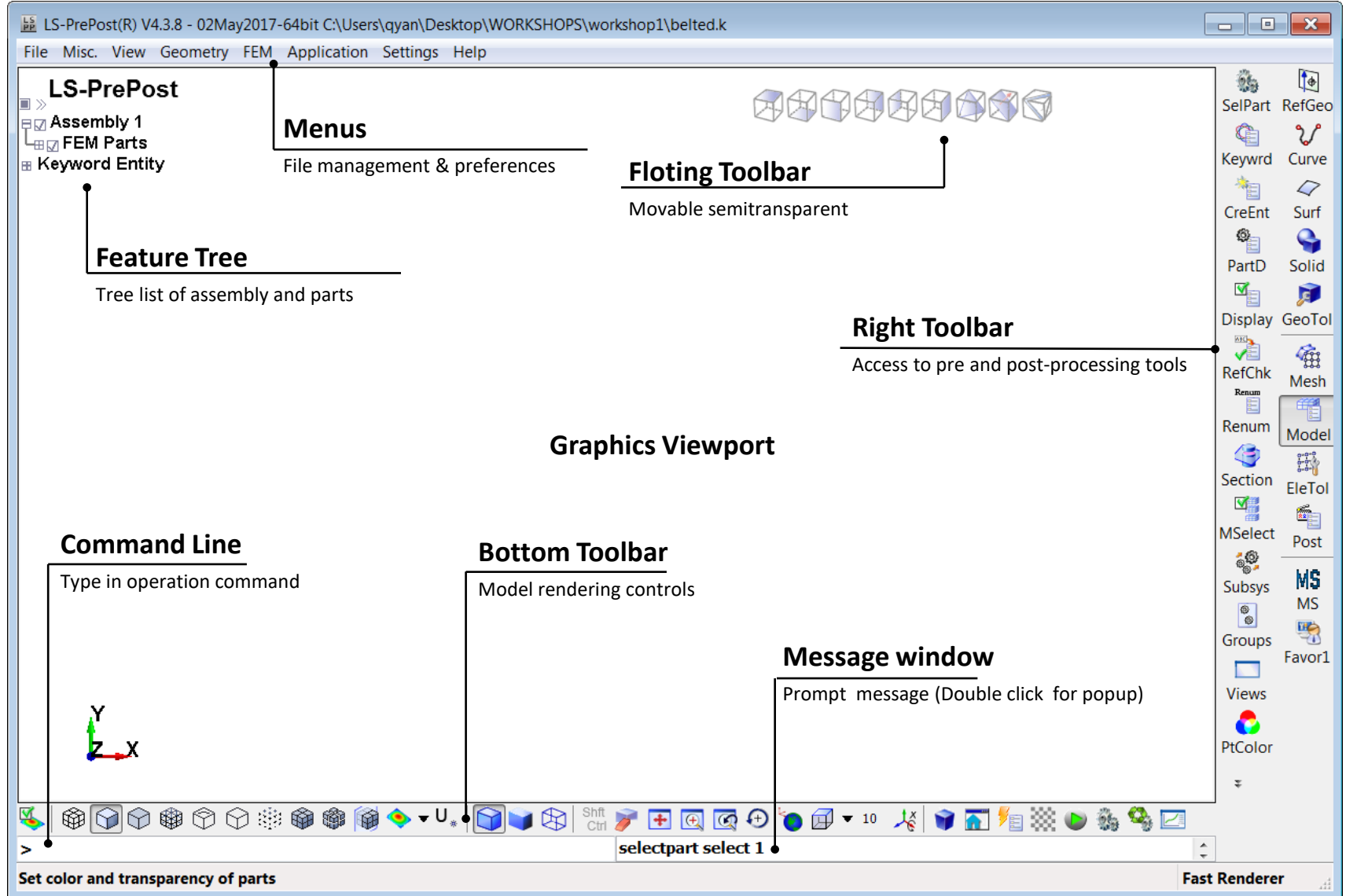
About LS-PrePost

- LS-PrePost is an advanced pre and post-processor designed specifically for LS-DYNA
- LS-PrePost is developed for Windows and Linux
- LS-PrePost is ***Free***
- Core Functionality
 - Full support of LS-DYNA keyword files
 - Full support of LS-DYNA results files
 - Robust handling of geometry data (new CAD engine)
 - Pre-processing (meshing, model clean-up, entity creation)
 - Post-processing (animation, fringe plotting, curve plotting)

Online Resources

- Official Website
 - <http://www.lstc.com/lspg>
- User Group
 - <http://groups.google.com/group/lc-prepost>
- Latest Release Version:
 - <http://ftp.lstc.com/anonymous/outgoing/lcprepost/4.5/>
 - <ftp://ftp.lstc.com/outgoing/lcprepost/4.5/>
- Beta Version:
 - <http://ftp.lstc.com/anonymous/outgoing/lcprepost/dev>
- Training notes:
 - <ftp://ftp.lstc.com/outgoing/qyan/Class>

GUI Layout



Input / Output

- Input (partial list)
 - FEM: LS-DYNA Keyword, Nastran, I-DEAS Universal, PAM-CRASH, RADIOSS, ABAQUS
 - CAD: IGES, STEP
 - ASCII: glstat, matsum, etc...
 - Binary: d3plot, binout, etc...
- Output (partial list)
 - FEM: LS-DYNA Keyword, Nastran, STL
 - Image: PNG, TIFF, BMP, GIF, JPG, PostScript
 - Movie: AVI, MPEG, Animated GIF, JPEG
 - XY Data: CRV, CSV, XML
 - Other: Post.db, Project File

Mouse and Keyboard

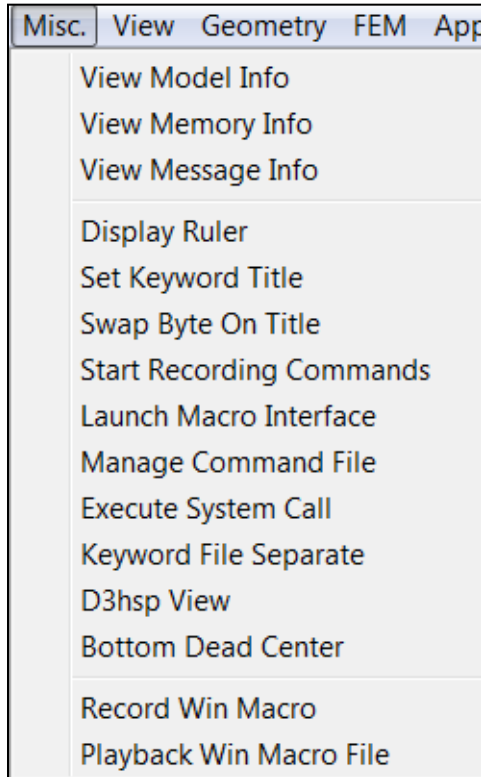
- Dynamic Model Operation
 - Rotate: Shift + Left-click
 - Translate: Shift + Middle-click
 - Zoom: Shift + Right-click/Scroll-wheel
(Using Ctrl instead of Shift for edge mode)
- Graphics Selection
 - Pick (single): Left Click
 - Area (rectangle): Left-click + Drag
 - Poly (polygon): Left-click at corners / Right-click to finish
- List Selection
 - Multi-Select: Left-click + Drag / Ctrl + Left-click
- Mouse over controls for status bar help comments

File Menu

File	Misc.	View	Geometry	File
New				
Open				▶
Import				▶
Recent				▶
Save				▶
Save As				▶
Update			Ctrl+U	
Run LS-DYNA				
Print...			Ctrl+P	
Movie...			Ctrl+M	
Exit			Ctrl+X	
Save and Exit				

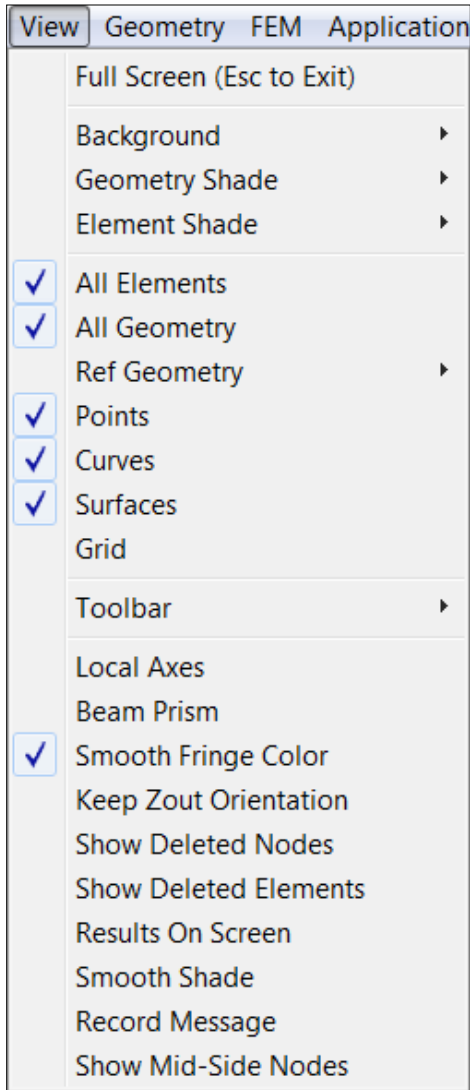
- **New** – Launch a new session of LS-PrePost, all model/data will be closed (only in version 4.0 and later)
- **Open** – Open file (new model created for each file opened)
- **Import** – Import file (adds keyword data to current model)
- **Recent** – Open recent files (stored in /user/.lspp_recent)
- **Save** – Over-write current *Keyword* or *Project* file
- **Save As** – Save any of the following file formats using advanced options: *Keyword*, *Active Keyword* (visible data), *Project*, *Post.db* (condensed d3plot data), *Geometry*, *Keyword and Project* (using same file name)
- **Update** – Load new d3plots for run in progress
- **Run LS-DYNA** – pop up LS-DYNA job submission dialog, currently only limited to the same local machine LS-PrePost is running
- **Print...** – Launch printing interface (send to printer or image file)
- **Movie...** – Launch movie generation interface
- **Exit** – Exit LS-PrePost
- **Save and Exit** – Save data to current file and exit LS-PrePost

Miscellanies Menu



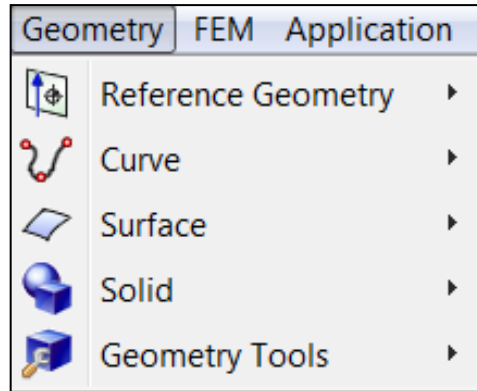
- **View Model Info** – Launch model information interface
- **View Memory Info** – Launch memory usage interface
- **View Message Info** – Launch keyword reader message interface
- **Display Ruler** – Launch ruler interface
- **Set Keyword Title** – Launch title interface
- **Swap Byte On Title** – Swap byte order for title
- **Start Recording Commands** – Start/stop recording macro commands
- **Launch Macro Interface** – Launch Macro interface
- **Manage Command File** – Launch command file interface
- **Execute System Call** – Launch system call interface
- **Keyword File Separate** – Separate a single keyword file into multiple files based on *KEYWORD title

View Menu



- **Full Screen (Esc to Exit)** – Show graphics area in full screen mode
- **Background** – Set background style (plain, faded, or image)
- **Geometry Shade** – Set geometry rendering style
- **Element Shade** – Set element rendering style
- **All Element/Geometry** – Show all elements/geometry
- **Ref Geometry** – Show reference geometry
- **Points/Curves/Surfaces** – Show points/curves/surfaces
- **Toolbar** – Hide/display toolbars, set text/icon mode, and set font size
- **Local Axes** – Toggle between global/local
- **Beam Prism** – Toggle between line/prism
- **Smooth Fringe Color** – Enable smooth contours (when in fringe mode)
- **Keep Zout Orientation** – Maintain orientation when zooming out
- **Show Deleted Nodes** – Display deleted nodes when viewing results
- **Show Deleted Elements** – Display deleted elements when viewing results
- **Result On Screen** – Display results on screen when Ident > Show Results is active (when in fringe mode)
- **Smooth Shade** – Use smooth shading
- **Record Message** – Write messages to lspost.msg
- **Show Mid-Side Nodes** – for 10-node Tetrahedron element

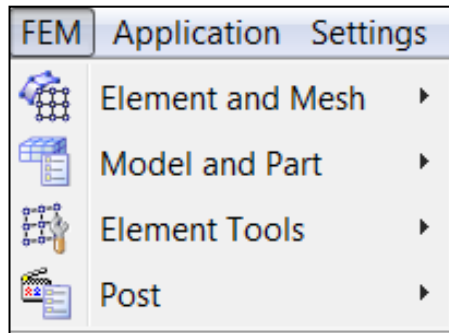
Geometry Menu



- **Reference Geometry** – Access tools for creating and editing reference geometry (Axis, Plane, Coordinate System, and Point)
- **Curve** – Access tools for creating and editing curves (Point, Line, Circle, Circular Arc, Ellipse, Elliptical Arc, BSpline Curve, Helix, Composite Curve, Break Curve, Merge Curve, Bridge Edge, Smooth Curve, Middle Curve, Morphing Curve, Fillet Curve)
- **Surface** – Access tools for creating and editing surfaces (Plane, Cylinder, Cone, Sphere, Torus, Fill Plane, Extrude, Revolve, Sweep, Loft, N-Side Surface, Patch Surface, Bridge Two Faces, Combine Faces, Fit From Points/Mesh, Middle Surface, Surface Morphing)
- **Solid** – Access tools for creating and editing solids (Box, Cylinder, Cone, Sphere, Torus, Extrude, Revolve, Sweep, Loft, Fillet, Chamfer, Draft, Thicken, Wedge, Boolean)
- **Geometry Tools** – Access other geometry tools (Delete Face, Extend Curve, Extend Face, Intersection, Offset, Project, Replace Face, Stitch Faces, Trim Transform, Copy Entity, Management, Heal, Topology Simplify, Measure)



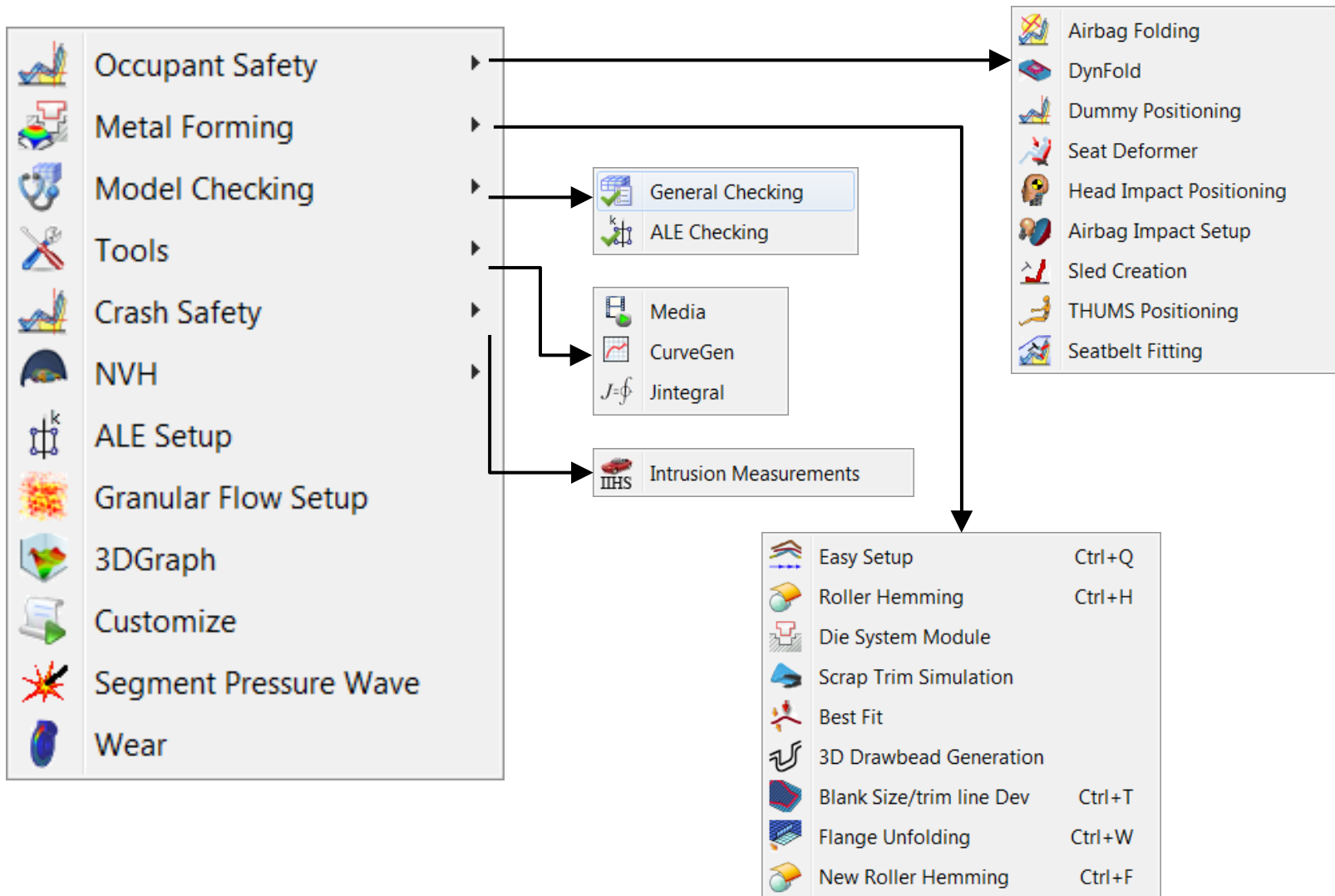
FEM Menu



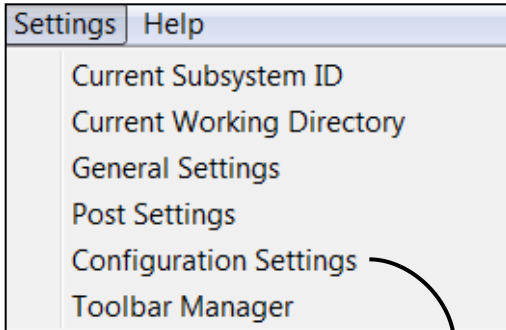
- **Element and Mesh** – Access mesh creation tools (Shape Mesher, Auto Mesher, Solid Mesher, Block Mesher, N-Line Mesher, Tetrahedral Mesher, Blank Mesher, Element Generation, Node Editing, Element, Editing, Mass Trimming, Spot Welding, SPH Generation)
- **Model and Part** – Access model and part tools (Assembly and Select Part, Keyword Manager, Create Entity, Display Entity, Reference Check, Renumber, Section Plane, Model Selection, Subsystem Manager, Group, View, Part Color, Appearance, Annotation, Split Window, Explode, Lighting Setup)
- **Element Tools** – Access element tools (Identify, Find, Blank, Move or Copy, Offset, Transform, Normals, Detach, Measure, Morph, Smooth, Part Trim, Part Travel)
- **Post** – Access post-processing tools (Fringe Component, Fringe Range, History, XY Plot, ASCII, Binary Output, Follow, Trace, State, Particle, Circle Grid, Chain Model, FLD, Output, Setting, Vector)
- **Favorites** – Customizable toolbar (see Setting → Toolbar Manager)



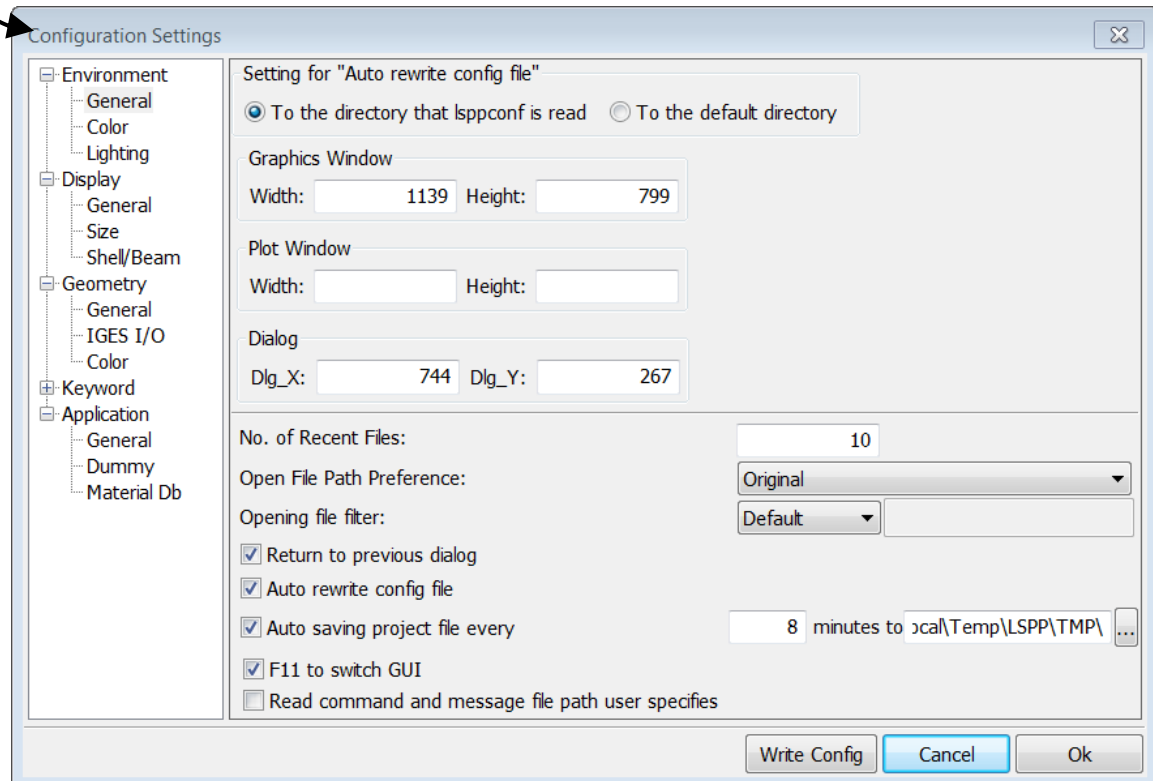
Application Menu



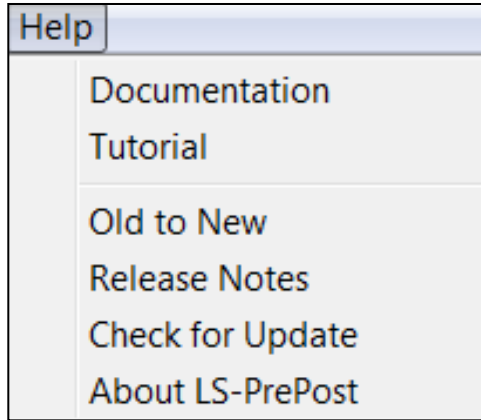
Setting Menu



- **Current Subsystem ID** – Set current subsystem ID
- **Current Working Directory** – Set current working directory
- **General Settings** – Set general parameters
- **Post Settings** – Set post processing parameters
- **Configuration Settings** – Set configuration settings
- **Toolbar Manager** – Customize toolbars

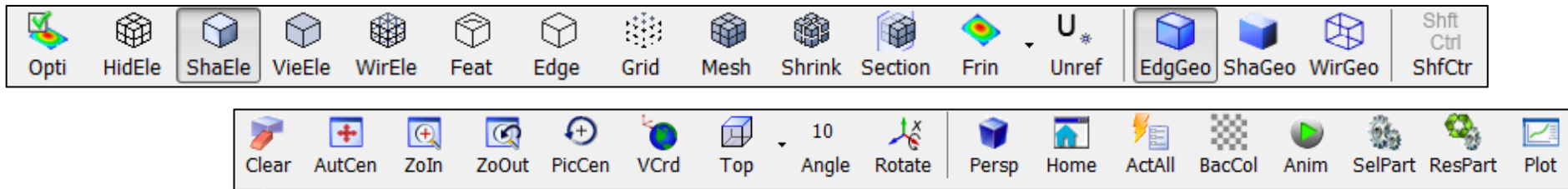


Help Menu



- **Document** – View LS-PrePost documentation
- **Tutorial** – View LS-PrePost tutorials
 - Will need to download the Help and Tutorial documents from LSTC's ftp site the first time these documents are being accessed
- **Old to New** – View mapping between old (v2.4) and new (v3.0+) GUI
- **Release Notes** – View release notes
- **Check for Update** – Check server for newer version of LS-PrePost
- **About LS-PrePost** – View version info

Bottom Toolbar



Opti Toggle Title, Legend, Min-Max, Time Stamp, Triad, Background Color, Mesh Color, Performance Stats on/off, Feature Tree, ISO View and Animation Play

HidEle Display elements with hidden lines removed

ShaEle Display elements in shaded mode with mesh lines off

VieEle Display elements in plain color mode

WirEle Display elements in wireframe mode

Feat Display elements in feature line mode (default angle=30°)

Edge Display elements in edge line mode

Grid Display each nodal point as a colored pixel

Mesh Toggle element mesh on/off

Shrink Draw elements in shrunken mode (default=0.85)

Section Display section view with plane selected

Frin Toggle Fringe/Line-contours/Iso-surfaces

Unref Toggle unreferenced nodes on/off

EdgGeo Display geometry in shaded mode with edges on

ShaGeo Display geometry in shaded mode with edges off

WirGeo Display geometry in wireframe mode

ShfCtr Toggle Off/Shift/Ctrl (for one-handed rotate/pan/zoom)

Clear Clear all picked or highlighted information

AutCen Automatically center model to fit within window

ZoIn Zoom in, click and drag to draw a box

ZoOut Zoom out to previous zoom position

Pcen Pick node as new center point for model rotation

VCrd View coordinate systems

Top Choose Top, Bottom, Front, Back, Right, or Left view

Angle Left-click to reverse rotation direction
Right-click to modify rotation angle

Rotate Left-click to rotate about axis shown
Right-click to switch rotation axis (X/Y/Z)

Persp Toggle Parallel/Perspective view mode

Home Restore default view and fit model

ActAll Restore all entities to be active

BacCol Toggle background color black/white (Plain background mode only)

Anim Display animation controls or start/stop animation

SelPart Launch Assembly and Select Part interface

ResPart Restore the last removed part (Shift+R)

Plot Open XY plot management interface

General Operations

FEM General Selection

Pick Select a single entity

Area Select using a rectangular window

Poly Select using an irregular polygon

Sel1 Pick 1 entity (only 1 will be in buffer)

Sphe Select entities within a sphere

Box Select entities within a box

Prox Select entities within proximity to a part

Circ Select entities within a circle

Frin Select entities within a fringe plot range

Plan Select entities within a plane

In Select entities inside Area/Poly

Out Select entities outside Area/Poly

Add Add entities to a selection set

Rm Remove entities to a selection set

ID Enter Ids of the entity to be selected into the text box for highlighting

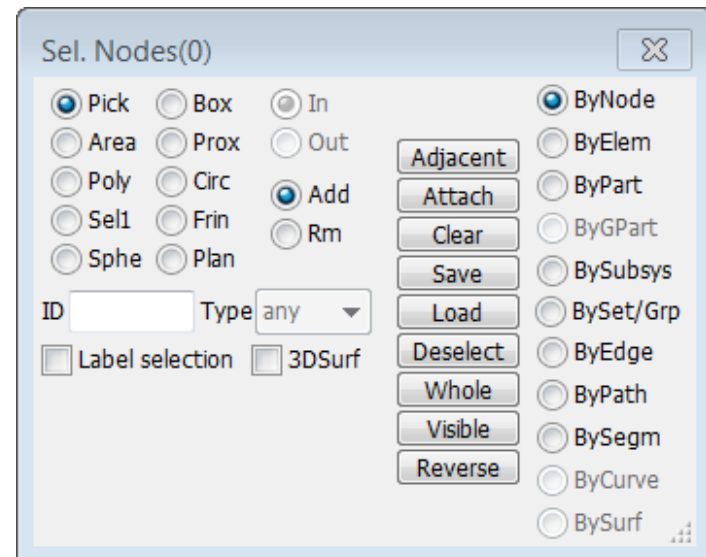
Type When selecting element, choose element type to be selected

Label selection put a label on the selected entities

Prop Propagate selection (pick seed)

Adap Propagate across adaptive elements

Ang Feature angle for propagation to stop



FEM General Selection

Adjacent Select adjacent elements

Attach Select attached elements

Clear Clear selection

Save Save selection to buffer or to file

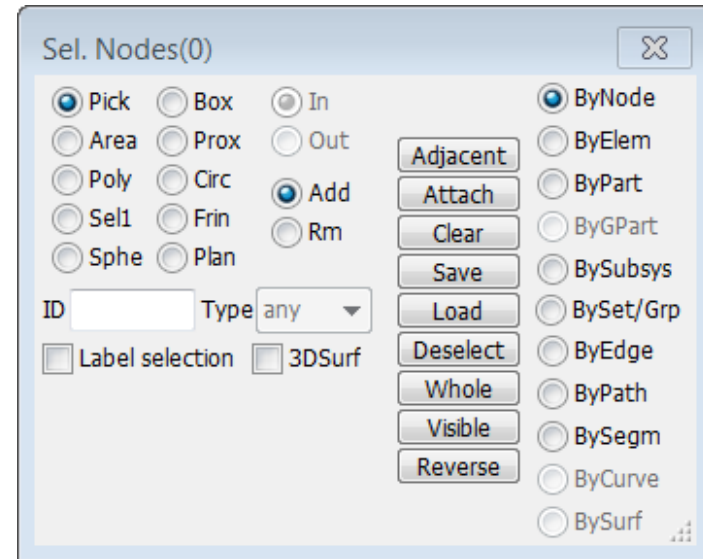
Load Load selection from buffer or from file

Deselect Undo last selection

Whole Select all entities in model

Visible Select all visible entities

Reverse Reverse selection



ByNode Select nodes

ByElem Select elements

ByPart Select parts

BySet Set based selection

ByEdge Edge based selection

ByPath Select first and last along a path

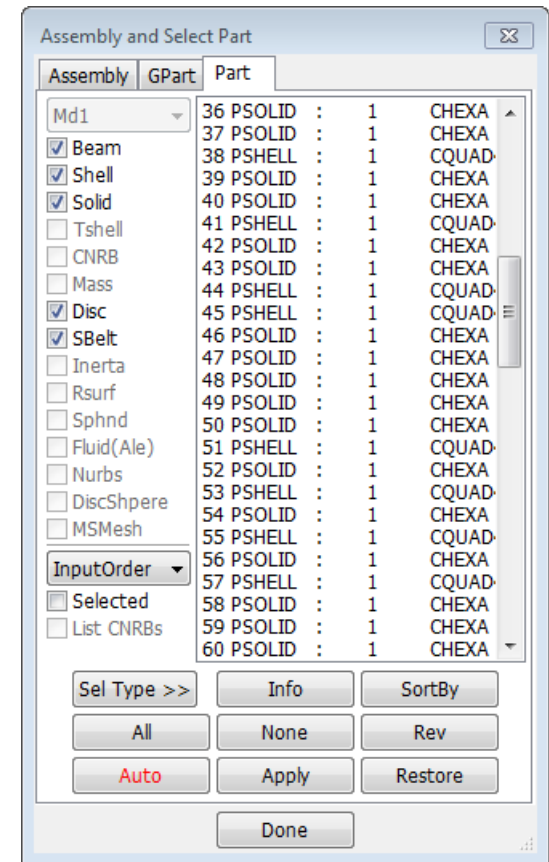
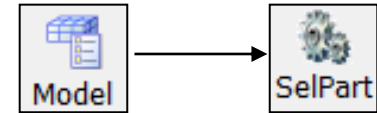
BySegm Select segments

BySurf Select surface (geometry)

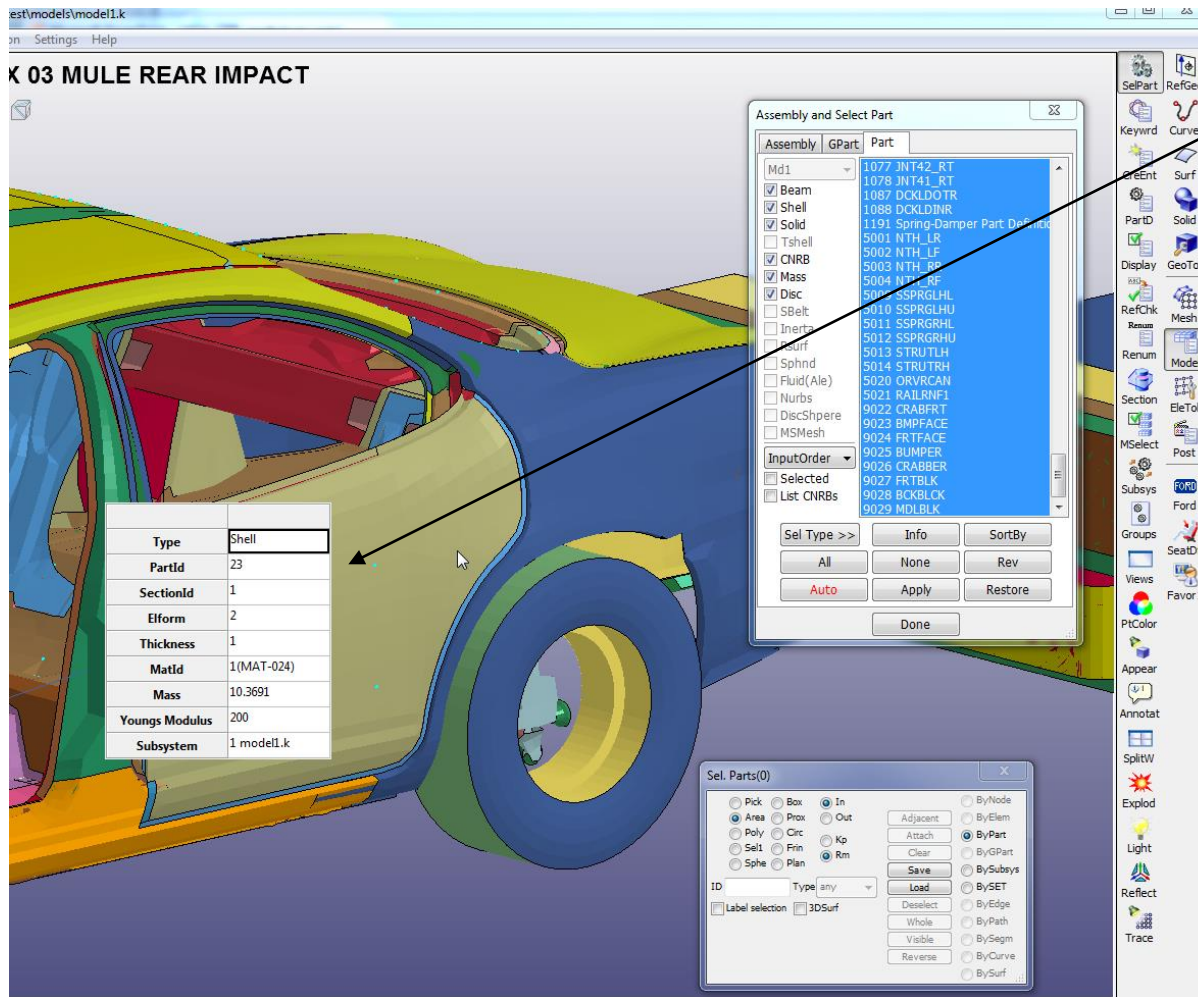
Model → Assembly and Select Part

Purpose: turn parts on/off (on=“active”)

- Parts listed by PID and name (or element type)
- Top drop-down menu selects current model
- Parts can be turned on/off by element type
- Use General selection to select parts
- Active parts can be Saved/Loaded from buffers (via Save button in General selection)
- Selected displays active parts only in list
- Info button launches Part Information interface for active parts
- SortBy button launches Part Sort interface



Model → Assembly and Select Part



Right click on part to show the part information and statistics

Keep mouse pressed and move to other part to show information dynamically

Keyword data gives more information than post-processing data

Model->Select Part->SortBy

Sort Part

	Type	PartId	PartKwd	PartName	SectionId	SectionName
1	Solid	1	PART	PSOLID : 1	1	
2	Solid	2	PART	PSOLID : 1	2	
3	Solid	3	PART	PSOLID : 1	3	
4	Solid	4	PART	PSOLID : 1	4	
5	Solid	5	PART	PSOLID : 1	5	
6	Solid	6	PART	PSOLID : 1	6	
7	Solid	7	PART	PSOLID : 1	7	
8	Solid	8	PART	PSOLID : 1	8	
9	Solid	9	PART	PSOLID : 1	9	
10	Solid	10	INERTIA	PSOLID : 1	10	
11	Solid	11	PART	PSOLID : 1	11	
12	Solid	12	PART	PSOLID : 1	12	
13	Shell	13	INERTIA	PSHELL : 1	13	
14	Solid	17	PART	PSOLID : 1	17	
15	Shell	18	PART	PSHELL : 1	18	
16	Shell	19	PART	PSHELL : 1	19	
17	Solid	20	PART	PSOLID : 1	20	
18	Shell	21	PART	PSHELL : 1	21	
19	Shell	22	PART	PSHELL : 1	22	
20	Solid	23	PART	PSOLID : 1	23	
21	Shell	24	INERTIA	PSHELL : 1	24	
22	Solid	25	PART	PSOLID : 1	25	
23	Shell	26	INERTIA	PSHELL : 1	26	
24	Solid	27	PART	PSOLID : 1	27	
25	Shell	28	INERTIA	PSHELL : 1	28	
26	Shell	30	INERTIA	PSHELL : 1	30	
27	Shell	32	INERTIA	PSHELL : 1	32	

Setting Column:

- ☒ Type
- ☒ PartId
- ☒ PartKwd
- ☒ PartName
- ☒ SectionId
- ☒ SectionName
- ☒ MatId
- ☒ MatName
- ☒ MatTypeName
- ☒ EosId
- ☒ Hrglass
- ☒ Elform
- ☒ Thickness
- ☒ Mass
- ☐ Cent_XYZ
- ☒ NumElem
- ☐ Area
- ☐ Volume
- ☐ NIP
- ☐ Density
- ☐ YoungModulus

Set Active

All None Rev Write Done

Each header can be clicked to sort the IDs of that column

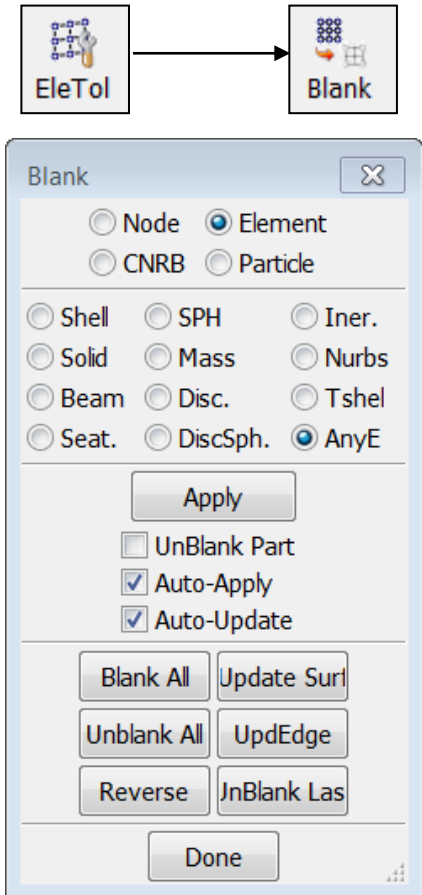
Options to select info to be tabulated

Highlighted parts can be set as active part in the graphics rendering

Element Tools → Blank

Purpose: temporarily hide or “mask” elements

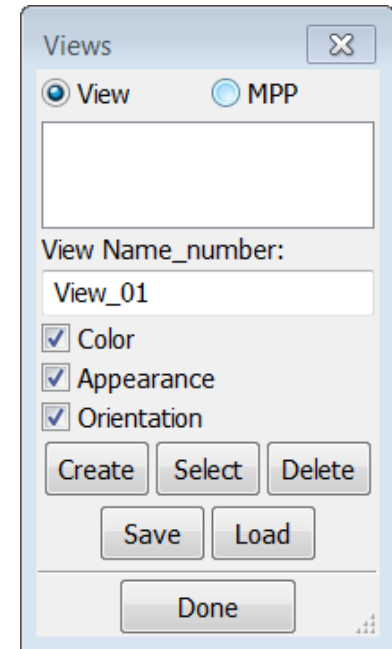
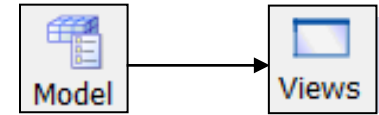
- Uses General Selection interface
- Entity types:
 - Node, Element, CNRB, Curve, Surface, Particle
- Element types:
 - Shell, Solid, Beam, TShell, SPH, Mass, Discrete, Seatbelt, Inertia, Nurbs, DiscSph, Any element
- UnBlank Part – click to restore partially hidden parts
- Auto-Apply – blank automatically (no need to click Apply button)
- Auto-Update – update display of solid surfaces and edges automatically (no need to click Update Surf or Update Edge)



Model → Views

Purpose: create views (based on color, appearance, orientation, and active parts)

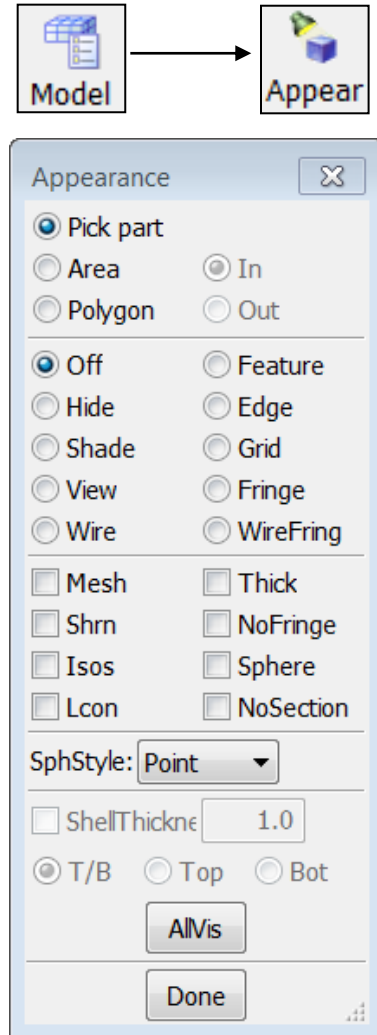
- Create – create a new view
- Select – load the selected view
- Delete – delete the selected view
- Save – save views to a binary file
- Load – load views from file



Model → Appearance

Purpose: modify the appearance of parts

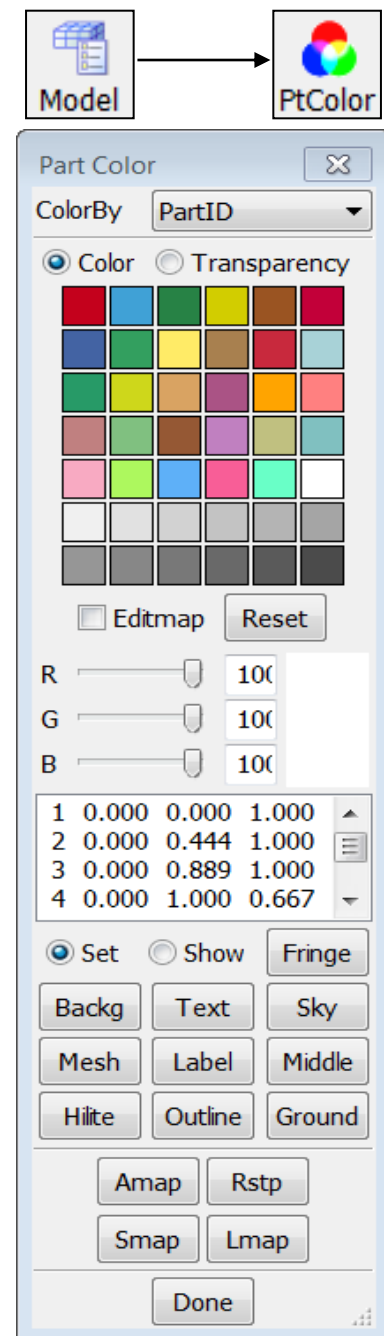
- Parts can be selected by Single/Area/Polygon
- Each selected part will have the selected rendering options applied
- Thick – draw shells with true thickness
- Sphere – draw solid part nodes as spheres (for EFG)
- AllVis – selected rendering options are applied to all active parts
- Render buttons are disabled while using Appear interface



Model → Part Color

Purpose: modify part color and transparency

- Set/Show colors for parts or other entities (background, text, mesh, labels)
- Sky (top), Middle, Ground (bottom) refer to background color (when View > Background > Tri Fade is selected)
- Select color from palette or form new color using RGB
- Editmap – to assign new color to the color map
- Fringe colors can also be changed – select a fringe color, then after the color selection, click Fringe button.
- Amap – apply the current color map to part drawing
- Rstp – restore transparency of all parts to opaque

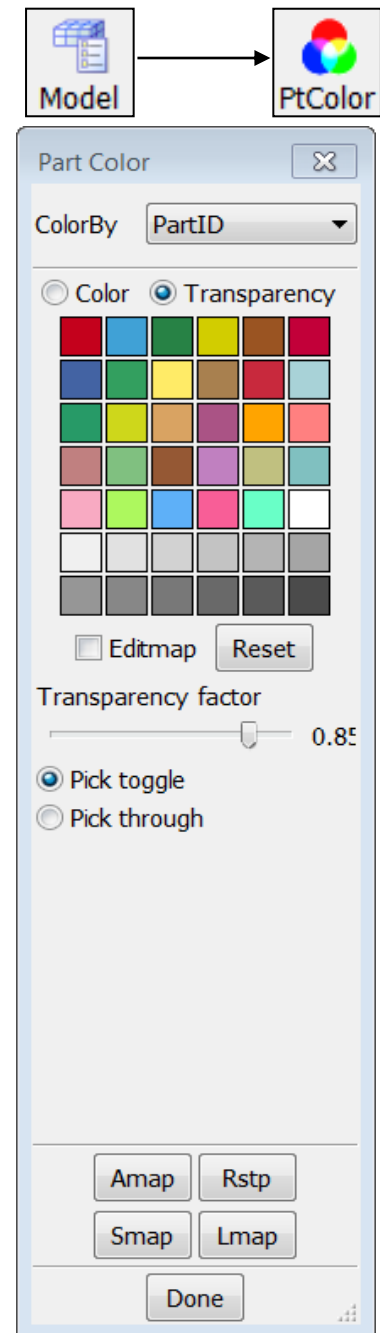


Model → Part Color

- Smap – save the current color map to a file
- Lmap – load color map from a previous saved file

Transparency – draw part in transparent mode, transparency factor:

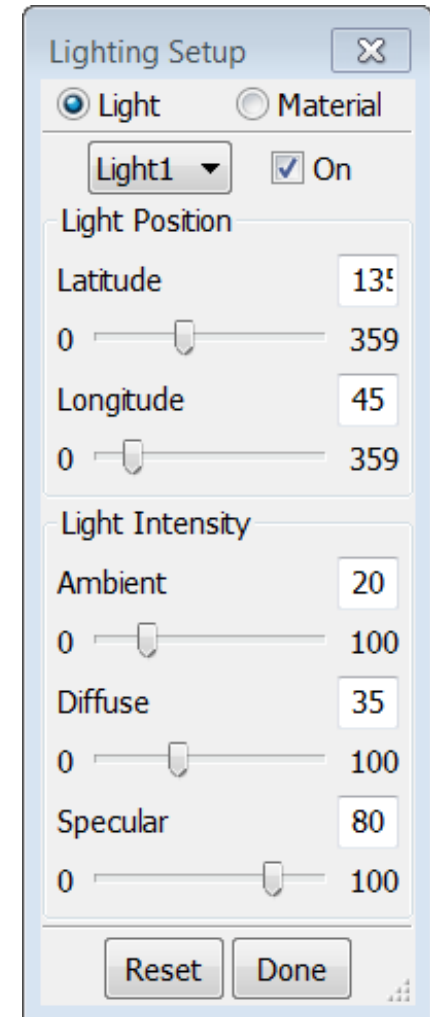
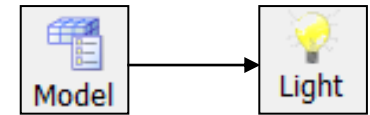
- 0.0 fully opaque (once is set to 0.0, need to click part again to turn it into transparent)
- 1.0 totally transparent
- Pick toggle – pick a part to toggle it from opaque to transparent and vice versa
- Pick through – pick through a part to set part that is behind this part



Model → Lighting Setup

Purpose: adjust model and material lighting

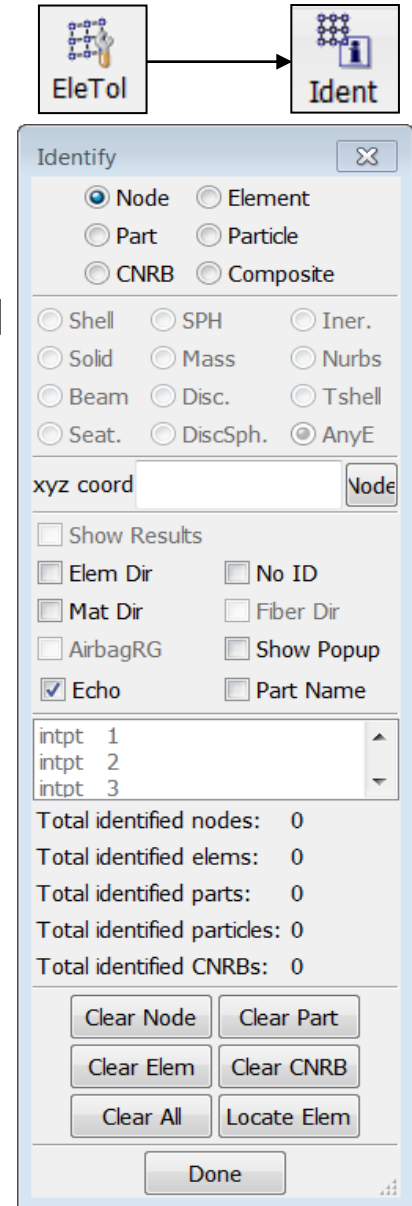
- Light – adjust model lights
 - 2 lights are on by default
 - Front – position (0.0, 0.0, 1.0)
 - Back – position (0.0, 0.0, -1.0)
 - Click the Latitude and Longitude to control the light position
 - Up to 8 more can be activated
 - The following properties can be adjusted for each
 - Ambient
 - Diffuse
 - Specular
- Material – adjust material lighting properties and the Shininess



Element Tools → Identify

Purpose: identify nodes/elements/parts

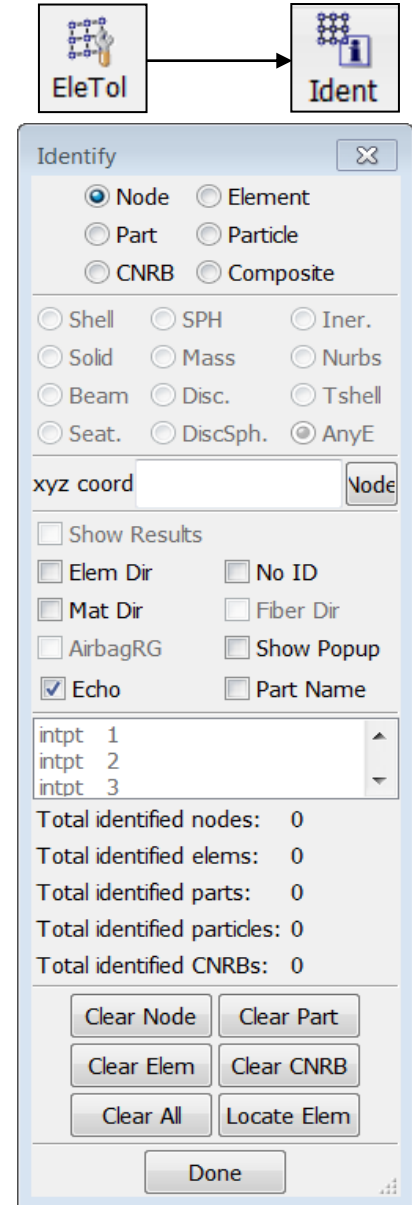
- Uses General Selection interface
- Entity types:
 - Node, Element, Part, Curve, Surface, Particle, constrained nodal rigid body (cnrb)
- Element types:
 - Shell, Solid, Beam, TShell, SPH, Mass, Discrete, Seatbelt, Inertia, Nurbs, DiscSph, Any
- Max. number of IDs that can be shown is 12800
- Can key in xyz coordinates to show position
- Can show element directions (shell and beam)
- Can show orthotropic material directions



Element Tools → Identify

Purpose: identify nodes/elements/parts

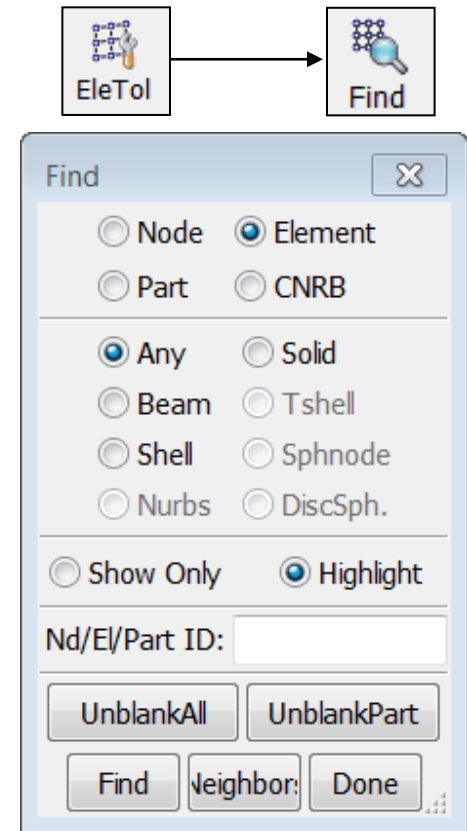
- Can show node/element results (after fringing in post-processing)
- Can show element results at integration points
- To show result on screen, go to pull down menu View, and turn on “Results on Screen”
- When identifying a very large no. of nodes or elements, make sure to turn off “Echo” and turn on “No ID”, otherwise will take a long time to come back



Element Tools → Find

Purpose: find nodes/elements/parts/Cnrb

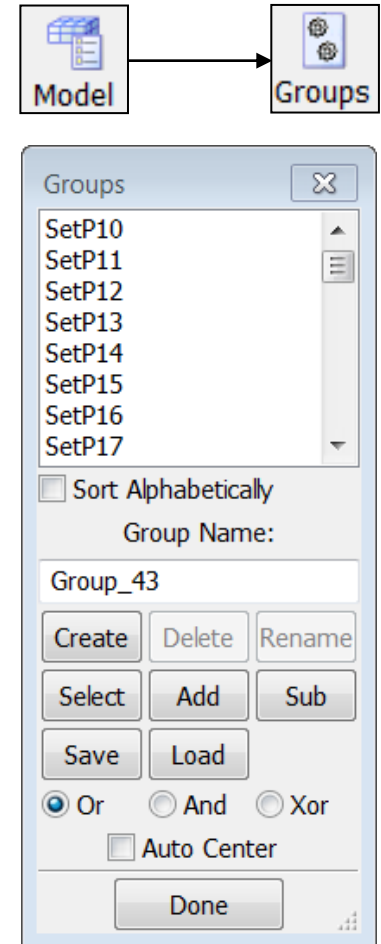
- Find any element by ID or specified type
- Show Only – shows found entity by itself
- Highlight – highlights found entity
- Neighbors – propagates to neighboring elements
- UnblankAll – turns on all elements and parts
- UnblankPart – turns on all elements in part that found node/element belongs to



Model → Groups

Purpose: create part groups

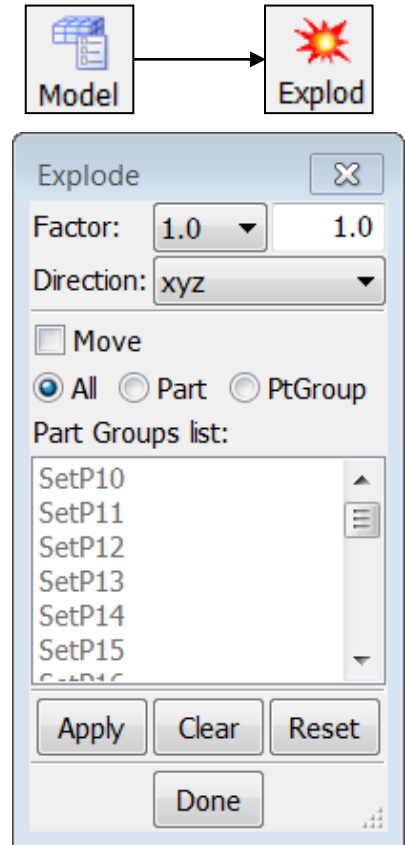
- Groups are automatically generated for all *SET_PART that exist in the model
- Create – create a new group from active parts
- Select – load the selected group
- Add – add a group to the current display
- Subtract – subtract a group from the current display
- Save – save groups to an ASCII file
- Load – load groups from file
(Save and Load effective for presentations)
- Or, And, Xor – used for Adding groups
- Auto Center – automatically center the selected group



Model → Explode

Purpose: separate (explode) parts for better visualization

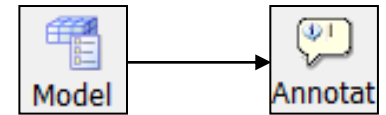
- Factor – scale factor for parts movement
- Direction – direction for part movement
- All – explode all parts
- Part – explode selected parts only
- PtGroup – explode a group of parts (defined using Page 1: Group)



Model → Annotation

Purpose: annotate Graphics and XY-Plot windows

- Position – interactively position text
- Arrow – add fixed arrow
- Nd Arrow – add arrow tied to node location
- Move – move annotations
- Text size, color, and orientation can be adjusted
- Annotations can be saved to / loaded from a file



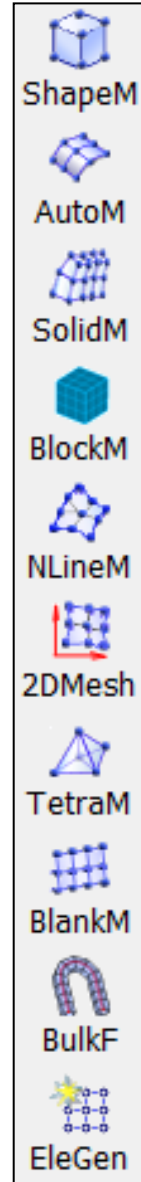
The screenshot shows a software window titled 'Annotation' with a close button (X) in the top right corner. The window contains a large text area at the top with the text 'HYBRID III RIGID (W/SPRING C'. Below the text area is a 'Text:' label followed by an empty input field. Underneath the input field are three buttons: 'Clear', 'Add', and 'Delete'. Below these buttons are three sections of controls, each with a label and three radio buttons: 'Annotation Type' with 'Pos' (selected), 'Arrow', and 'N Arrow'; 'Operation' with 'Move' (selected), 'Select', and 'Delete'; and 'Property' with 'Anno Colour' (set to '1'), 'Label Angle' (set to '0'), and 'Font Size' (set to '14'). At the bottom of the window are four buttons: 'Reset', 'Save', 'Load', and 'Done'.

Workshop 1

General Operations

- ❖ Parts on/off
- ❖ Render buttons (bottom toolbar)
- ❖ Group, appearance and view
- ❖ Identify and find
- ❖ Element blank(mask)
- ❖ Part color

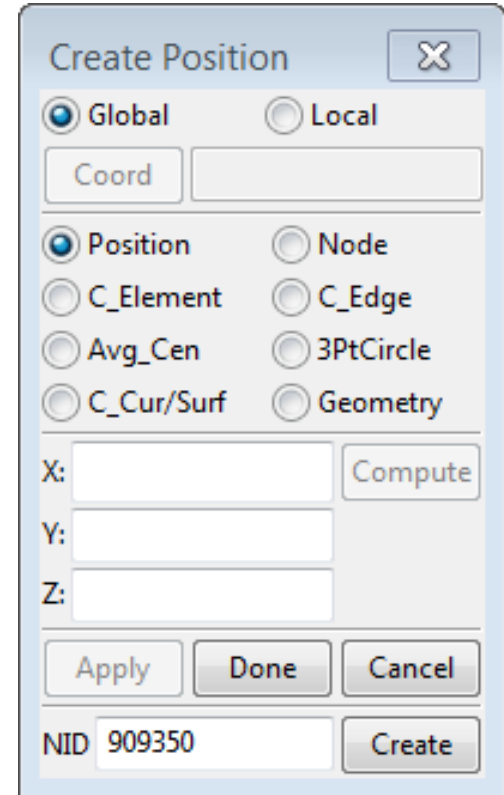
Mesh Generation



Create Position Interface

Purpose: define an XYZ location

- By picking a position
- By picking a node
- At an element center
- At an edge midpoint
- At the average center of nodes or elements
- At the center of a circle
- New *NODE can be created



The 'Create Position' dialog box is shown with the following options and fields:

- Global** (selected) and **Local** (unselected) radio buttons.
- Coord** button and an empty text field.
- Position** (selected), **Node** (unselected), **C_Element** (unselected), **C_Edge** (unselected), **Avg_Cen** (unselected), **3PtCircle** (unselected), **C_Cur/Surf** (unselected), and **Geometry** (unselected) radio buttons.
- X:**, **Y:**, and **Z:** text input fields.
- Compute** button.
- Apply**, **Done**, and **Cancel** buttons.
- NID** 909350 and a **Create** button.

Create Direction Interface

Purpose: define a direction

- By picking 2 positions
- By picking 2 nodes
- Along an element edge
- Along an element normal
- Direction can be rotated about global or local coordinate system
- New *DEFINE_VECTOR can be created

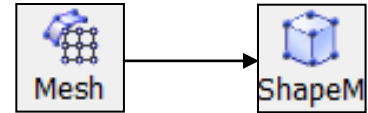
The 'Create Direction' dialog box is shown with the following settings:

- Global** (selected) / Local
- Coord** (empty text box)
- 2Position** (selected) / 2Nodes
- Edge (unselected) / El_normal (unselected)
- CurveTangent (unselected) / Surf_normal (unselected)
- Rotate (unselected) / 3Nodes (unselected)
- Position1** (0.0) / **Position2** (0.0)
- X** (0.0) / **Y** (0.0) / **Z** (1.0)
- Clear** / **Reverse**
- Title** (empty text box)
- Vector ID** (1) / **Create**
- Done** / **Cancel**

Mesh → Simple Shape Meshing

Purpose: mesh basic geometries

- Box_Solid, Box_Shell – define min/max coordinates and mesh density
- 4N-Shell – define 4 corners and mesh density
- Sphere_Solid, Sphere_Shell – define center, radius and mesh density (quarter circle)
- Cylinder_Solid, Cylinder_Shell – define center, direction, radius, length and mesh density (if number of elements in circumferential direction is multiple of 4, butterfly mesh will be create)
- Circle_Shell – define outer radius, inner radius (if hole is desired), angle (to create an arc), mesh density, and normal direction



Shape Mesher

Entity: Sphere Solid

Radius: 5

Density: 5

Center[Position]

x	y	z
0	0	0

Direct1

dx	dy	dz
1	0	0

Direct2

dx	dy	dz
0	1	0

Target Name

Target Part ID: 1524

Start Element ID: 160040

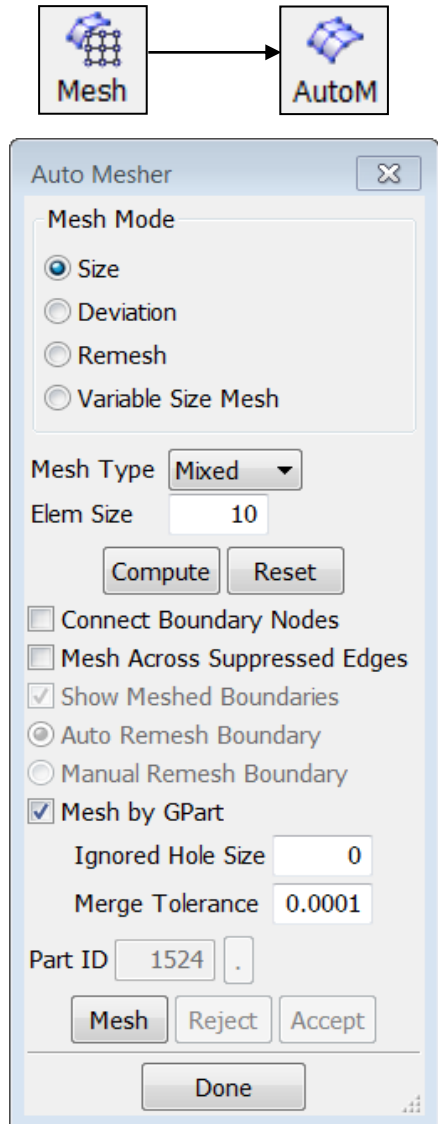
Start Node ID: 909350

Create Reject Accept

Done

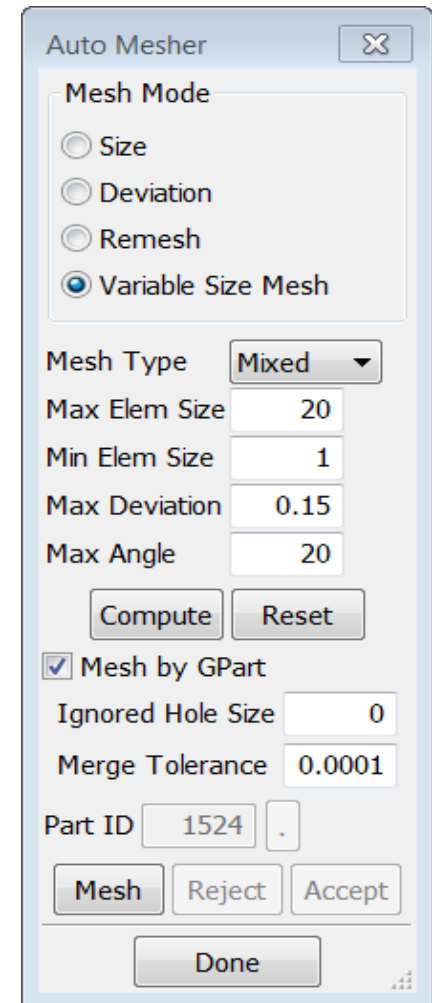
Mesh → Auto Mesher (Size)

- Mesh by Size creates uniformly sized elements
- Can be mix of quad and tri element or all tri only
- Compute button will give rough estimate of element size
- Connect Boundary Nodes will connect with the nodes on existing mesh
- Mesh Across Suppressed Edges will ignore edges that is suppressed in the geometry Heal->Edge
- Mesh by GPart will create separate LS-DYNA part for each geometry part
- Ignore Hole Size, hole size smaller than this value will be ignore and be filled with elements
- Merge Tolerance controls nodes being merged when close to each other



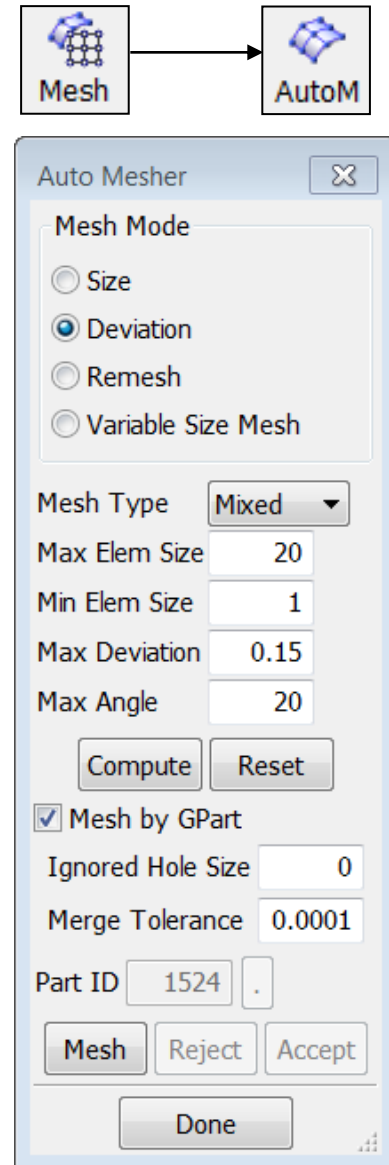
Mesh → Auto Mesher (Variable size)

- Mesh by variable size will create mesh that has smaller elements at high curvature area and larger elements on flat surfaces
- It is similar to “By Size” method, in general gives better results
- 4 parameters control the element size
 - Max element size
 - Min element size
 - Max deviation
 - Max angle



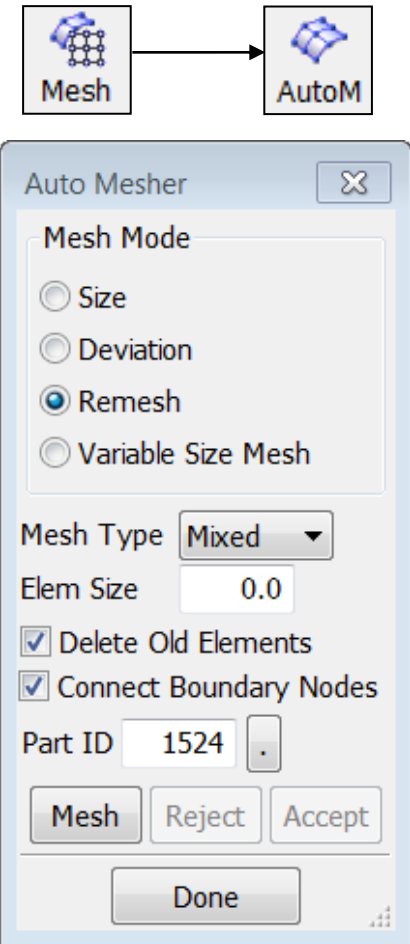
Mesh → Auto Mesher (Deviation)

- Mesh by Deviation is for tooling in metal forming stamping applications
- Deviation mode creates small elements on curved surfaces and large elements on flat surfaces
- 4 parameters control the element size
 - Max element size
 - Min element size
 - Max deviation
 - Max angle



Mesh → Auto Mesher (Remesh)

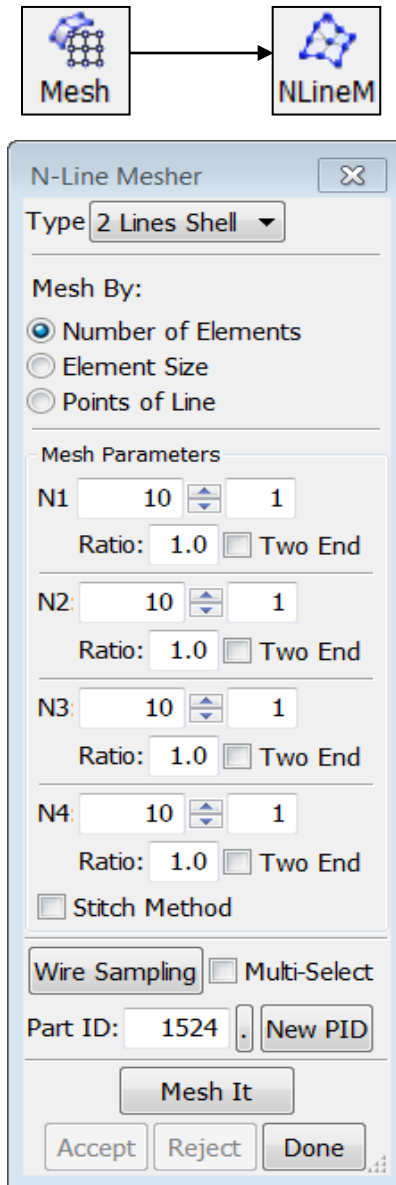
- Remesh mode allows user to remesh part of the model
- Options to delete the original elements and connect to the nodes on the existing mesh



Mesh → N-Line Mesher

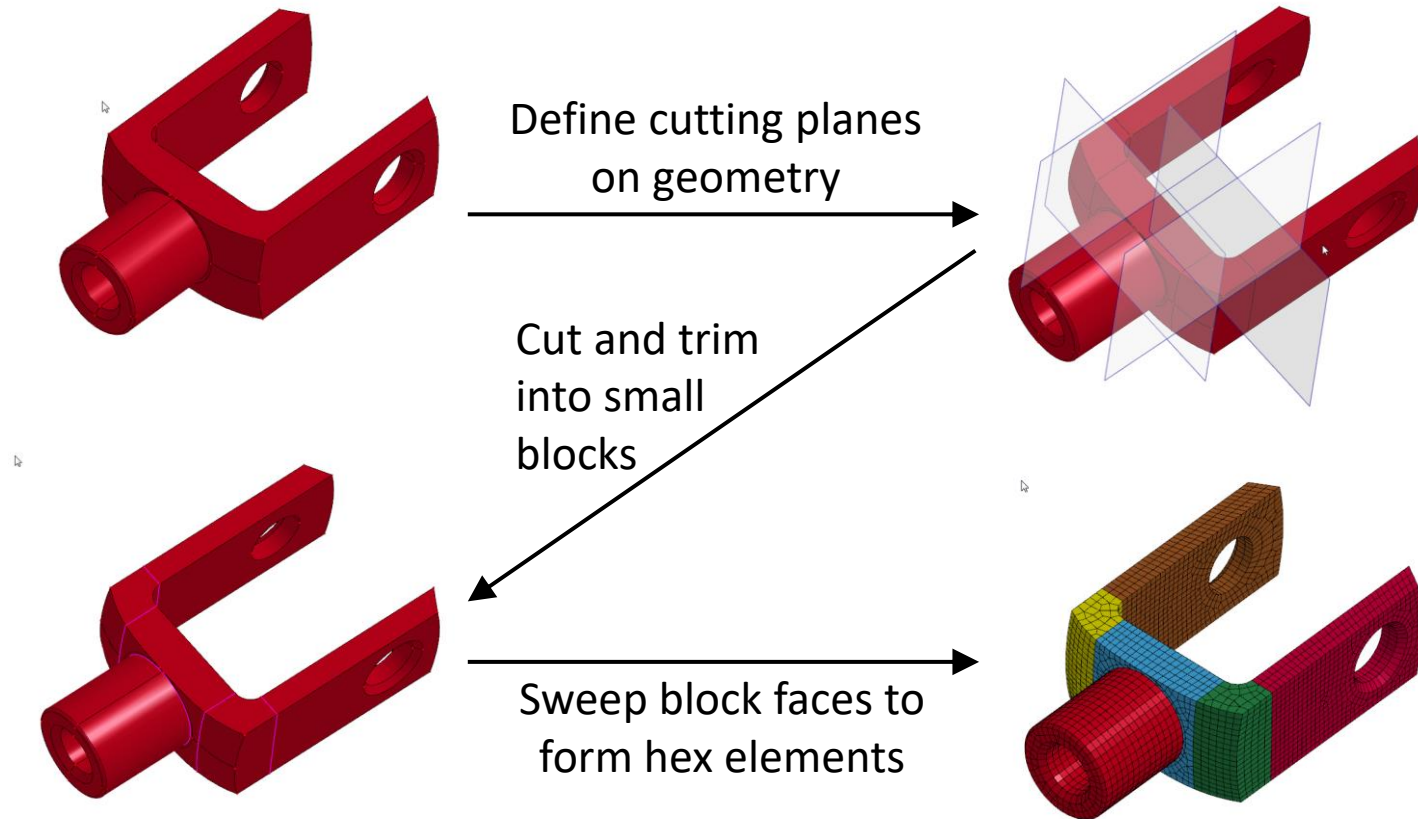
Purpose: create mesh from 2/3/4 lines

- 2 Line Shell – create mesh between 2 lines
- 3 Line Shell – create mesh between 3 lines
- 4 Line Shell – create mesh between 4 lines
 - Enter number of elements on each edge
 - Or enter element size
 - Or use points on lines to create elements
- Line Sweep – sweep one line along another line



Mesh → Solid Mesher

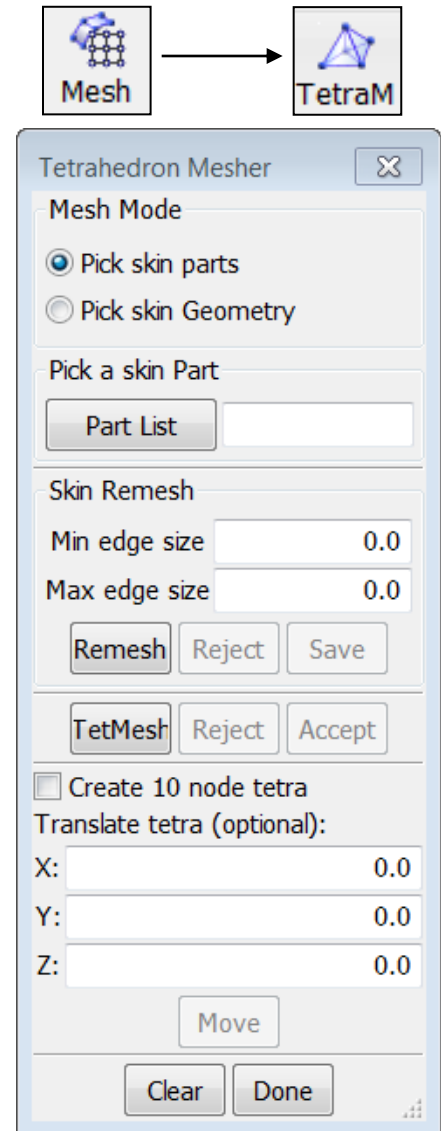
- Solid meshing by blocks
- Uses cut & dice method followed by sweeping



Mesh → Tetrahedron Mesher

Purpose: create 4-node solid elements (tetrahedron) inside an enclosed shell volume

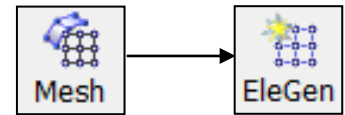
- Pick Skin Parts – shell parts that form a water tight enclosure, multiple parts are allowed as long as each part form the enclosure
- Requirement: skin normal must be aligned
- Skin can be re-meshed to give better solid tetrahedron mesh
- Pick Skin Geometry – surface geometry can be used instead of shell part, LSPP will first create shell mesh internally, element size is needed for shell meshing
- 10-node tets can also be created
- Created Solid mesh can be translated in space



Mesh → Element Generation (Beam)

Purpose: create beam elements

- On an edge
- On a curve
- By dragging nodes
- By spinning nodes
- Along shell diagonals



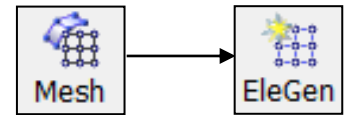
The 'Element Generation' dialog box is shown with the following settings:

- Beam** is selected (radio button).
- Element ID:** 160040 (with a 'New EID' button).
- Part ID:** 1524 (with a 'Pick PID' checkbox and a 'New PID' button).
- New node:** 909350 (with a 'New NID' button).
- Beam By:** Edge (dropdown menu).
- Third Node:** (checkbox) with a 'Direction' button and three input fields for X, Y, and Z.
- Node ID:** (checkbox) with an input field.
- Buttons:** Create, Reject, Accept, and Done.

Mesh → Element Generation (Shell)

Purpose – To create shell elements from:

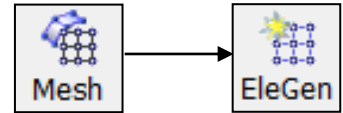
- Solid Surface - On all outer surfaces of a solid
- Solid Face - On individual solid faces
- Edge Drag - By dragging an edge
- Edge Spin - By spinning an edge
- Edge Extend - By extending an edge
- Curve Drag - By dragging a curve
- Curve Spin - By spinning a curve (geometry curve)
- Segment Set – Create shell element from segment set
- Fill Holes - By filling holes
- 4Nodesto8Nodes – Create 8-node shell element
- Helix – Create helix shell loop
- Wrap – Wrap over open parts
- Solid/Tshell Midplane – Create shell from a single layer solid part at the mid-plane of the solid part



The 'Element Generation' dialog box is shown. It has a title bar with a close button. Inside, there are three radio buttons: 'Beam', 'Shell' (which is selected), and 'Solid'. Below these are fields for 'Element ID:' (containing '160040') and 'Part ID:' (containing '1524'). Each field has a 'New' button next to it. There is also a 'New node:' checkbox and a field containing '909350' with a 'New NID' button. A 'Shell By:' dropdown menu is set to 'Solid Surface'. At the bottom, there are 'Create', 'Reject', 'Accept', and 'Done' buttons.

Mesh → Element Generation (Solid)

Purpose: create solid elements



- Solid Face Drage – Create another solid By dragging a solid face
- Solid Face Offset - By offsetting solid face
- Solid Face Spin - By spinning solid face
- Shell Drag – By dragging a shell element
- Shell Offset – By offsetting a shell element
- Shell Spin – By spinning a shell element
- Shell Thickness – Apply the thickness of shell
- Two Shell Sets – Take 2 sets of shells to form solid between them (must have identical mesh connectivity)
- Shell Sweep – Sweep shell element along a curve
- Tet4 to Tet10 – Change 4-node tet to 10-node tet
- Hex to Tet4 – Convert hexahedron into 4-node tetrahedron
- Helix – Create helix form of solid elements

The 'Element Generation' dialog box is shown. It has a title bar with a close button. Inside, there are three radio buttons: 'Beam', 'Shell', and 'Solid', with 'Solid' selected. Below these are fields for 'Element ID:' (160040), 'Part ID:' (1524), and 'New node:' (909350), each with a 'New' button. A 'Solid By:' dropdown menu is set to 'Solid_Face_Drag'. Below this are 'Thickness' and 'Segment' fields, both set to 1. There are three buttons labeled 'Position', 'X', 'Y', 'Z', and 'Direction'. At the bottom are 'Create', 'Reject', 'Accept', and 'Done' buttons.

Workshop 2

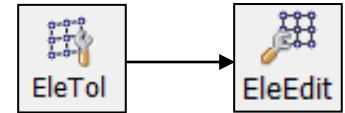
Mesh Generation

- ❖ Shape mesh
- ❖ N-line mesh
- ❖ Auto mesh
- ❖ Solid mesh

Pre-Processing

Element Tool → Element Editing

Purpose: modify an existing mesh by editing elements



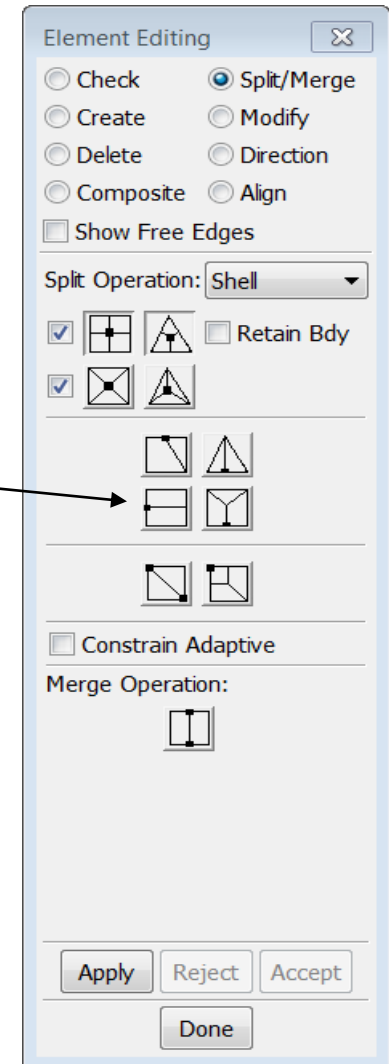
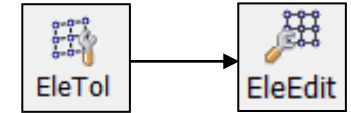
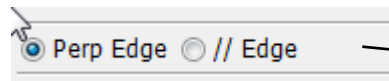
■ Element Editing Tools

- Check (quality)
- Create (beam, shell, tetra, penta, hexa, discrete)
- Delete
- Split / Merge
- Modify (thickness)
- Direction (change material direction for orthotropic materials)
- Composite (special operation to model composite laminated shell elements, see tutorial no. 4 in the model section)
- Align (re-orient element connectivity according to a seed element)

The screenshot shows the 'Element Editing' dialog box. It has a title bar with a close button. Inside, there are two columns of radio buttons: 'Check', 'Create' (selected), 'Delete', 'Composite', 'Split/Merge', 'Modify', 'Direction', and 'Align'. Below these is a checkbox for 'Show Free Edges'. A dropdown menu for 'Elem Type' is set to 'Shell'. There are input fields for 'Elem ID' (value 1) and 'PID' (value 1), each with a 'New' button. Below these are four checkboxes for 'Node1' through 'Node4', with 'Node1' checked. At the bottom are checkboxes for 'Tria Only' and 'Replace'. At the very bottom are three buttons: 'Redefine', 'Clear', and a group containing 'Reject', 'Accept', and 'Done'.

Element Tool → Element Editing (Split/Merge)

- Shell elements can be split into quads or tris
- Constrain Adaptive button – If checked (*CONSTRAINED_ADAPTIVITY automatically created when splitting deformable elements)
- Click edge of element for splitting into 2 elements, can be perpendicular or parallel to the picked edge
- Quad can also be split into Tris
- Two quads can be merged into a single quad
- Solid elements can also be split



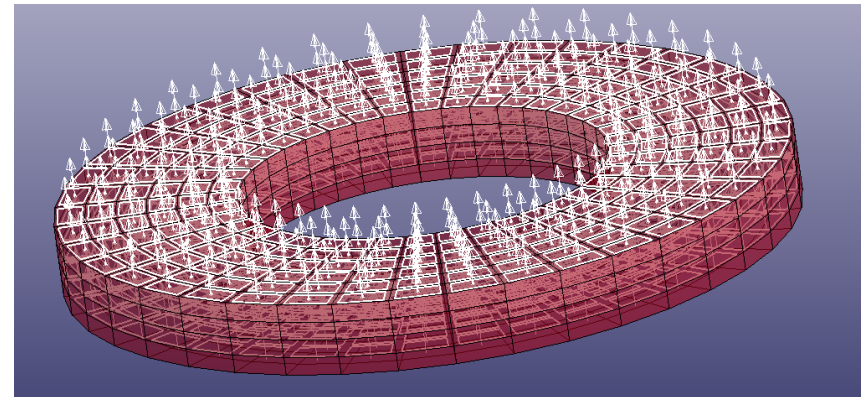
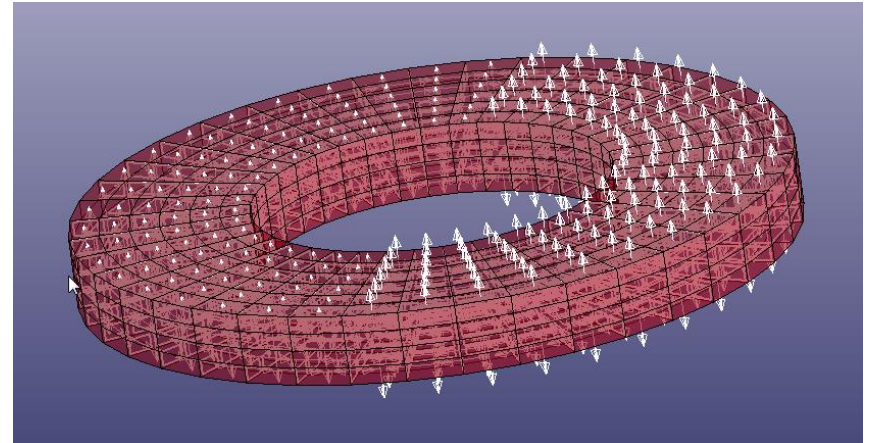
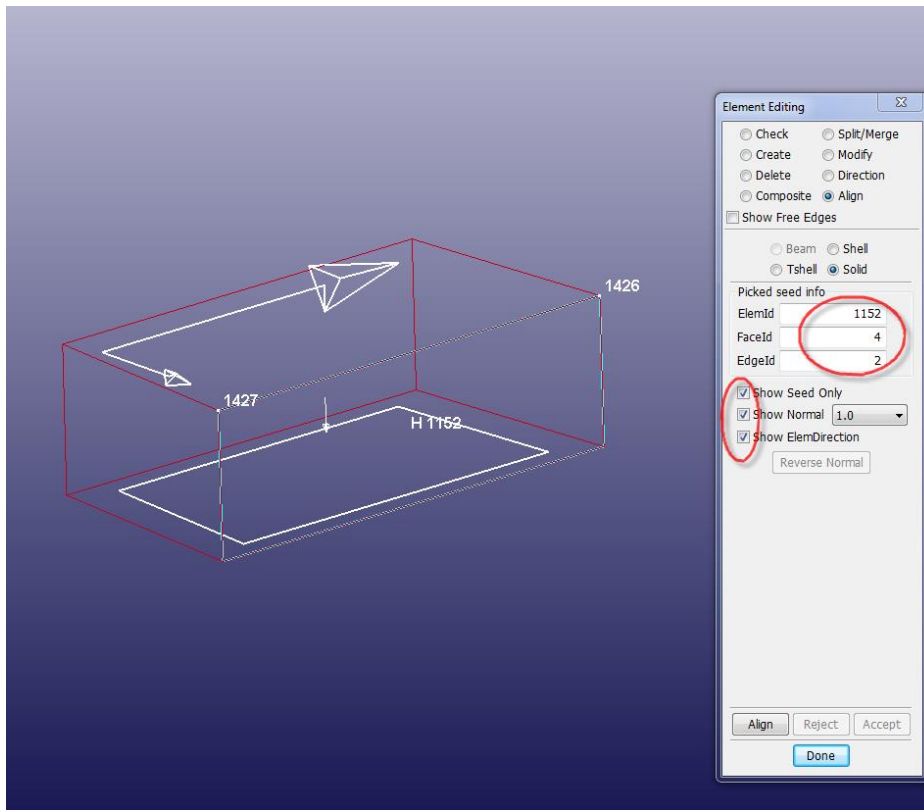
Element Tool → Element Editing (Align)

- Align – to re-align the connectivity of a group of shell/solid/tshell elements such that the orientation of the elements will be consistent
- Pick the face and edge of a seed element, the picked face will be used as face one, and the picked edge will be used as n1->n2 (first edge)
- Show Seed only will show the picked element and allow user to select different face/edge
- Show normal, show direction will show element orientation

The screenshot shows the 'Element Editing' dialog box with the 'Align' tab selected. The dialog has a title bar with a close button. Inside, there are two columns of radio buttons for selecting the element type: 'Check', 'Create', 'Delete', 'Composite', 'Split/Merge', 'Modify', 'Direction', and 'Align' (which is selected). Below these is a checkbox for 'Show Free Edges'. Further down, there are radio buttons for 'Beam', 'Shell', 'Tshell', and 'Solid' (which is selected). A section titled 'Picked seed info' contains three text input fields for 'ElemId', 'FaceId', and 'EdgeId'. Below these is a checked checkbox for 'Modify seed connectivity'. There are three checkboxes: 'Show Seed Only', 'Show Normal' (with a value of 1 in a spinner), and 'Show ElemDir.(nd1-2 and'. Below these is a 'Reverse Direction' button. At the bottom, there are four buttons: 'Align', 'Reject', 'Accept', and 'Done'.

Element Tool → Element Editing (Align)

Solid/Tshell connectivity re-alignment – to re-align the connectivity of a group of solid/tshell elements such that the orientation of the elements will be consistent

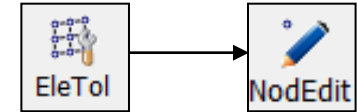


Element Tool → Node Editing

Purpose: modify an existing mesh by editing nodes

- Node Editing Tools

- Create – to create new nodes, use Advance button to create nodes using different methods
- Delete – to delete unreferenced (no other element or entity attached to it)
- Replace – to replace one node with another node (merge) or take the center point of the 2 nodes
- Align – to align nodes that is close to a line formed by 2 points
- Modify (xyz coordinates)

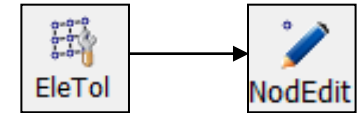


The screenshot shows the 'Node Editing' dialog box with the following options and controls:

- Radio buttons:** Create (selected), Replace, Delete, Align, Modify.
- ☐ Show Free Edges
- Radio buttons:** ByOne (selected), ByTwo, OnCurve.
- Node Id:** 1, with a 'NewID' button.
- Coordinate fields:**
 - ☒ Node X: 0
 - ☐ Position(FE) Y: 0
 - ☐ Position(Geom) Z: 0
- Buttons:** Advanced, Create, Reject, Accept, Done.

Element Tool → Node Editing (Replace)

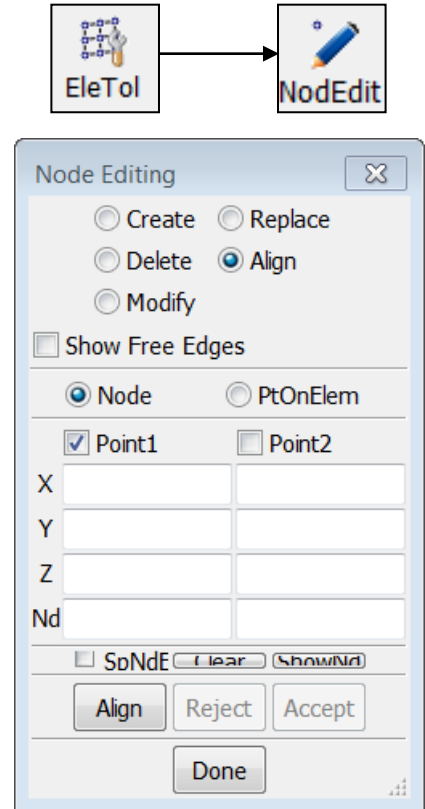
- Two nodes replacement
 - Merge two nodes into one
 - Option to choose final position of merged node (Node1, Node2, MidPoint)
 - Single pick or area select
- Multiple (Many) nodes replacement
 - Merge multiple nodes into one
 - Option to choose final position of merged node (Center, Node, Position)
 - General Selection interface used to select nodes



The screenshot shows a 'Node Editing' dialog box with a close button (X) in the top right corner. It contains several radio button options: 'Create', 'Replace' (selected), 'Delete', 'Align', and 'Modify'. There is a checkbox for 'Show Free Edges'. Below these are two groups of radio buttons: 'Two' (selected) and 'Many', and 'Pick Node' (selected) and 'Area'. Under 'Pick Node', there are two checked checkboxes: 'Node1' and 'Node2', each followed by a text input field. Below these are three radio button options: 'At Node1', 'At Node2' (selected), and 'MidPoint'. At the bottom are four buttons: 'Clear', 'Reject', 'Accept', and 'Done'.

Element Tool → Node Editing (Align)

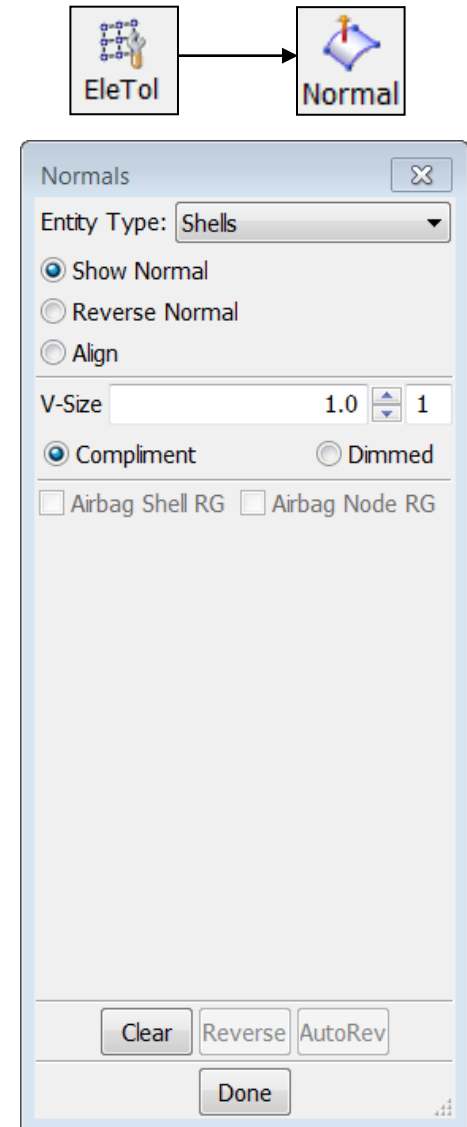
- Define line by picking two points
- Select nodes that will be snapped to the line
- Option to use Nodes or Point on Element
- SpNdEQ – special option to put node in equal spacing when align to the line



Element Tools → Normals

Purpose: show/reverse/align element normal

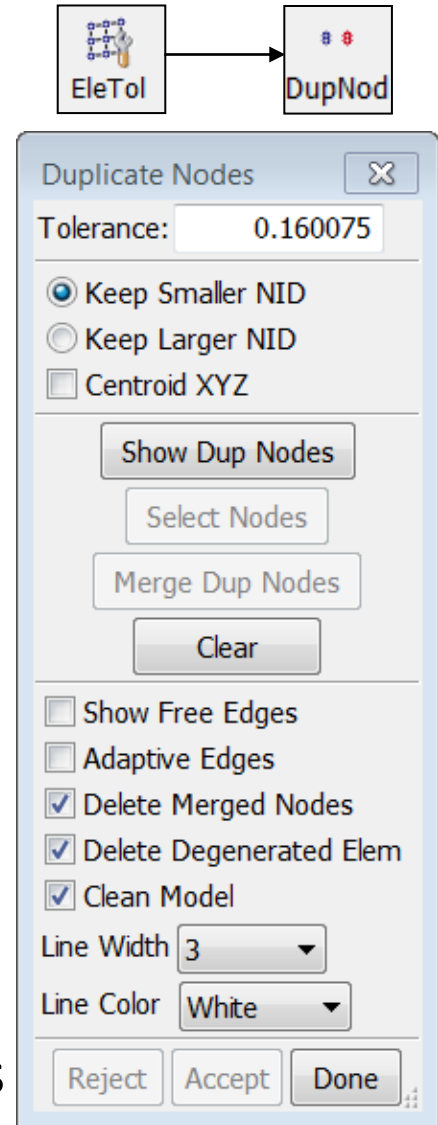
- Apply to Shell/Segment/Tshell/Cohesive/Solid
- For Shell element, positive normal has different color than negative normal
- General Selection interface can be used to manually show or reverse normal vectors
- Automatic alignment (Auto Reverse) can be performed by picking a “seed” element



Element Tools → Duplicate Nodes

show and merge duplicate (coincident) nodes

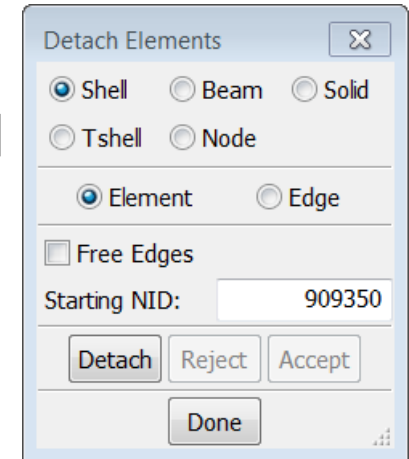
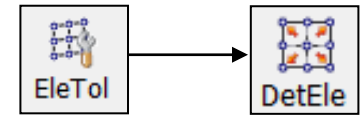
- Tolerance is automatically calculated but can be overridden
- Option to keep smaller or larger Node ID
- Option to put new node at the xyz centroid of all merged nodes
- Show Dup Nodes – will show all the nodes that meet the tolerance criterion
- Select Nodes – allow user to select subset of nodes shown
- Merge Dup Nodes – will perform the merging of the duplicated Nodes
- Option to delete degenerated elements (elements with zero area)



Element Tools → Detach

Purpose: detach elements from an existing mesh

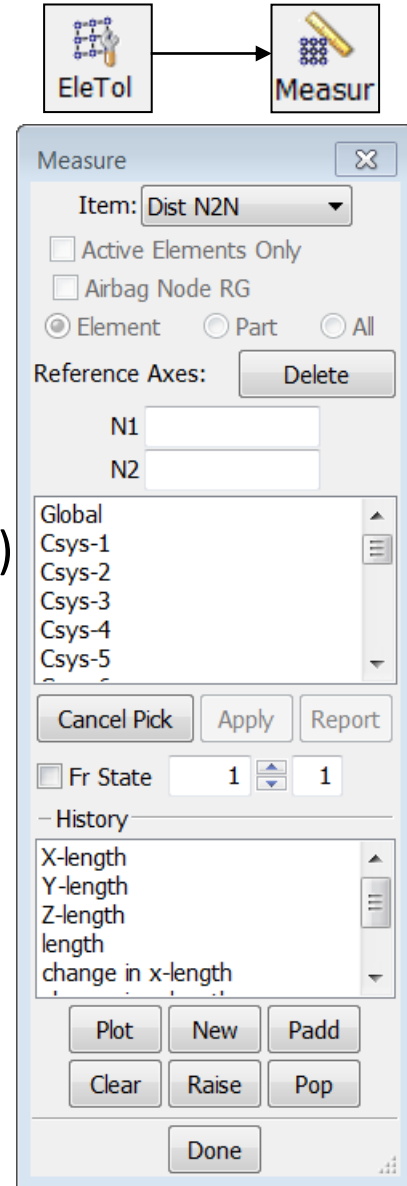
- Shell, Beam, Solid, Tshell – detach elements by type
- Node – detach all elements connected to a node
- Element – Select group of elements to be detached
- Edge – Select edges of elements to be detached
- Starting NID – Enter starting ID for new nodes



Element Tools → Measure

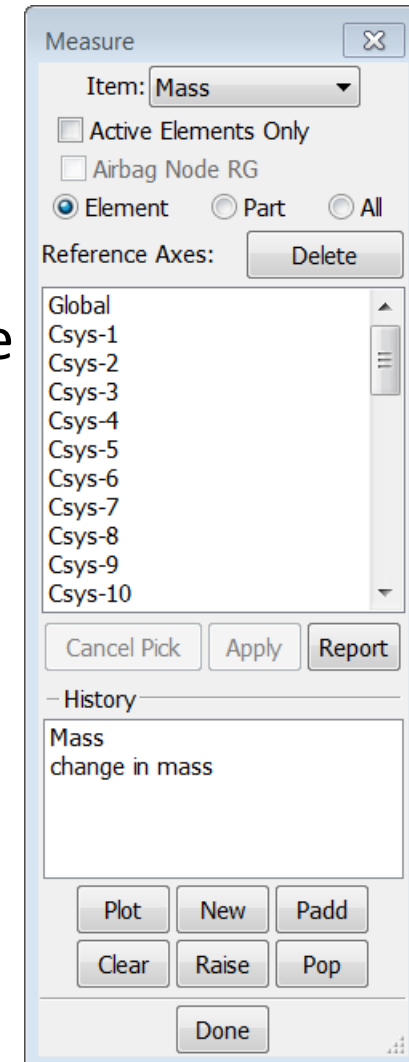
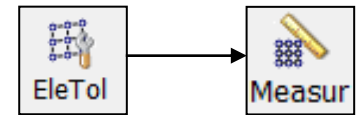
Purpose: take measurements of various items

- Item – list of available measurements
 - Coordinate – xyz coordinate
 - Dist N2N – distance between 2 nodes
 - Dist N2S – distance between a node and a surface
 - Dist P2P – distance between 2 points
 - Dist E2E 4Node – distance between edge to edge (4nodes)
 - Angle 3Node – angle between 3 nodes
 - Angle 4node – angle between 2 lines formed by 4 nodes
 - 3Pt Radius – the radius formed by 3 nodes
 - Area / Volume / Mass / Inertia
 - Ang Vel – angular velocity
 - Create Axes – to create a local coordinate systems
 - Separation – measure distance between two parts in fringe color



Element Tools → Measure *(continued...)*

- Active Elements Only – for certain quantities (e.g., Area, Volume, Mass, Inertia)
- Element/Part/All – measure by element, by part, or all
- For shell part volume, beside the volume computed by (area x thickness), it also computes the enclosure volume if the part form a water tight enclosure
- Cancel pick – remove last picked entity
- Apply – take measurement
- Reference Axes – select current reference axes for measurement (define axis using Item: Create Axis)
- History – depends on selected item



Workshop 3

Model Editing

- ❖ Model free edge
- ❖ Duplicate nodes merge
- ❖ Node and element Editing
- ❖ Fill holes
- ❖ Element normal align
- ❖ Save a keyword file

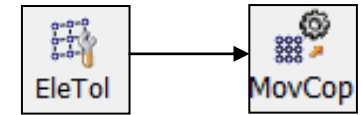
Pre-Processing

(continued...)

Element Tools → Move/Copy

Purpose: move or copy elements from one part to another

- Use General Selection interface to select elements
- Target part can be a non-existent
- Starting IDs for new elements and nodes can be specified when performing a copy

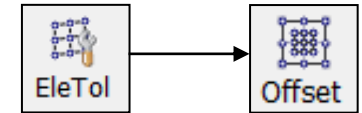


A screenshot of a software dialog box titled 'Move or Copy'. It features a close button (X) in the top right corner. Below the title bar, there are two radio buttons: 'Move' (which is selected) and 'Copy'. A checkbox labeled 'Pick Target Part' is located below the radio buttons. Under this checkbox, there is a text field for 'PID:' followed by a 'Plist' button. Below the 'PID' field is another text field for 'PName:'. Further down, there are two more text fields: 'Starting element ID:' with the value '160040' and 'Starting node ID:' with the value '909350'. At the bottom of the dialog, there are four buttons: 'Apply', 'Reject', 'Accept', and 'Done'.

Element Tools → Offset

Purpose: offset shells (always in normal direction)

- Offset Distance – distance to offset
- Element normals must be aligned before offsetting
- Beware of offsetting small radii inward
- Elements can be copied while offsetting



A screenshot of the 'Offset' dialog box. The dialog has a title bar with a close button (X). The main area contains the following fields and options:

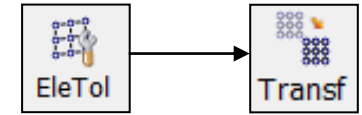
- 'Offset Dis.:' with an empty text input field.
- 'Curv. Sens.:' with a text input field containing '0.9999'.
- A checkbox labeled 'Copy Elem' which is currently unchecked.
- 'No of Copies:' with a text input field containing '1'.
- A checkbox labeled 'Part List' which is currently unchecked.
- 'Starting EID:' with a text input field containing '160040'.
- 'Starting NID:' with a text input field containing '909350'.
- A checkbox labeled 'Transfer' which is currently unchecked.
- A checkbox labeled 'Advanced' which is currently checked.

At the bottom of the dialog, there are four buttons arranged in two rows: 'Offset -' and 'Offset +' in the top row, and 'Reject' and 'Accept' in the bottom row. Below these is a single 'Done' button.

Element Tools → Transform (Translate)

Purpose: translate nodes/elements/parts

- Translate Distance – distance to translate
- X,Y,Z directions can be in global or local system
- N1-N2 – translate in the direction from N1 to N2
- N1-N2-N3 – translate in the direction normal to the plane formed by N1, N2, and N3
- N1-to-N2 – translate using the distance and direction between N1 and N2
- Pt-to-Pt – translate using the distance and direction between P1 and P2
- Sh-Normal – pick a shell element and use its normal as the direction of the translation

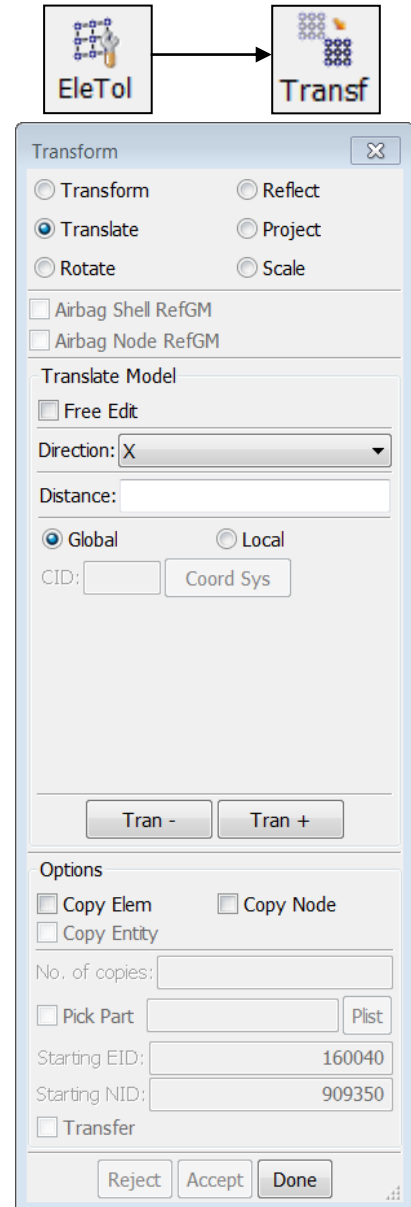


The screenshot shows the 'Transform' dialog box with the following settings:

- Transform** (selected), **Reflect**, **Project**, **Rotate**, **Scale**
- ☐ Airbag Shell RefGM
- ☐ Airbag Node RefGM
- Translate Model**
- ☐ Free Edit
- Direction:** X
- Distance:** [empty field]
- ☒ **Global**, ☐ Local
- CID:** [empty field] **Coord Sys**
- Tran -** **Tran +**
- Options**
- ☐ Copy Elem, ☐ Copy Node
- ☐ Copy Entity
- No. of copies:** [empty field]
- ☐ Pick Part [empty field] **Plist**
- Starting EID:** 160040
- Starting NID:** 909350
- ☐ Transfer
- Reject** **Accept** **Done**

Element Tools → Transform (Translate)

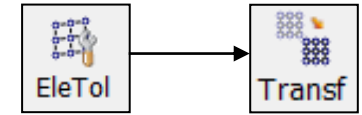
- Free Edit – after distance, direction and node selection, user can drag the selected nodes on screen
- Copy Elem – copy elements while translating
- Copy Node – copy nodes while translating
- Transfer – transfer selection to copied elements
- Multiple copies can be created
- New elements can be placed in a new part ID
- User can enter starting element ID for new elements
- User can enter starting node ID for new nodes



Element Tools → Transform (Rotate)

Purpose: rotate nodes/elements/parts

- Rot. Angle – angle to rotate in degree
- X,Y,Z axes can be in global or local coordinates
- N1-N2 – rotate about an axis from N1 to N2
- N1-N2-N3 – rotate about the normal formed by N1, N2, and N3
- Origin of rotation can be a node or XYZ location
- Elements/Nodes can be copied while rotating



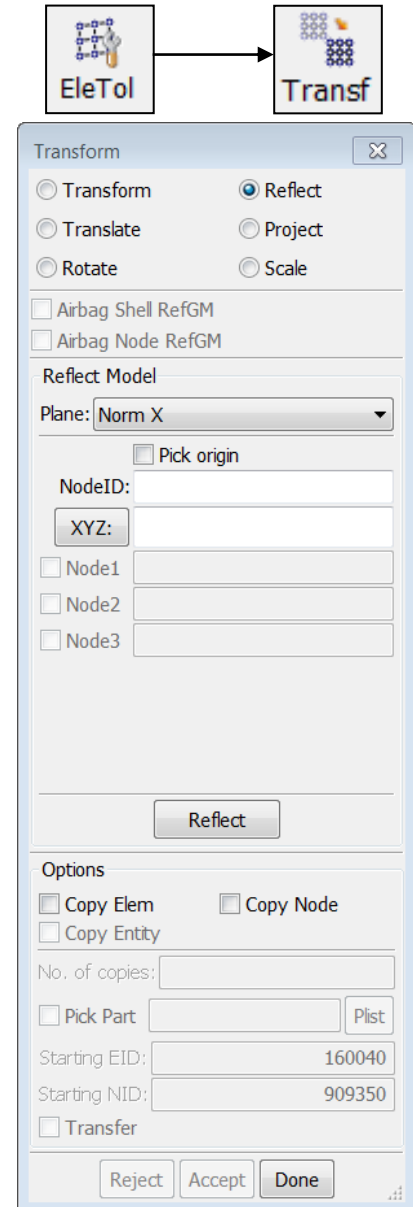
The screenshot shows the 'Transform' dialog box with the following settings:

- Transform** (selected)
- Translate** (selected)
- Rotate** (selected)
- Reflect** (selected)
- Project** (selected)
- Scale** (selected)
- ☐ Airbag Shell RefGM
- ☐ Airbag Node RefGM
- Rotate Model**
- ☐ Free Edit
- Rot. Axis:** X
- ☐ Pick origin
- NodeID:**
- XYZ:**
- Rot. Angle:**
- ☒ Global
- ☐ Local
- CID:**
- Coord Sys**
- Rotate-** **Rotate+**
- Options**
- ☐ Copy Elem
- ☐ Copy Node
- ☐ Copy Entity
- No. of copies:**
- ☐ Pick Part
- Plist**
- Starting EID:** 160040
- Starting NID:** 909350
- ☐ Transfer
- Reject** **Accept** **Done**

Element Tools → Transform (Reflect)

Purpose: reflect nodes/elements/parts

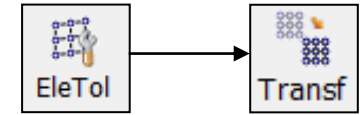
- Norm X/Y/Z – reflect along global axes
- N1-N2 – reflect along the direction from N1 to N2
- N1-N2-N3 – reflect along the normal formed by N1, N2, and N3
- Origin of reflection can be a node or an XYZ location
- Elements/Nodes can be copied while reflecting



Element Tools → Transform (Project)

Purpose: project nodes and elements

- Norm X/Y/Z – project along global axes
- N1-N2 – project along the direction from N1 to N2
- N1-N2-N3 – project along the normal formed by N1, N2, and N3
- Can also project To Mesh, To Curve, or To Surface
- Point on plane of projection can be a node or an XYZ location
- Elements can be copied while projecting



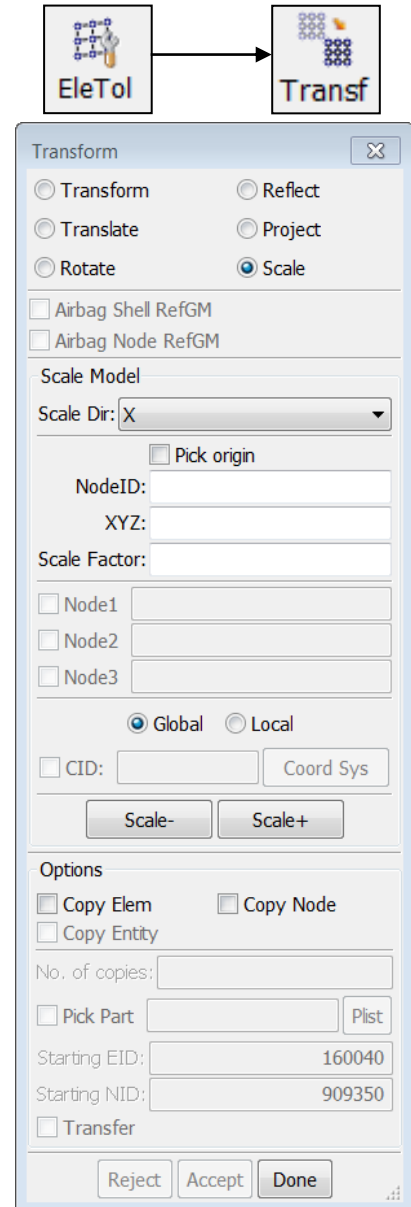
The screenshot shows the 'Transform' dialog box with the following settings:

- Transform** (selected), **Reflect**, **Translate**, **Rotate**, **Project** (selected), **Scale**
- ☐ Airbag Shell RefGM
- ☐ Airbag Node RefGM
- Project Model**
- Plane:** Norm X
- ☐ Pick location
- NodeID:**
- XYZ:**
- ☐ Node1
- ☐ Node2
- ☐ Node3
- ☒ **Along** **Vector**
- X** **Y** **Z**
- 1** **0** **0**
- Project**
- Options**
- ☐ Copy Elem ☐ Copy Node
- ☐ Copy Entity
- No. of copies:**
- ☐ Pick Part **Plist**
- Starting EID:** 160040
- Starting NID:** 909350
- ☐ Transfer
- Reject** **Accept** **Done**

Element Tools → Transform (Scale)

Purpose: scale nodes/elements/parts

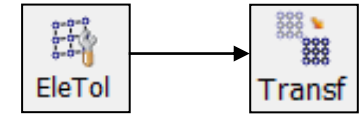
- Scale Factor – greater than 1 to enlarge, less than 1 to shrink (when using Scale+)
- X,Y,Z – scale in either global or local coordinate system
- N1-N2 – scale in direction from N1 to N2
- N1-N2-N3 – scale in direction of normal formed by N1, N2, and N3
- Origin can be a node or a XYZ location
- Elements/Nodes can be copied while scaling



Element Tools → Transform (Transform)

Purpose: transform nodes/elements/parts from one coordinate system to another

- From – pick 3 nodes (first set of P1,P2,P3) to define a starting system
- To – pick 3 nodes (second set of P1,P2,P3) to define a destination system
- P1-P3 buttons launch Create Position interface
- Elements can be copied while transforming



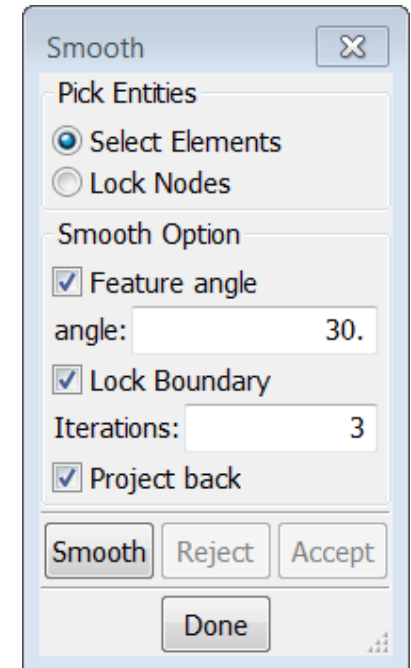
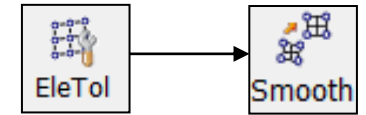
The screenshot shows the 'Transform' dialog box with the following options and settings:

- Transform** (selected), **Reflect**, **Translate**, **Project**, **Rotate**, **Scale**
- ☐ Airbag Shell RefGM
- ☐ Airbag Node RefGM
- Transform Model**
 - ☒ From
 - ☒ P1
 - ☐ P2
 - ☐ P3
 - ☐ To
 - ☐ P1
 - ☐ P2
 - ☐ P3
- Reset** button
- Transform** button
- Options**
 - ☐ Copy Elem ☐ Copy Node
 - ☐ Copy Entity
 - No. of copies:
 - ☐ Pick Part **Plist**
 - Starting EID: 160040
 - Starting NID: 909350
 - ☐ Transfer
- Reject** **Accept** **Done** buttons

Element Tools → Smooth

Purpose: smooth a mesh to improve element quality

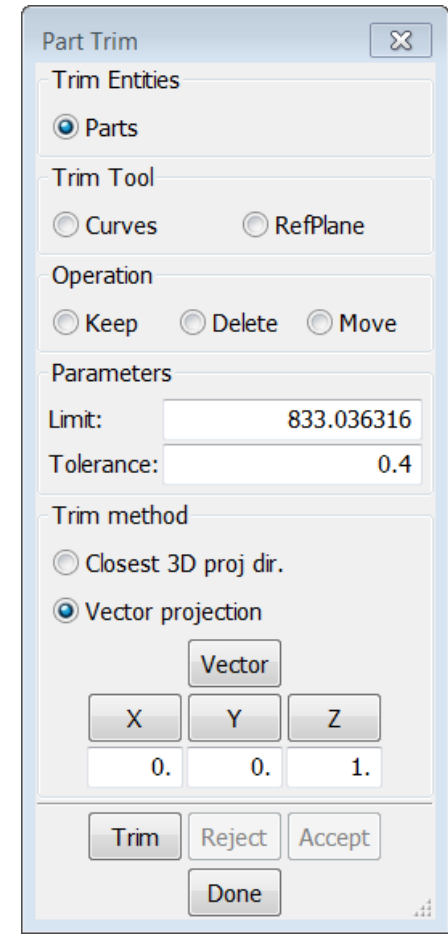
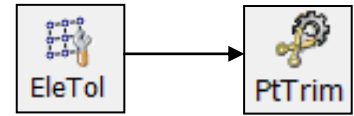
- Use General Selection interface to select elements
- Specified nodes can be locked to prevent movement
- Nodes along feature angles are not moved
- Boundary nodes can be fixed or free
- User specified number of smoothing iterations can be applied
- Nodes can be projected back to geometry after smoothing



Element Tools → Part Trim

Purpose: trim a mesh using a curve

- Parts – first, choose the parts to be trimmed
- Trim Curves – second, select curves for trimming
- Trim Methods (projecting curves to mesh)...
 - Closest 3D projection
 - Vector projection (along a specified vector)



Workshop 4

Build a fan model

- ❖ Create a surface
- ❖ Surface mesh
- ❖ 2Line mesh
- ❖ Part trim
- ❖ Model transform (rotate)
- ❖ Save a keyword file

Pre-Processing ***(continued...)***

Keyword Input Form

- Example Keyword Input form for *PART

Keyword type: *PART_(TITLE) (0)

List of existing keywords of this type:

Keyword Input Form

NewID Draw

☐ Use *Parameter

(Subsys: 1 New_Subsystem_1) Setting

RefBy Pick Add Accept Delete Default Done

1 TITLE

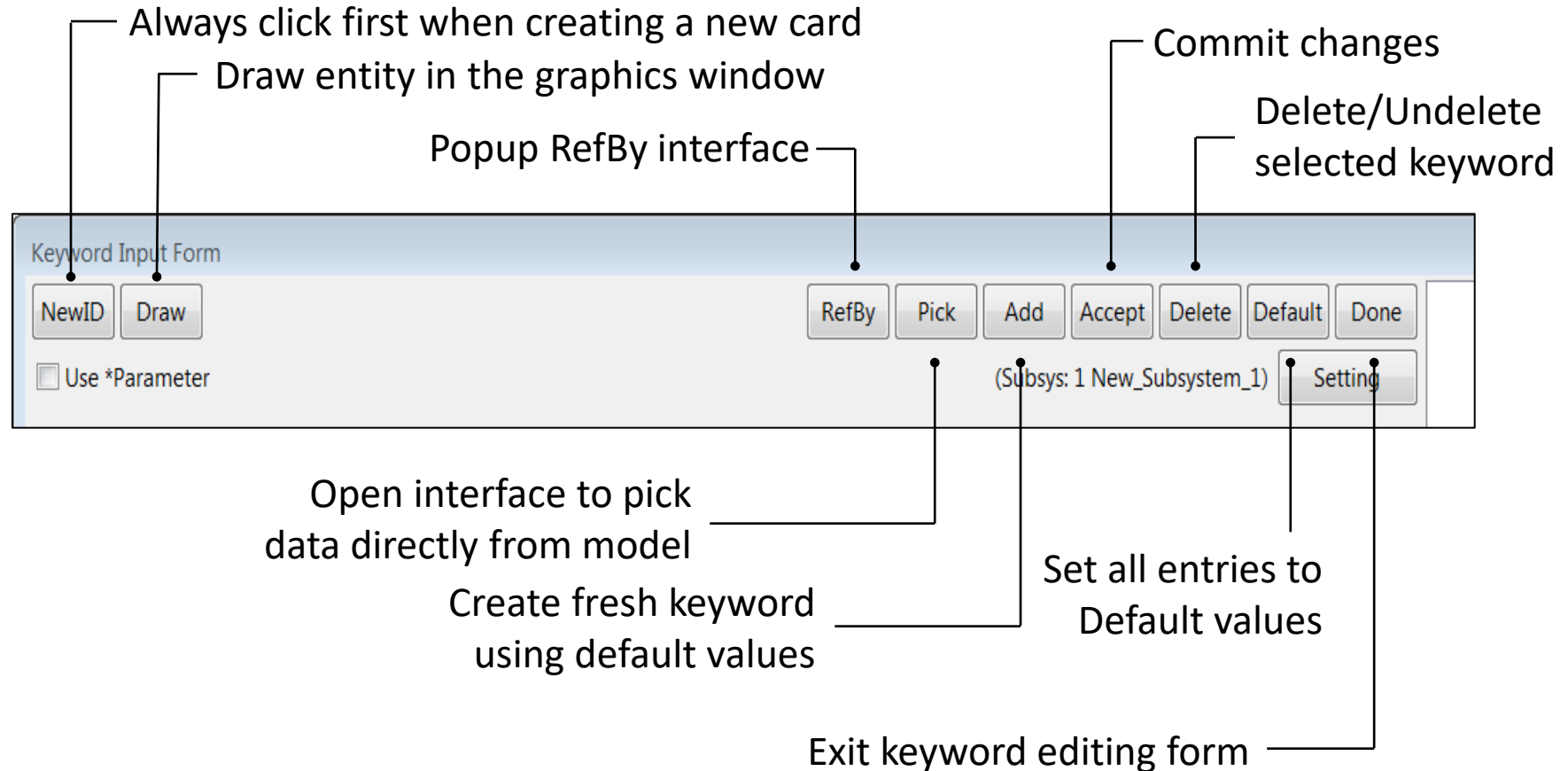
2 PID SECID MID EOSID HGID GRAV ADPOPT TMID

COMMENT:

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

Keyword Input Form *(continued...)*

- Keyword Input form buttons



Keyword Input Form *(continued...)*

- Keyword Input form controls

The image shows a screenshot of a software interface for entering keyword data. The interface is divided into two main sections. The top section is a header bar with a light gray background, containing the text '*PART_(TITLE) (0)'. Below this is a form area with a white background. The form contains two rows of input fields. The first row is labeled '1' and contains a single input field for 'TITLE'. The second row is labeled '2' and contains eight input fields: 'PID', 'SECID', 'MID', 'EOSID', 'HGID', 'GRAV', 'ADPOPT', and 'TMID'. Each of these fields has a small square icon next to it. The 'GRAV' field is a drop-down menu showing the value '0'. The 'EOSID', 'HGID', and 'TMID' fields show the value '0'. To the right of the input fields is a vertical scrollbar. Labels with leader lines point to various components: 'Card Number' points to the '1' label; 'Parameter Names' points to the 'TITLE' label; 'Link Button' points to the 'ADPOPT' field; 'Parameter Values' points to the 'PID' field; and 'Drop-down Menu' points to the 'GRAV' field.

Card Number

Parameter Names

Link Button

Parameter Values

Drop-down Menu

*PART_(TITLE) (0)

1 TITLE

2 PID SECID MID EOSID HGID GRAV ADPOPT TMID

0 0 0

Keyword Input Form *(continued...)*

- Other Keyword Input form features
 - **Blue** parameter titles can be clicked to display description in bottom text area (see below)
 - **Red** parameters indicate that additional cards may be displayed depending on the parameter value

User comments can be entered here

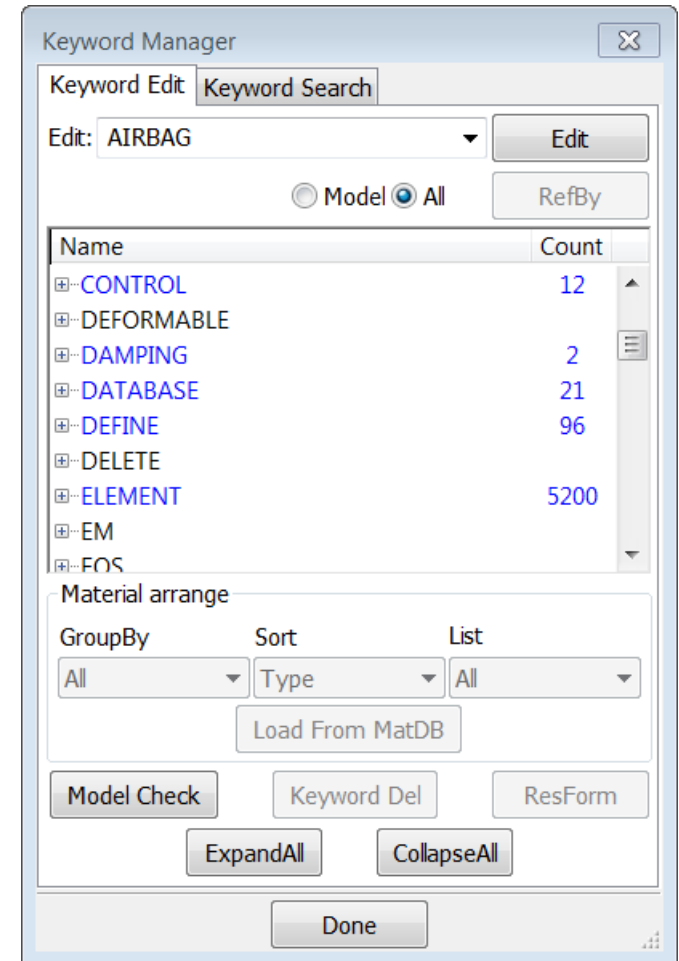
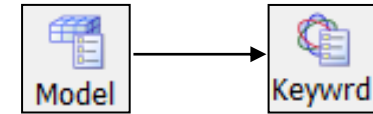
Click in any field above to display description here
(same information found in LS-DYNA Keyword Manual)

COMMENT:

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

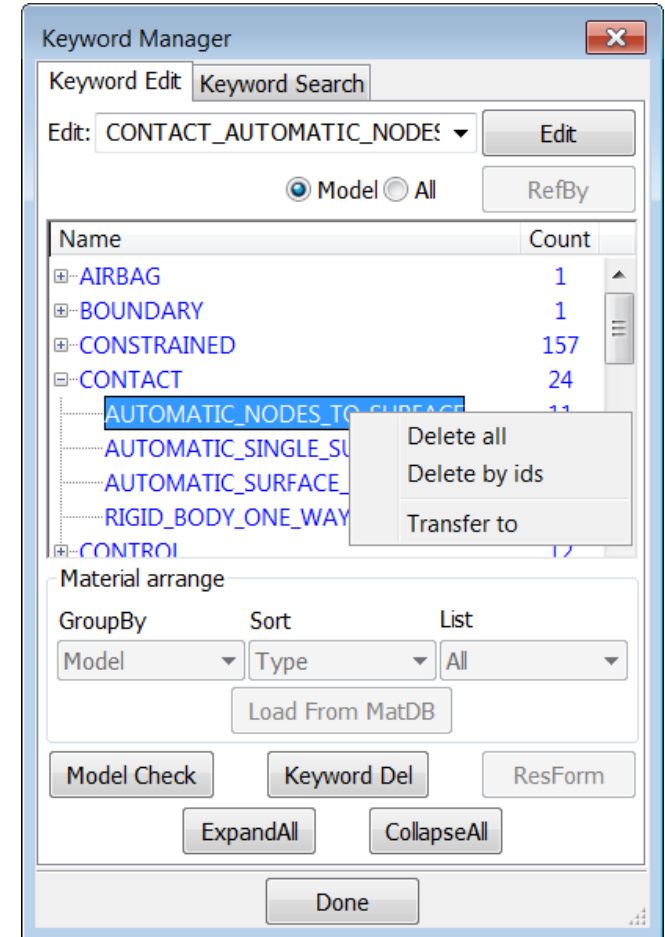
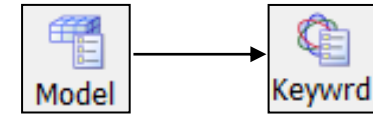
Model → Keyword Manager

- All LS-DYNA keywords are accessible through Keyword Manager tree list (over 900 total)
- Keywords that exist in the current model are shown in **Blue**
- Click **All** to display all keywords (or click **Model** to display only those that exist in the current model)
- Expand the tree and double-click to edit a keyword (opens Keyword Input Form)



Model → Keyword Delete and Transfer

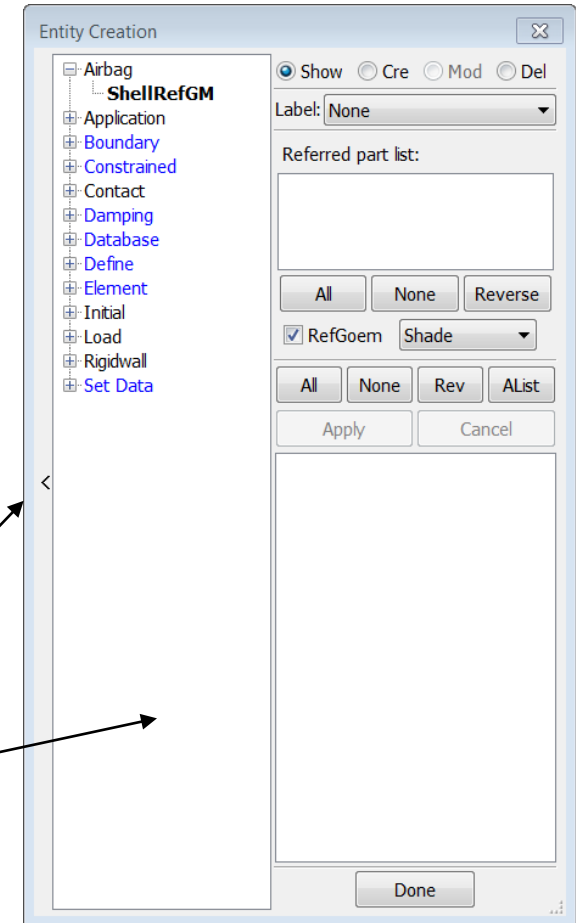
- Right click on a keyword in the keyword tree opens up the Deletion and Transfer menu
- Keyword data can be deleted by all or by IDs
- A keyword data can be transferred to another similar keyword data, common fields will be preserved, missing fields will need to be entered. e.g.
 - *ELEMENT_SHELL_BETA to
 - *ELEMENT_SHELL_THICKNESS



Model → Entity Interface

A uniform interface to deal with LS-DYNA entities that can be shown graphically

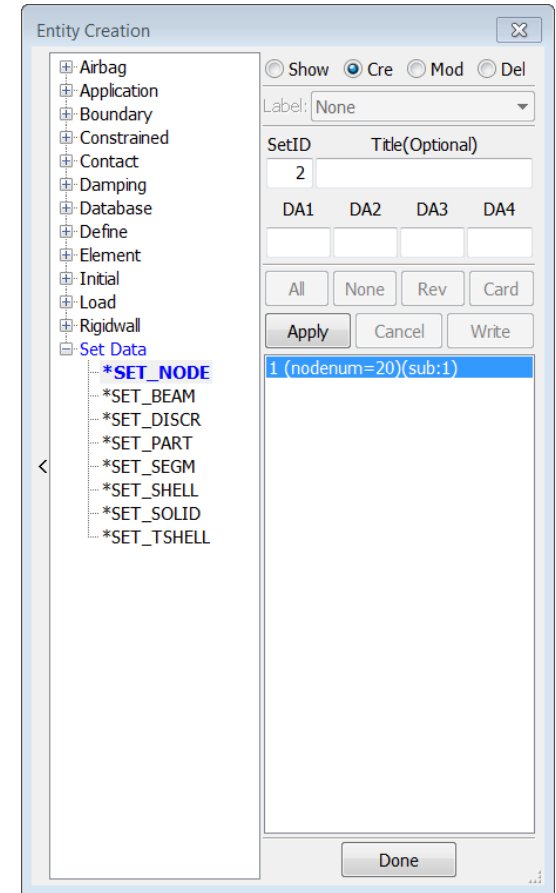
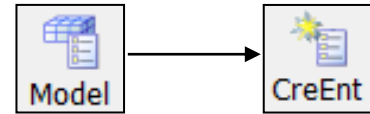
- Show – to show the entities
- Create – to create the entity
- Modify – modify the existing entities
- Delete – delete entities
- Write entity data to a file
- General selection is used whenever it is possible for selecting the entities



Arrow to collapse this area

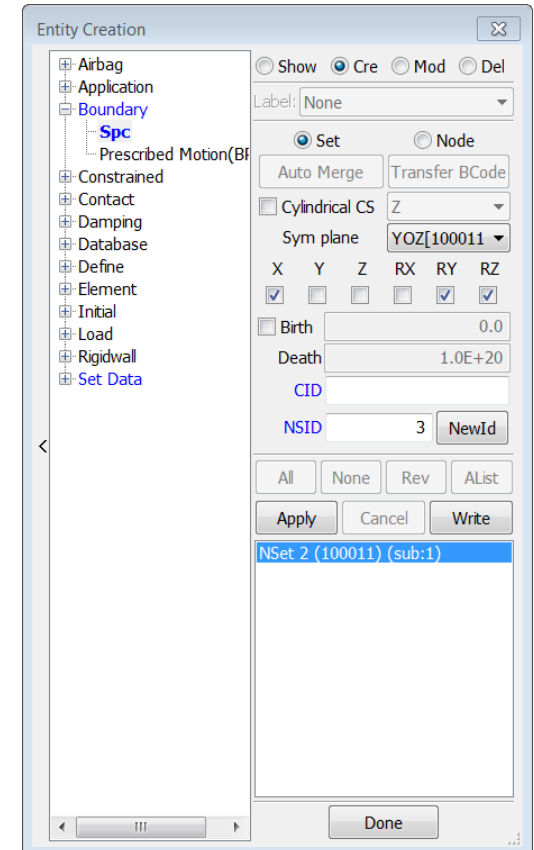
Model → Create Entity → Set Data

- Show *SET_{OPTION}
- Create *SET_{OPTION}
 - *OPTIONS* – NODE, BEAM, DISCRETE, PART, SEGMENT, SHELL, SOLID, THICK SHELL
 - Select entities using General Selection interface
- Modify *SET_{OPTION}
 - Sets must be shown before they can be modified
- Delete *SET_{OPTION}
- Write *SET_{OPTION} to a file



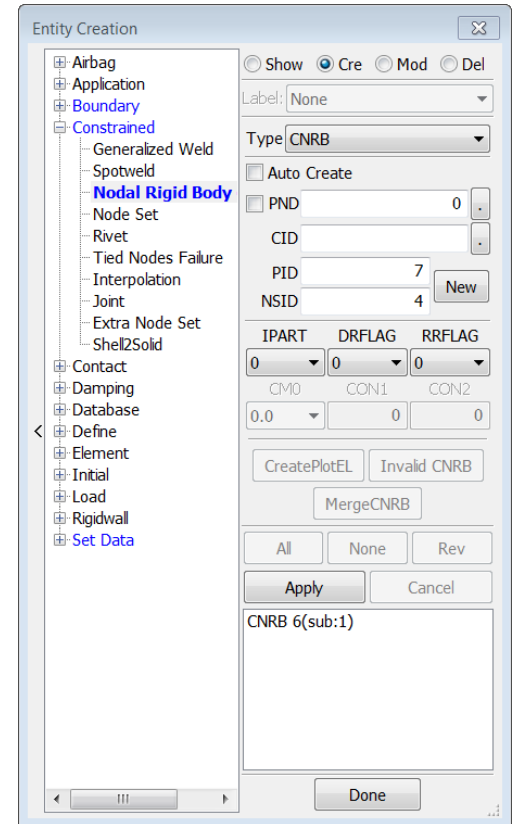
Boundary → Spc

- Show *BOUNDARY_SPC_{*OPTION*}
 - Label – None/Symbol/Detail
 - Local coordinate systems will be shown if used
 - Match – Filter by constrained DOFs
- Create *BOUNDARY_SPC_{*OPTION*}
 - *OPTIONS* – NODE, NODE_SET
 - Use General Selection interface to select nodes
 - Activate constrained DOFs
 - Use global or local coordinate system
- Modify *BOUNDARY_SPC_{*OPTION*}
- Delete *BOUNDARY_SPC_{*OPTION*}



Constrained → Nodal Rigid Body (CNRB)

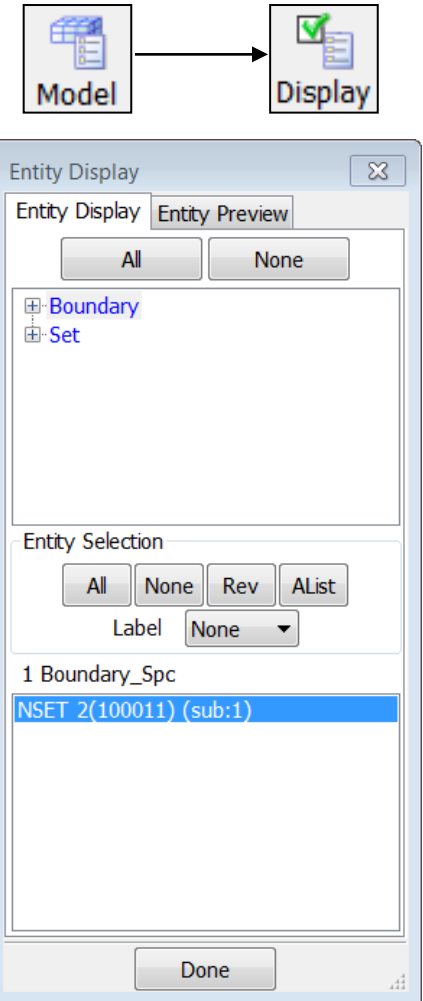
- Show *CONSTRAINED_NODAL_RIGID_BODY
- Create *CONSTRAINED_NODAL_RIGID_BODY
 - *OPTION* – SPC
 - Use General Selection interface to select nodes
 - Set additional flags
- Modify
*CONSTRAINED_NODAL_RIGID_BODY
- Delete *CONSTRAINED_NODAL_RIGID_BODY



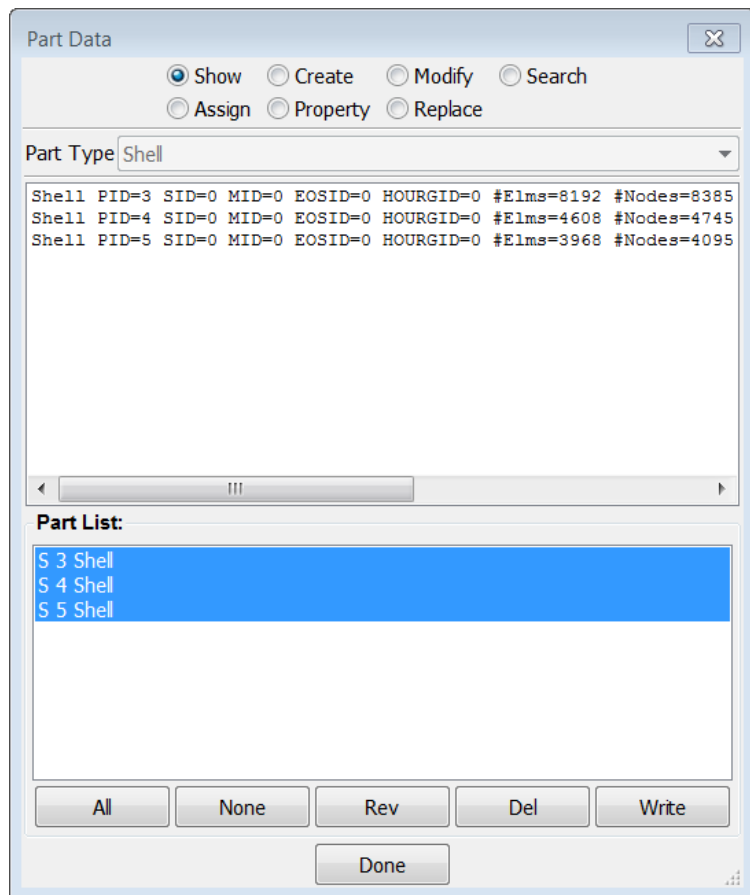
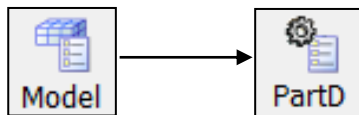
Model → Display Entity

Purpose: visualize model entities (keywords) other than nodes and elements (which are displayed by default)

- Available options include...
 - *BOUNDARY_{OPTION}
 - *CONSTRAINED_{OPTION}
 - *INITIAL_VELOCITY_{OPTION}
 - *LOAD_{OPTION}
 - *RIGIDWALL_{OPTION}
 - *SET_{OPTION}
- Use while post-processing by loading d3plots followed by the corresponding keyword file



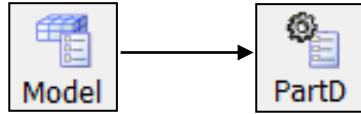
Model → Part Data



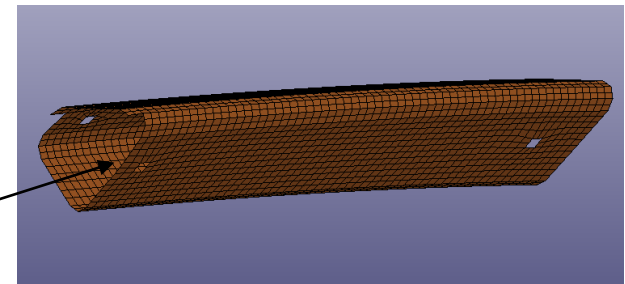
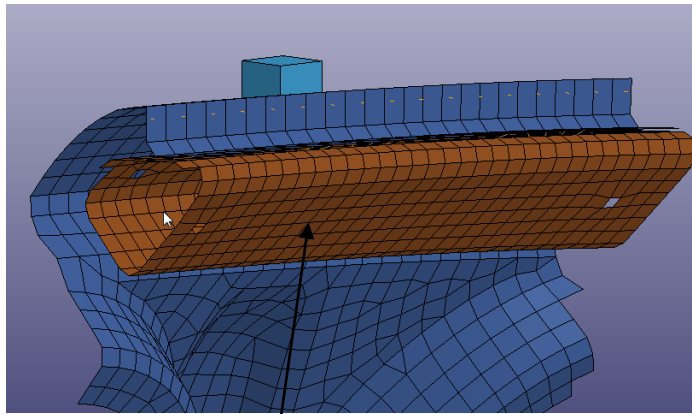
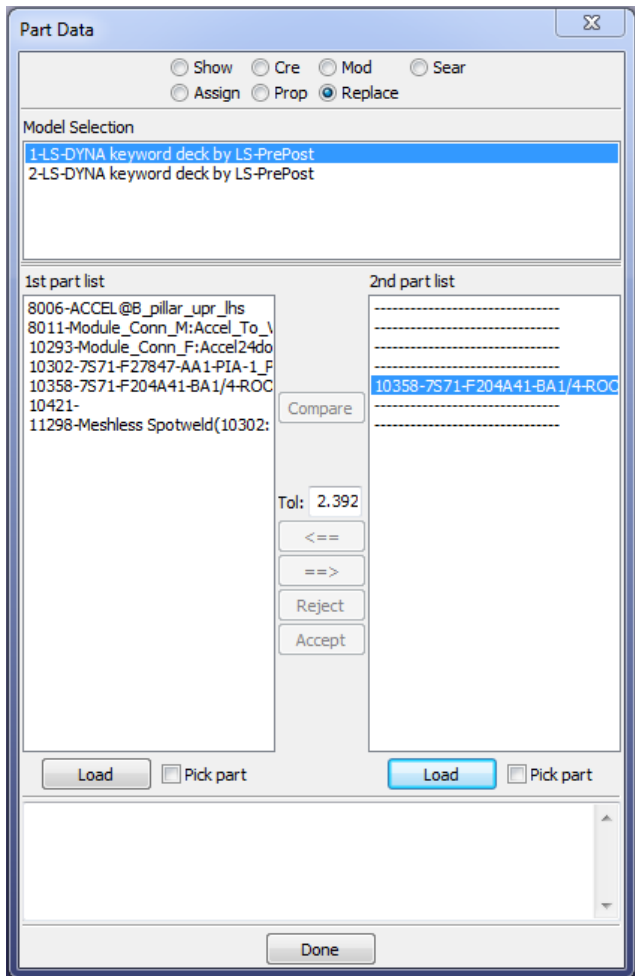
There are 7 functions in the Part Data dialog:

- Show – show existing part data
- Create – create new part data
- Mod – modify existing data
- Sear – search parts by parameters
- Assign – assign part data properties
- Prop – edit properties of part data
- Replace – replace a part with another part

Model → Part Data



- Replace – replace a part in one model with another part from another model
- The second model can be loaded with the “Load” button

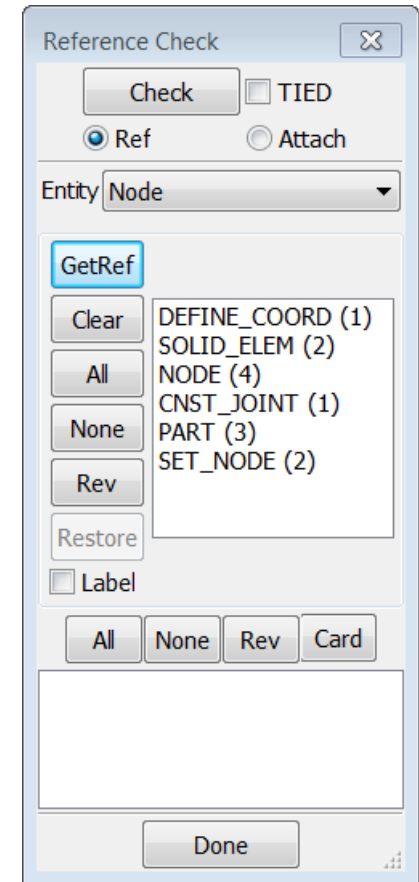


Replace this part with
another part that has
finer mesh

Model → Reference Check (Attach)

Purpose: identify entities that are attached to other model entities

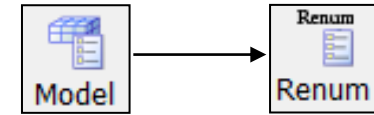
- Initial attachment set can be all visible or as selected (using General Selection interface)
- Attachment set can grow By Part or By Element
- Final attachment set can be written to a file



Model → Renum

Purpose: renumber and offset model entities

- By Keyword – Entities can be renumbered/offset by Keyword
 - ID range can be specified
 - Entities can be picked using the General Selection interface
- By Part – Parts/Elements/Nodes can be renumbered/offset by Part
- By Selected – Parts/Elements/Nodes can be renumbered/offset by Selected

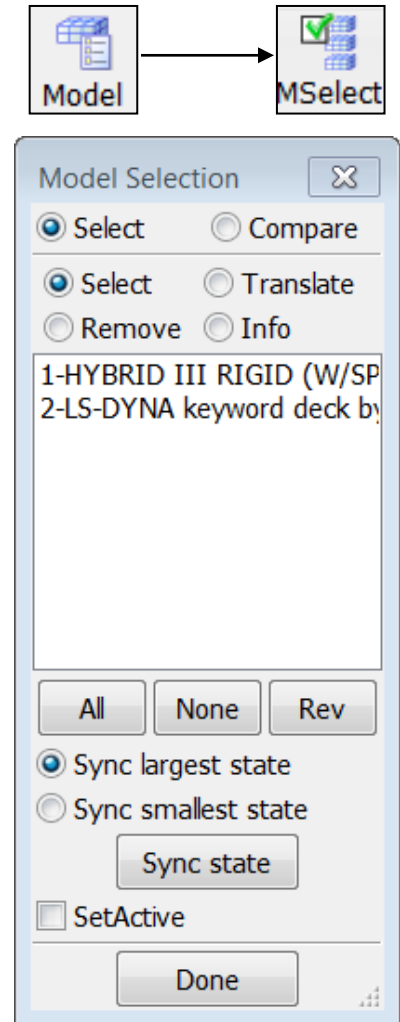


Name	Count	StartID	From	To
✓ NODE	7787	1	1	909349
✓ ELEMENT_BEAM	1	1	20	20
✓ ELEMENT_SHELL	3114	2	295	160039
✓ ELEMENT_SOLID	1836	3116	1	6809
✓ ELEMENT_DISCRE	18	4952	21	90941
✓ ELEMENT_SEATBI	220	4970	90942	91161

Model → MSelect → Select

Purpose: switch between loaded models, view multiple models, perform keyword comparison

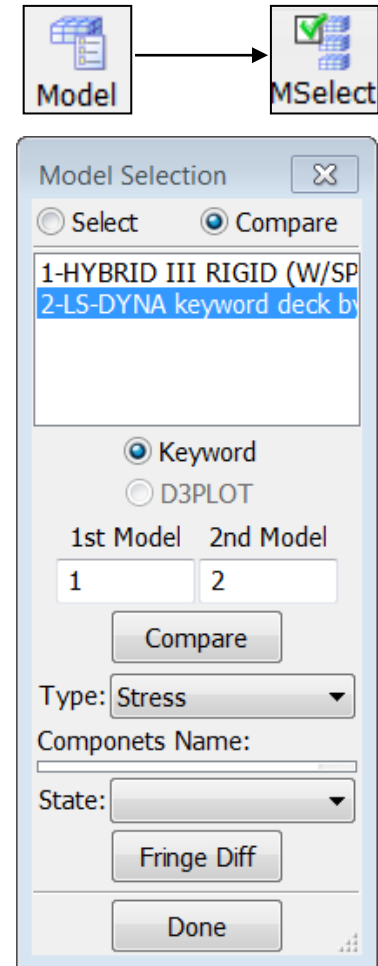
- Select – display selected model(s)
- Trans – translate a model (for visualization only, useful for side-by-side animation)
- Remove – unload a model
- Info – show model summary



Model → MSelect → Compare

Purpose: compare two similar LS-DYNA keyword input files that have minor differences

- Disregards keyword order, unlike other compare/difference software
- Normalizes data to ignore differences below a certain tolerance
- Provides a summary only when a large number of differences are detected

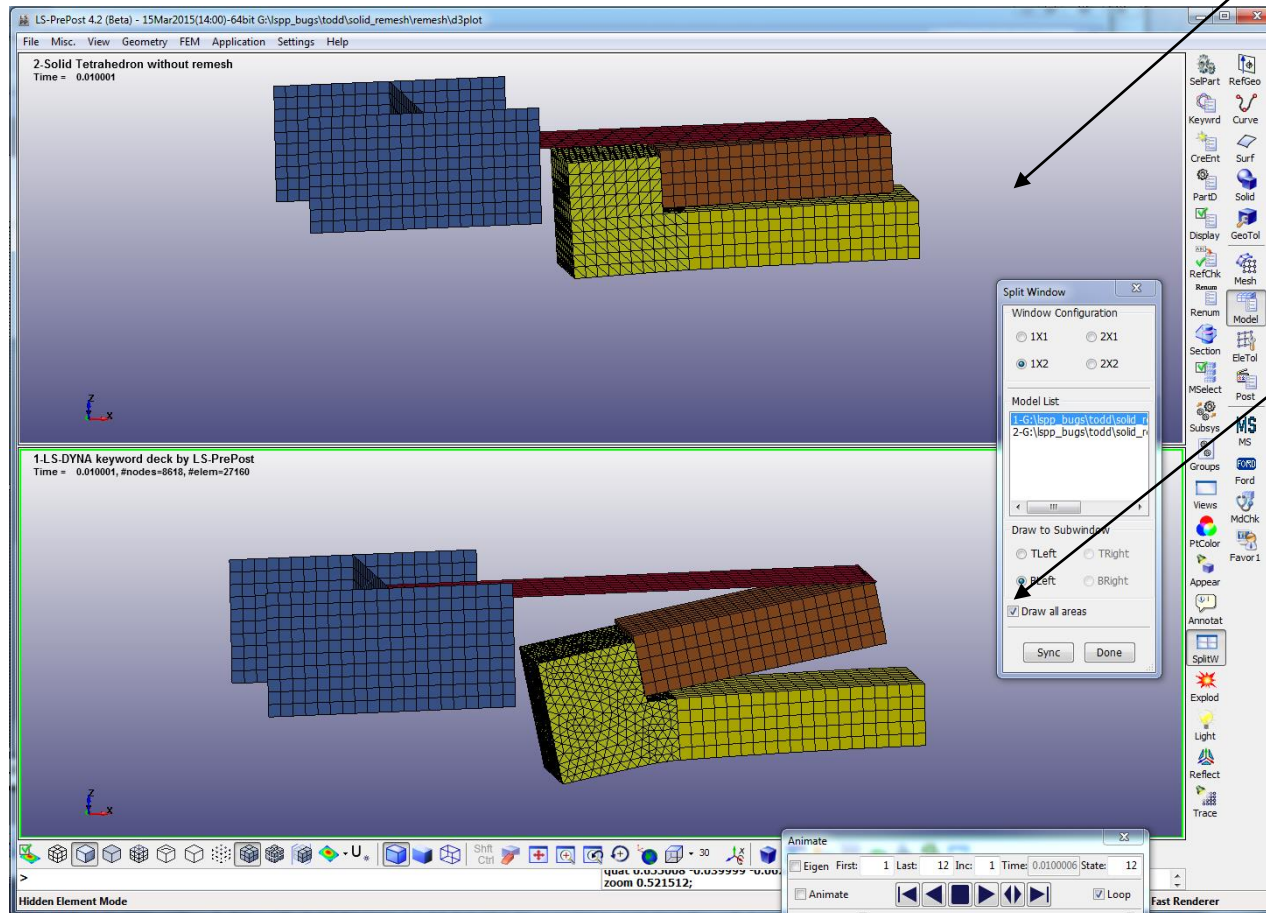


Model → SplitW (Split Windows)

- Allow multiple windows to display multiple models or same model with different views
- Configure with 2x1, 1x2, or 2x2 splits

Right click windows to activate windows to be focus and set as main window

Turn on Draw All Area to draw all windows, also true with most operations like AC, blank a part, blank element, fringe, etc.



Application → Model Checking

Element Quality Check: provides general idea of the quality of the finite element model

Define allowable value, LS-PrePost will show no. of elements that violate the allowable. The violated elements can be saved into general selection buffers

Different checking method for different element types

Model Checking

Element Quality | Keyword Check | Contact Check | Model Check Setting

☐ Beam ☐ Solid
☒ Shell ☐ Tshell

Checking method

Quality Check
Duplicate
Normal
Free - Unattached
UnderCut

Shell check item	Allowable	Min. val	Max. val	#violated(%)
<input type="checkbox"/> Min side len	3.0	***	***	***
<input type="checkbox"/> Max side len	30.0	***	***	***
<input type="checkbox"/> Aspect ratio	10	***	***	***
<input type="checkbox"/> Warp	10	***	***	***
<input type="checkbox"/> Min quad ang	45	***	***	***
<input type="checkbox"/> Max quad ang	135	***	***	***
<input type="checkbox"/> Min tria ang	30	***	***	***
<input type="checkbox"/> Max tria ang	120	***	***	***
<input type="checkbox"/> Taper	0.7	***	***	***
<input type="checkbox"/> Skew	45	***	***	***

☒ Frin

Clear Save Failed Delete Failed Report

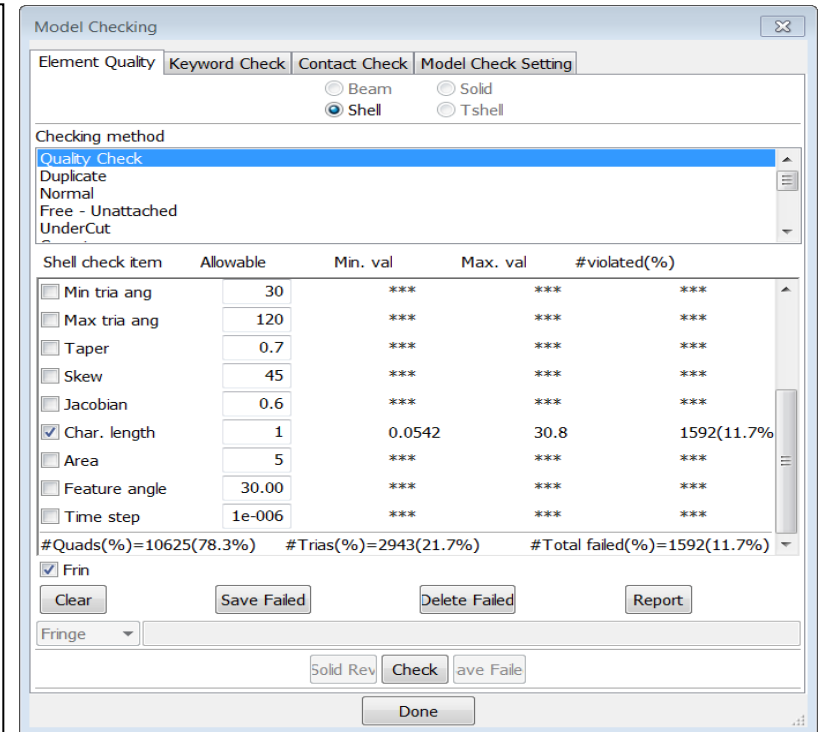
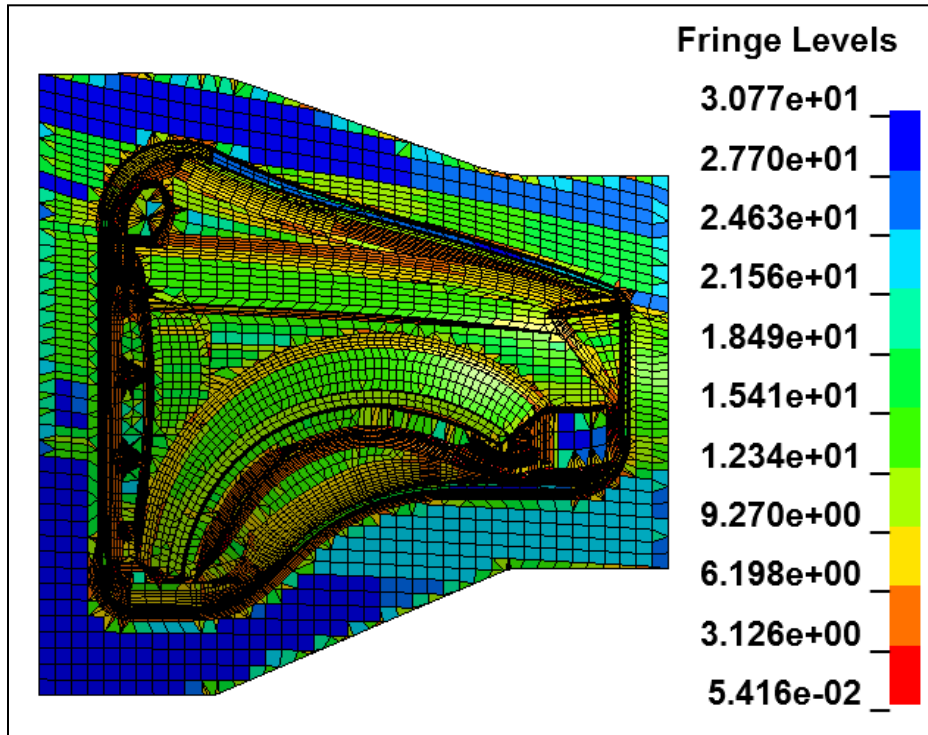
Fringe

Solid Rev Check Save Failed

Done

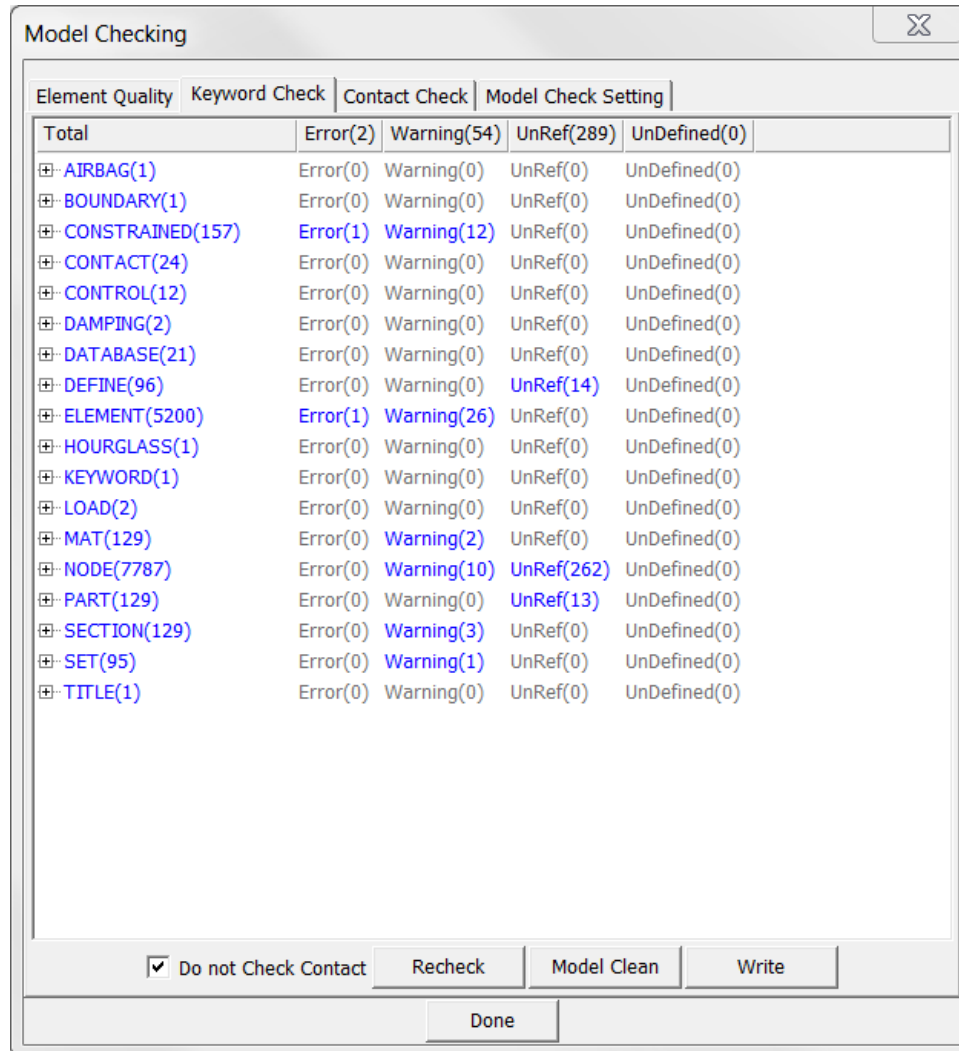
Application → Model Checking

Element Quality Check example: characteristic length



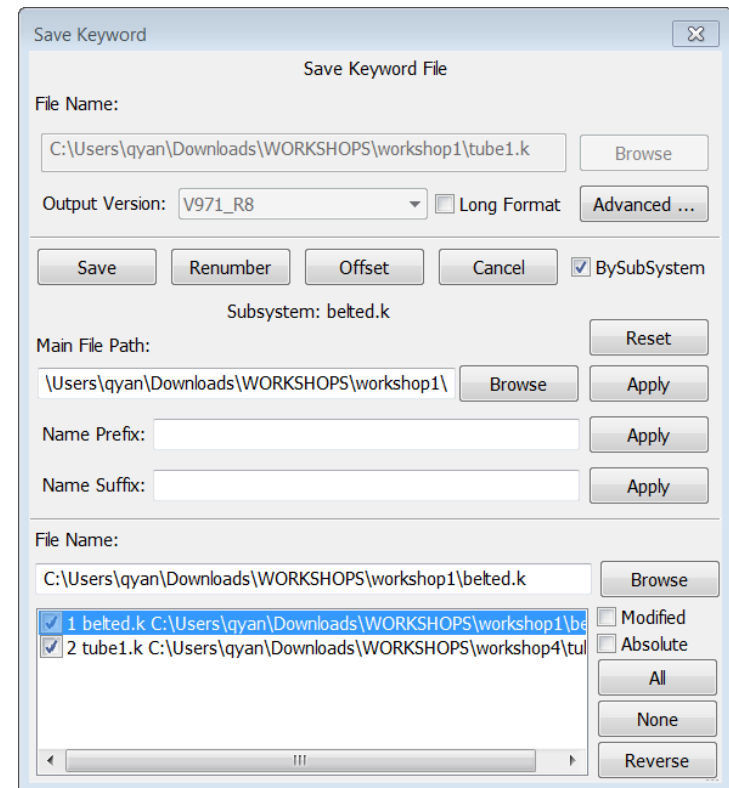
Application → Model Checking

Comprehensive LS-DYNA keyword data check, to indicate if the keyword data will or will not fail in LS-DYNA run



File → Save Keyword As...

- Output Version – 960/970/971 (only change if using an old version of LS-DYNA)
- Renumber/Offset – can renumber or offset before saving
- BySubSystem – activate to save subsystems to separate files (not visible unless model contains subsystems)
- Advanced... – additional options (see next slide)



Advanced Settings Interface

- Keyword order can be customized
- Keywords can be selectively omitted
- Title, Field Names, and Comments can be omitted
- Parameter names can be output instead of actual values

Keyword Output Order Setting

Input
☐ Input
☐ Alphabetical
☐ Expert
☒ Customer ☒ GroupOut

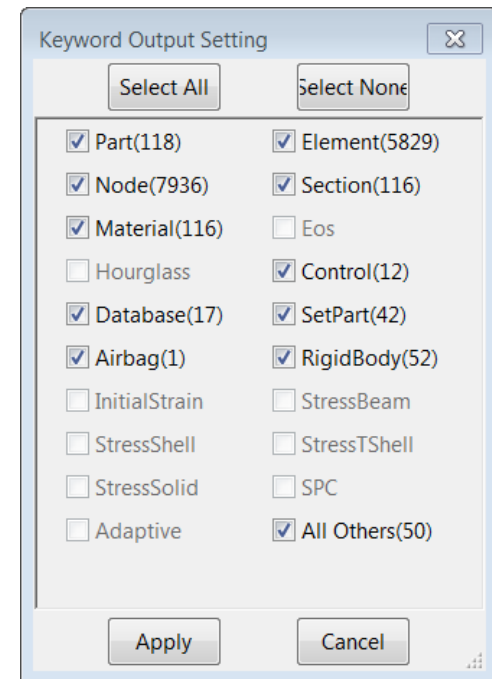
Advanced Setting Save Keyword File
☐ Freeformat style ☒ Field names ☒ Keyword title ☒ Parameter names ☒ Comments
☐ Non-Default only ☐ Refcheck ☐ Blank line ☐ Undefined Check ☐ Save Component
☐ Path Separator "\\"

Keyword	Out_order	Fname	Keyword	Out_order	Fname	Keyword	Out_order	Fname
<input checked="" type="checkbox"/> AIRBAG(1)	12	<input checked="" type="checkbox"/>	<input type="checkbox"/> DELETE		<input type="checkbox"/>	<input checked="" type="checkbox"/> NODE(8459)	17	<input checked="" type="checkbox"/>
<input type="checkbox"/> ALE		<input type="checkbox"/>	<input checked="" type="checkbox"/> ELEMENT(5840)	16	<input checked="" type="checkbox"/>	<input type="checkbox"/> PARAMETER		<input type="checkbox"/>
<input checked="" type="checkbox"/> BOUNDARY(1)	4	<input checked="" type="checkbox"/>	<input type="checkbox"/> EM		<input type="checkbox"/>	<input checked="" type="checkbox"/> PART(131)	7	<input checked="" type="checkbox"/>
<input type="checkbox"/> CASE		<input type="checkbox"/>	<input type="checkbox"/> EOS		<input type="checkbox"/>	<input type="checkbox"/> PARTICLE		<input type="checkbox"/>
<input type="checkbox"/> CESE		<input type="checkbox"/>	<input type="checkbox"/> FREQUENCY		<input type="checkbox"/>	<input type="checkbox"/> PERTURBATION		<input type="checkbox"/>
<input type="checkbox"/> CHANGE		<input type="checkbox"/>	<input checked="" type="checkbox"/> HOURLASS(1)	9	<input checked="" type="checkbox"/>	<input type="checkbox"/> RAIL		<input type="checkbox"/>
<input type="checkbox"/> CHEMISTRY		<input type="checkbox"/>	<input type="checkbox"/> ICFD		<input type="checkbox"/>	<input type="checkbox"/> RIGIDWALL		<input type="checkbox"/>
<input type="checkbox"/> COMMENT		<input type="checkbox"/>	<input type="checkbox"/> INCLUDE		<input type="checkbox"/>	<input checked="" type="checkbox"/> SECTION(129)	8	<input checked="" type="checkbox"/>
<input type="checkbox"/> COMPONENT		<input type="checkbox"/>	<input type="checkbox"/> INITIAL		<input type="checkbox"/>	<input type="checkbox"/> SENSOR		<input type="checkbox"/>
<input checked="" type="checkbox"/> CONSTRAINED(157)	14	<input checked="" type="checkbox"/>	<input type="checkbox"/> INTEGRATION		<input type="checkbox"/>	<input checked="" type="checkbox"/> SET(95)	13	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> CONTACT(24)	6	<input checked="" type="checkbox"/>	<input type="checkbox"/> INTERFACE		<input type="checkbox"/>	<input type="checkbox"/> STOCHASTIC		<input type="checkbox"/>
<input checked="" type="checkbox"/> CONTROL(12)	2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/> KEYWORD(2)	1	<input checked="" type="checkbox"/>	<input type="checkbox"/> STRESS		<input type="checkbox"/>
<input type="checkbox"/> DEFORMABLE		<input type="checkbox"/>	<input checked="" type="checkbox"/> LOAD(2)	5	<input checked="" type="checkbox"/>	<input type="checkbox"/> TERMINATION		<input type="checkbox"/>

Write Config Cancel Ok

File → Save Active Keyword As...

- Output Version – 960/970/971 (only change if using an old version of LS-DYNA)
- Keywords can be selectively output using the Advanced Setting interface



Workshop 5

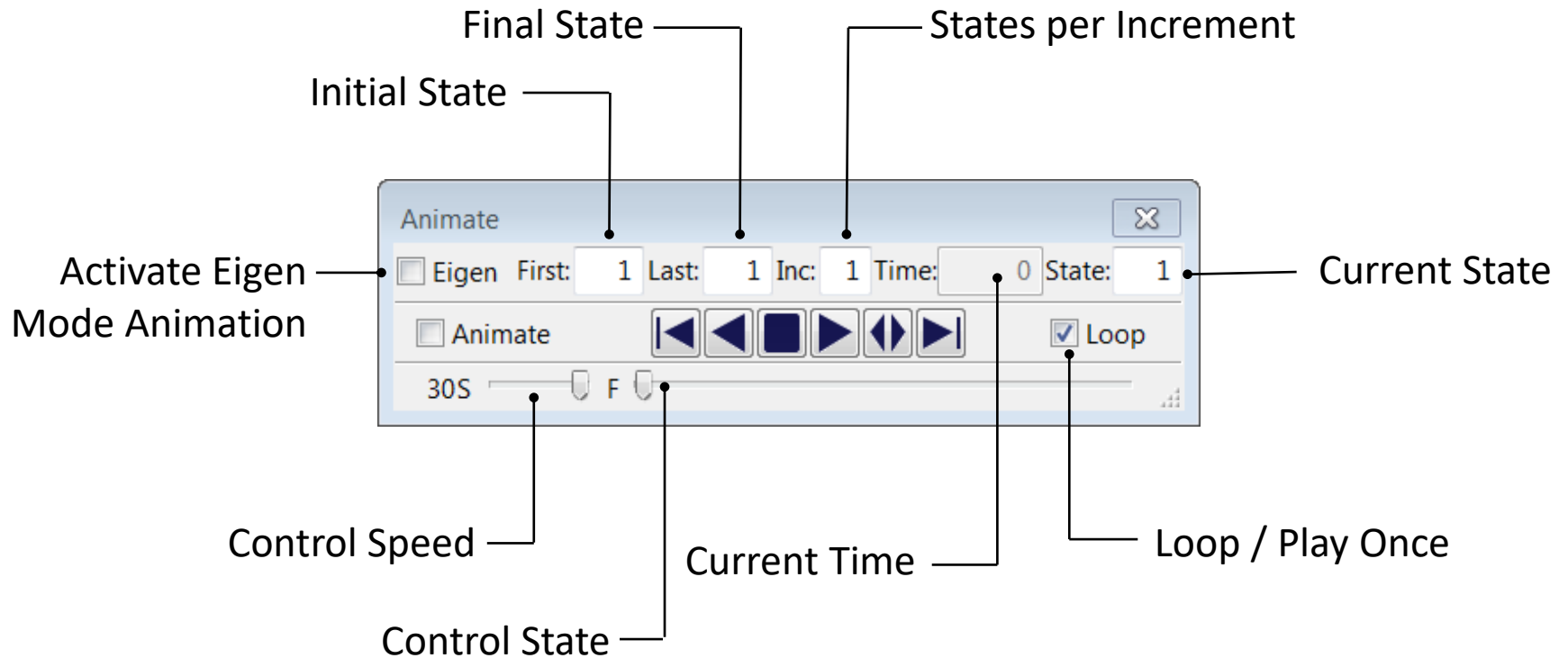
Create a Keyword File (S-Rail to Rigidwall Impact)

- ❖ Mesh model
- ❖ Assign material and property
- ❖ Apply mass, constraint and velocity
- ❖ Rigid wall creation
- ❖ Define spot welding
- ❖ Save a keyword file
- ❖ Model comparison

Post-Processing

Animation Interface

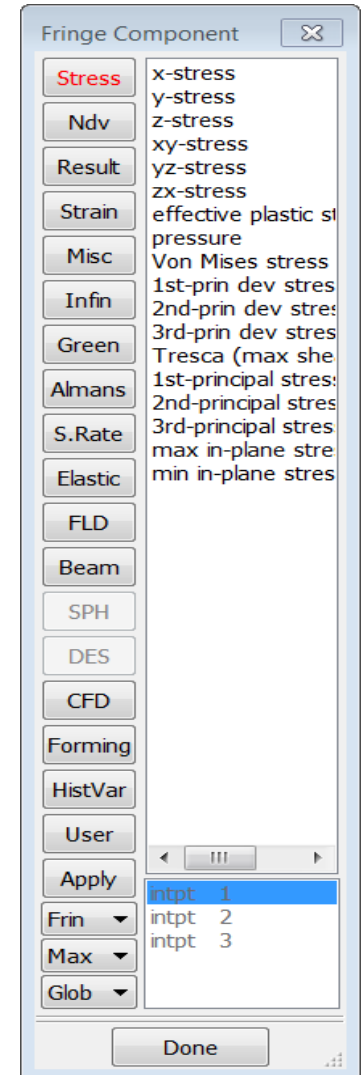
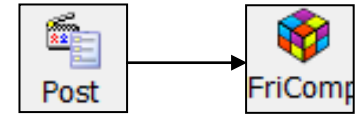
- The Animation controls are displayed when d3plots are loaded. If the animation interface is closed, it can be restored by clicking the **Anim** render button.



Post → Fringe Component

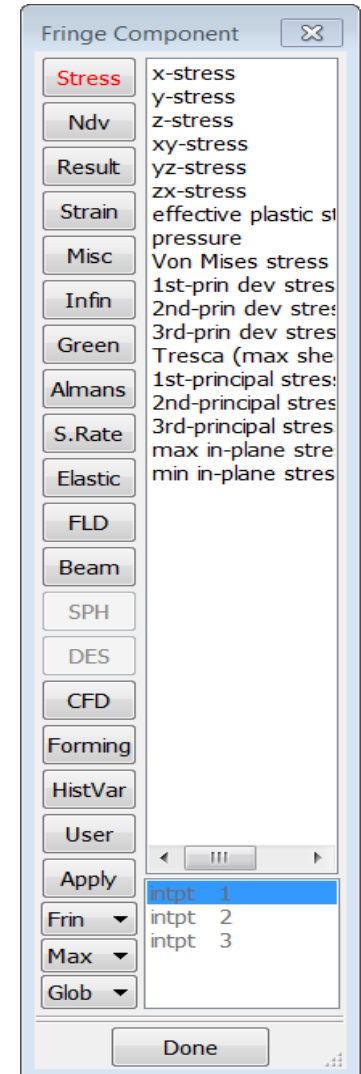
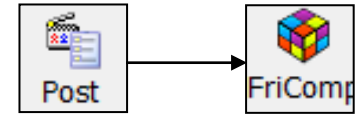
Purpose: to select a component for fringing

- Stress – regular stress components
- Ndv – nodal displacements and velocities
- Result – resultant stress components
- Strain – regular strain components
- Misc – pressure, temp, shell thickness, etc...
- Infin / Green / Almansi – infinitesimal, Green-St. Venant, and Almansi strains
- S.Rate – strain rates
- Residu – residual elastic strains
- FLD – forming limit diagram strain components
- Beam – beam element forces and stress resultants
- CFD – Navier-Stokes fluid components



Post → Fringe Component (*continued...*)

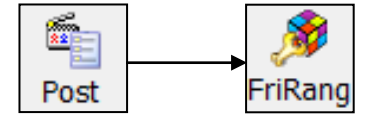
- Frin – choose rendering type
 - Frin – default fringing
 - Isos – iso-surfaces for solid
 - Lcon – line contours
 - XFrn – fringe max value through all states
 - FMes – fringe color on the mesh
 - Expr – user define expression
- Max – location of shell surfaces
 - Low / Mid / Upp – lower, mid-plane, and upper
 - Max / Ave / Min – maximum, average, minimum
 - Ipt – shell integration point
 - Bpt – beam integration point
- Glob – coordinate system
 - Glob / Loca – global or local



Post → Fringe Range

Purpose: control fringe and iso-surface ranges

- Dynamic – min/max adjusted for each time state
- Static – same min/max for all the states
- User – custom min/max for all the states
- Show – show elements within the specified range
- Entire Model – legend based on entire model
- Active Parts Only – legend based on displayed parts
- Active Elements Only – legend based on displayed elements only

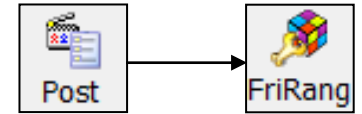


The 'Fringe Range' dialog box is shown with the following settings:

- Dynamic** (selected radio button)
- Static** (unselected radio button)
- User** (unselected radio button)
- Show** (unselected radio button)
- Entire Model** (unselected radio button)
- Active Parts Only** (selected radio button)
- Active Elements Only** (unselected radio button)
- Min:** (empty text box)
- Max:** (empty text box)
- Assign** (button)
- Report** (button)
- Avg:** **Nodal** (dropdown menu)
- Reverse Colors** (checkbox, unselected)
- Reverse Signs** (checkbox, unselected)
- Show Region Min/Max** (checkbox, unselected)
- Ident Min** (checkbox, unselected)
- Max** (checkbox, unselected)
- No. Min/Max** **5** (text box)
- Lower** **Gray90** (dropdown menu)
- Upper** **Gray90** (dropdown menu)
- Set Iso Range** (checkbox, unselected)
- Level** **10** (text box)
- 10** (text box)
- Palette** (button)
- Update** (button)
- Done** (button)

Post → Fringe Range *(continued...)*

- Reverse Colors – reverse legend colors
- Ident Min / Max – identify min and max elements by ID (user defines number of entities to identify)
- Lower / Upper – set color for out-of-range elements
- Levels – customize number of fringe levels
- Palette – allows colors to be modified

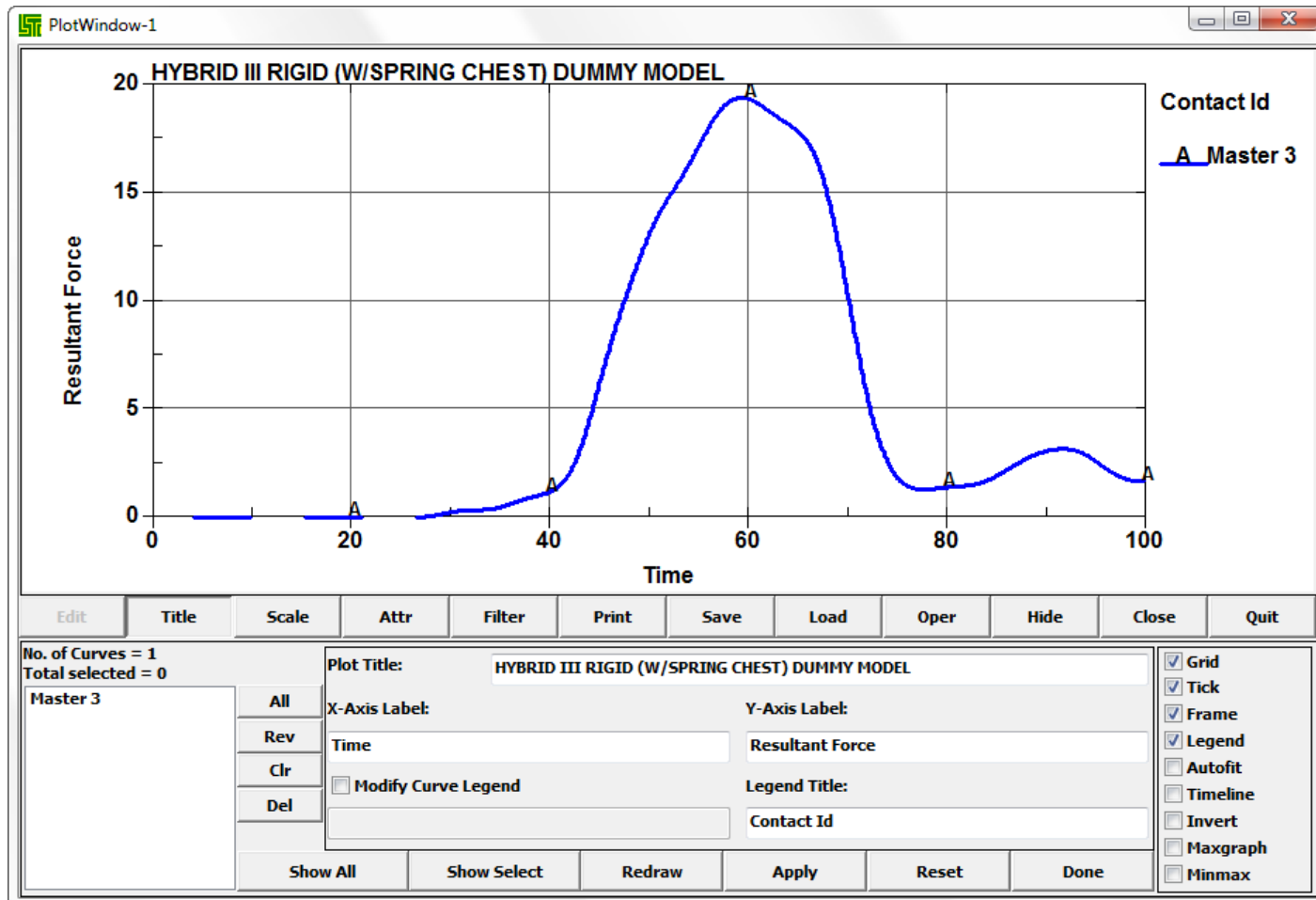


The screenshot shows the 'Fringe Range' dialog box. It has a title bar with a close button. The dialog contains several sections of controls:

- Radio Buttons:** ☐ Dynamic, ☐ Static, ☒ User, ☐ Show.
- Selection Radio Buttons:** ☐ Entire Model, ☒ Active Parts Only, ☐ Active Elements Only.
- Input Fields:** Min: 0, Max: 0.
- Buttons:** Assign, Report.
- Avg:** Nodal (dropdown).
- Checkboxes:** ☐ Reverse Colors, ☐ Reverse Signs, ☐ Show Region Min/Max, ☐ Ident Min, ☐ Max.
- No. Min/Max:** 5 (input field).
- Lower/Upper Color Selection:** ☐ Lower: Gray90 (dropdown), ☐ Upper: Gray90 (dropdown).
- Set Iso Range:** ☐ Set Iso Range.
- Level:** 10 (dropdown), 10 (input field).
- Buttons:** Palette, Update.
- Done:** Done (button).

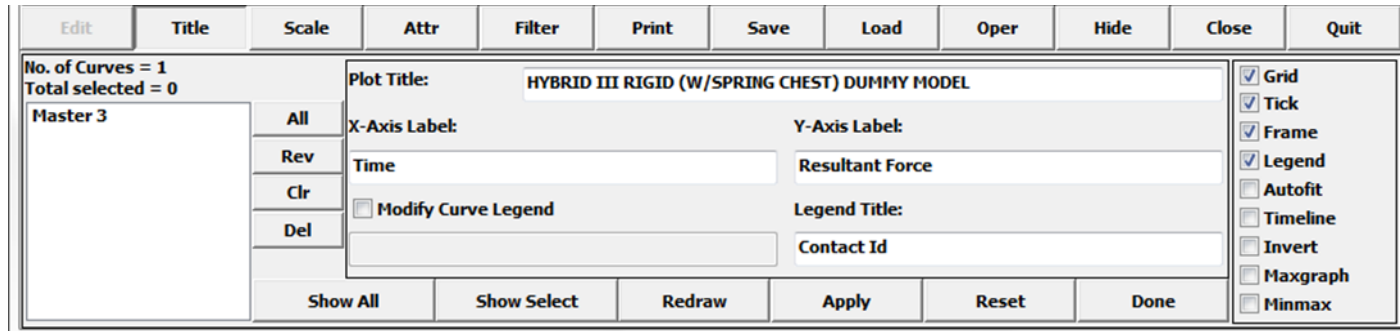
Plot Window

- Drawing XY curves for History/ASCII/Binout/XYPlot



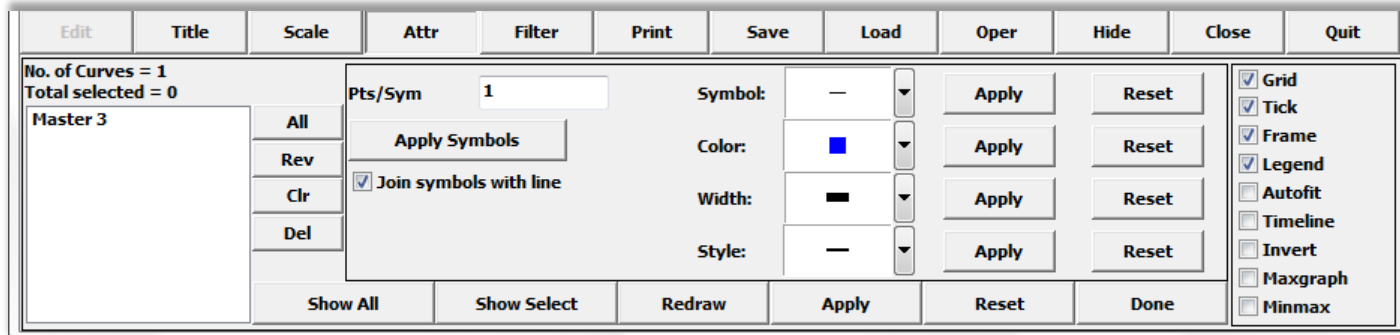
Plot Window (continued...)

- Title – modify main, axis, legend, and curve titles



The screenshot shows the 'Title' tab of the Plot Window. The top menu bar includes 'Edit', 'Title', 'Scale', 'Attr', 'Filter', 'Print', 'Save', 'Load', 'Oper', 'Hide', 'Close', and 'Quit'. The main area is divided into several sections. On the left, a list of curves shows 'Master 3' selected. In the center, the 'Plot Title' is 'HYBRID III RIGID (W/SPRING CHEST) DUMMY MODEL'. Below it, the 'X-Axis Label' is 'Time' and the 'Y-Axis Label' is 'Resultant Force'. There is a checkbox for 'Modify Curve Legend' and a 'Legend Title' field with 'Contact Id'. On the right, a vertical list of options includes 'Grid', 'Tick', 'Frame', 'Legend', 'Autofit', 'Timeline', 'Invert', 'Maxgraph', and 'Minmax', with 'Grid', 'Tick', and 'Frame' checked. At the bottom, there are buttons for 'Show All', 'Show Select', 'Redraw', 'Apply', 'Reset', and 'Done'.

- Attr – modify curve attributes (symbol, color, width, style)



The screenshot shows the 'Attr' tab of the Plot Window. The top menu bar is the same as the previous window. The main area is divided into sections. On the left, the same list of curves is shown. In the center, the 'Pts/Sym' is set to '1'. Below it, there is an 'Apply Symbols' button and a checkbox for 'Join symbols with line'. To the right, there are four rows of attributes: 'Symbol' (a line), 'Color' (a blue square), 'Width' (a thick black line), and 'Style' (a dashed line). Each attribute has a dropdown menu, an 'Apply' button, and a 'Reset' button. At the bottom, there are buttons for 'Show All', 'Show Select', 'Redraw', 'Apply', 'Reset', and 'Done'. On the right, the same vertical list of options is shown, with 'Grid', 'Tick', and 'Frame' checked.

Plot Window (continued...)

- Filter – filter curves (SAE, Butterworth, COS, etc...)

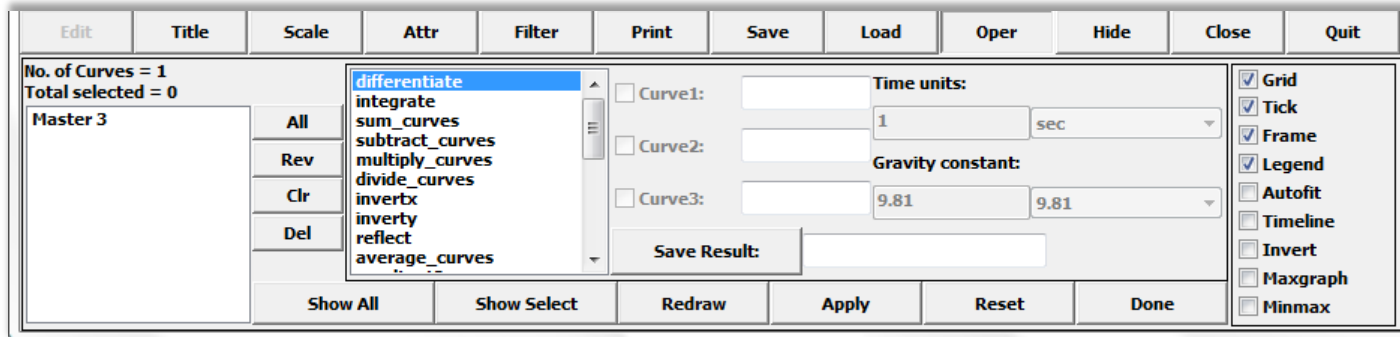
The screenshot shows the 'Filter' tab of the Plot Window. The top menu bar includes Edit, Title, Scale, Attr, Filter, Print, Save, Load, Oper, Hide, Close, and Quit. The main area is divided into several sections. On the left, it displays 'No. of Curves = 1' and 'Total selected = 0'. Below this is a list of curves, currently showing 'Master 3'. To the right of the list are buttons for 'All', 'Rev', 'Clr', and 'Del'. Further right is a 'Filter' dropdown menu set to 'sae', a 'Time' dropdown set to 'msec', and a 'C/s(Hz)' dropdown set to '60'. Below these is a 'Point Average' dropdown set to 'none'. At the bottom of the main area are buttons for 'Show All', 'Show Select', 'Redraw', 'Apply', 'Reset', and 'Done'. On the far right, there is a vertical list of checkboxes: 'Grid' (checked), 'Tick' (checked), 'Frame' (checked), 'Legend' (checked), 'Autofit' (unchecked), 'Timeline' (unchecked), 'Invert' (unchecked), 'Maxgraph' (unchecked), and 'Minmax' (unchecked).

- Save – write curves to file (.crv, Keep, XY Pairs, .csv, .xml)

The screenshot shows the 'Save' tab of the Plot Window. The top menu bar is the same as the previous window. The main area is divided into sections. On the left, it displays 'No. of Curves = 1' and 'Total selected = 0'. Below this is a list of curves, currently showing 'Master 3'. To the right of the list are buttons for 'All', 'Rev', 'Clr', and 'Del'. Further right is an 'Output Type' dropdown menu set to 'Curve file', and an 'Output Interval' dropdown set to '1'. Below these are checkboxes for 'Interpolate' (unchecked) and 'Curve Clip' (unchecked). To the right of these are radio buttons for 'Points' (selected) and 'Value' (unchecked). Below these are input fields for '#Pts' (set to 1000), 'Amin', and 'Amax'. Below these is a 'Path' input field set to 'C:\JACOB\' and a 'Filename' input field set to 'force_vs_time.crv'. To the right of the filename field is a 'Browse' button. At the bottom of the main area are buttons for 'Show All', 'Show Select', 'Redraw', 'Save', 'Reset', and 'Done'. On the far right, there is a vertical list of checkboxes: 'Grid' (checked), 'Tick' (checked), 'Frame' (checked), 'Legend' (checked), 'Autofit' (unchecked), 'Timeline' (unchecked), 'Invert' (unchecked), 'Maxgraph' (unchecked), and 'Minmax' (unchecked).

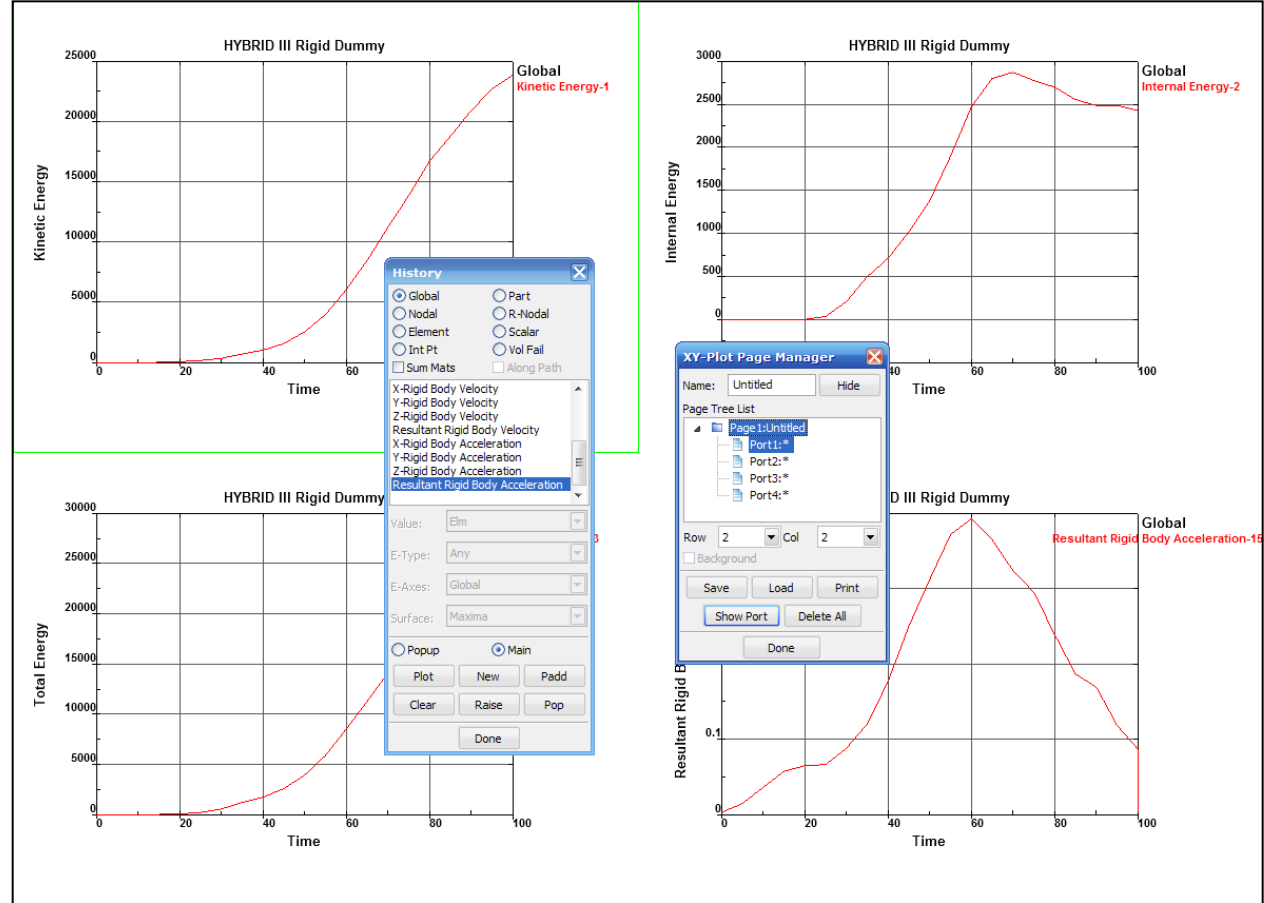
Plot Window (continued...)

- Oper – perform curve operations (integrate, sum, invert, etc...)



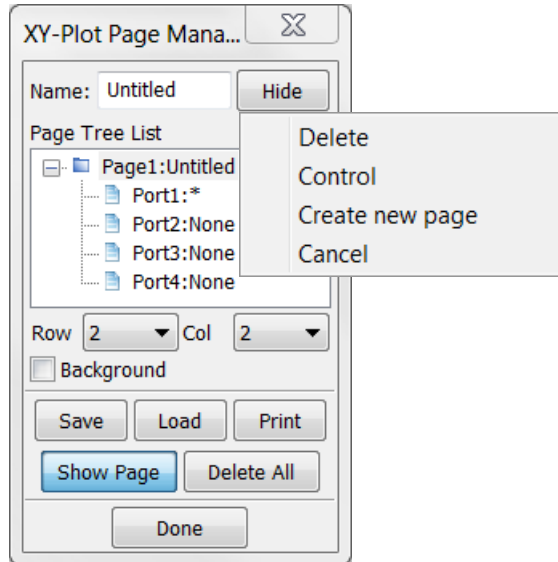
New Plot Interface

- New XYPlot interface puts plots in the **Main** window
- Multiple plots on a page / multiple pages
- Plots from one port can be easily moved or copied to another port

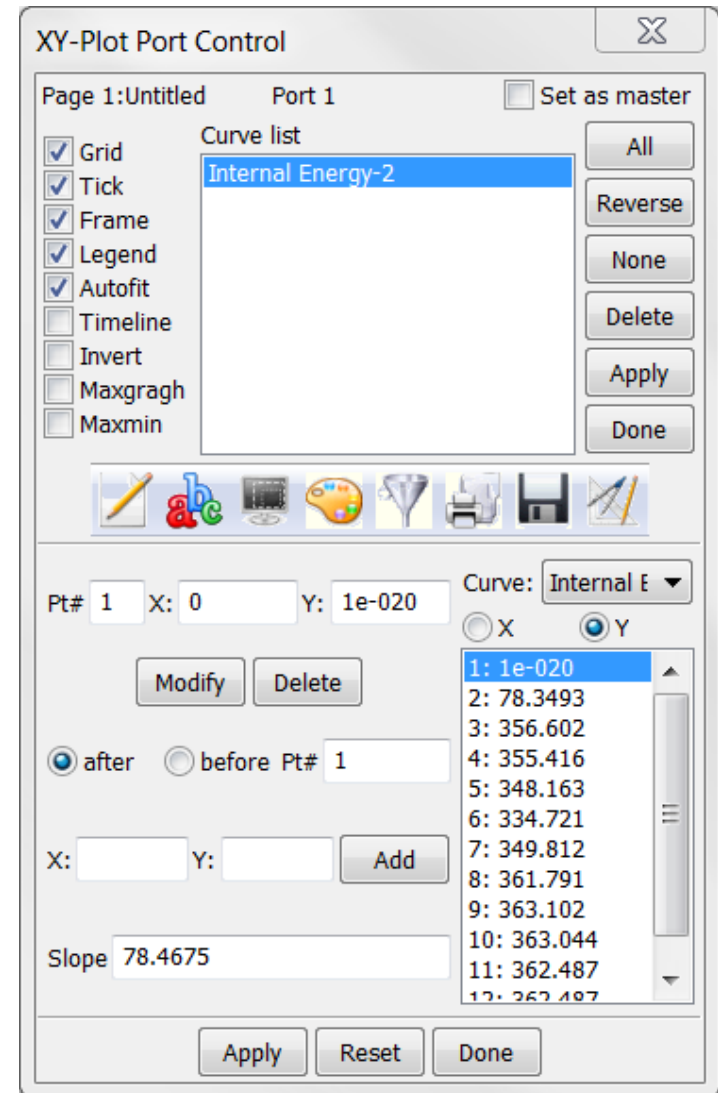


New Plot Interface *(continued...)*

- Right-click on a Port to **Delete** it or launch the **Control** interface



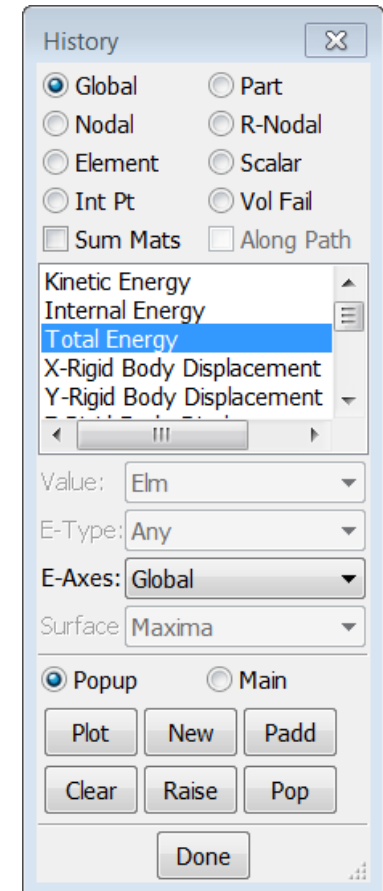
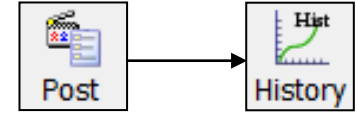
- Control interface allows plots to be customized



Post → History

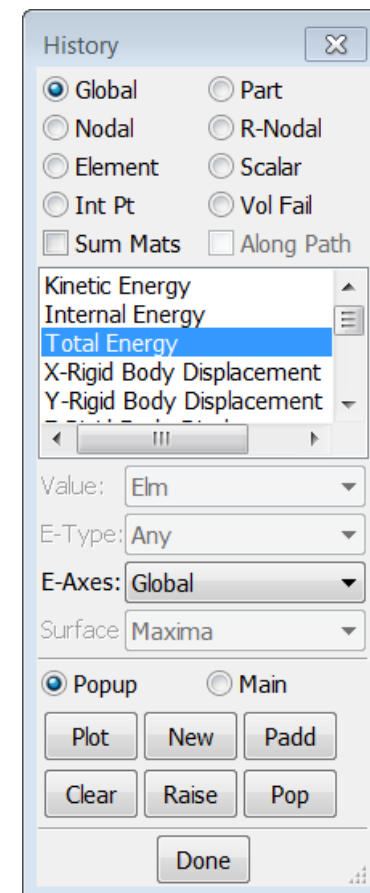
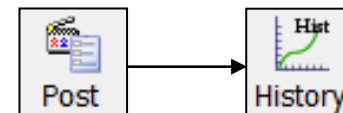
Purpose: plot time history data (from d3plots)

- Global – energies and kinematic data for full model
- Part – energies and kinematic data for parts
- Nodal – kinematic data for nodes
- Element – stress and strain data for elements
 - 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 location (through-thickness)
- Int Pt – integration point data
- Scalar – plot scalar value that is being fringed (choose component in Fcomp interface)



Post → History (continued...)

- Standard plotting options...
 - Plot – plot selected value in current Plot Window
 - New – plot selected value in new Plot Window
 - Padd – add selected value to current Plot Window
 - Raise – bring forward all open plot windows
 - Pop – open and bring forward all closed plot windows
- Interfaces that use these plot functions...
 - History
 - XYPlot
 - ASCII
 - Binout
 - FLD
 - Measure
 - Section > Force

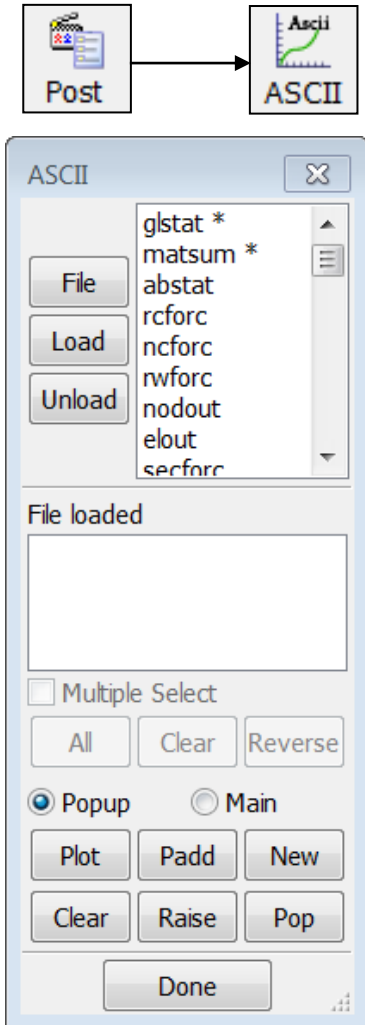


Post → ASCII

Purpose: create XY-Plots from ASCII output files

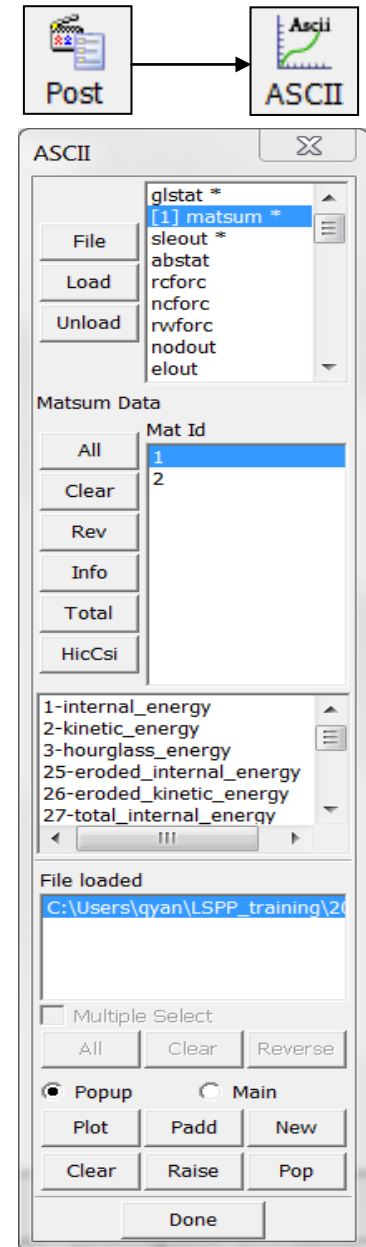
- ASCII File Operations...

- File – browse and load ASCII file from any directory
- Load – load selected file type from current directory
- Unload – unload files from memory
- All – select all items in the list
- Clear – clear selected items
- Rev – reverse selection
- Info – show information on the loaded ASCII file
- Multiple Select – plot multiple curves at once when multiple files are loaded (using File option above)



Post → ASCII (continued...)

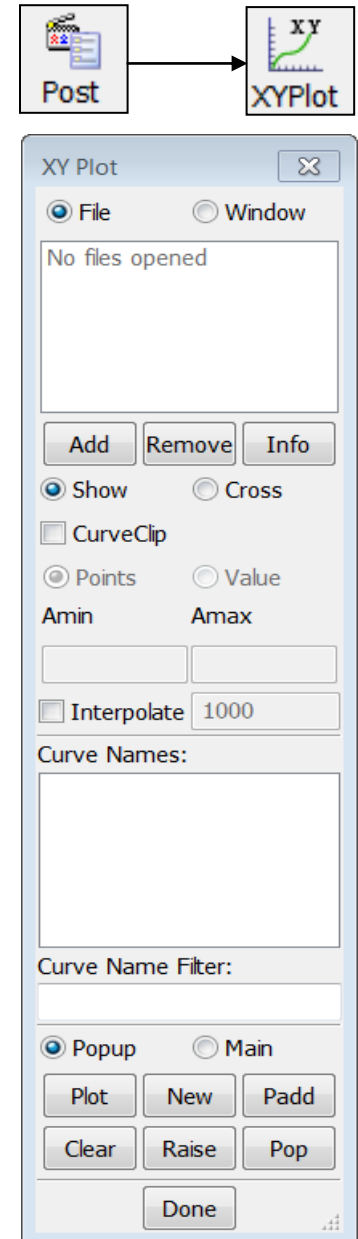
- ASCII Sub-branches...
 - Lists all data available in the selected ASCII file
 - Some ASCII files have special toggles that can be used to process data...
 - MATSUM, SECFORC, SPCFORC, NODFOR, BNDOUT, SLEOUT, and GCEOUT – [Total] combine multiple ASCII item selections
 - ABSTAT – [Airb/Mat] select airbag/material ID
 - ELOUT – [Clrcp] clear component list
 - ELOUT – [Stress/Strain/Beams]
 - DEFORC – [Trans/Rotat] force and moments
 - NODFOR – [Group] combine nodal forces of the group
 - NODOUT – [HicCsi] open HIC/CSI pre-filter options
 - RBDOUT – [Local] plot in local coordinate system
- SPCFORC – [Force/Momt] force and moments



Post → XYPlot

Purpose: create and manipulate XY-Plots (all saved XY data is automatically added to the file list)

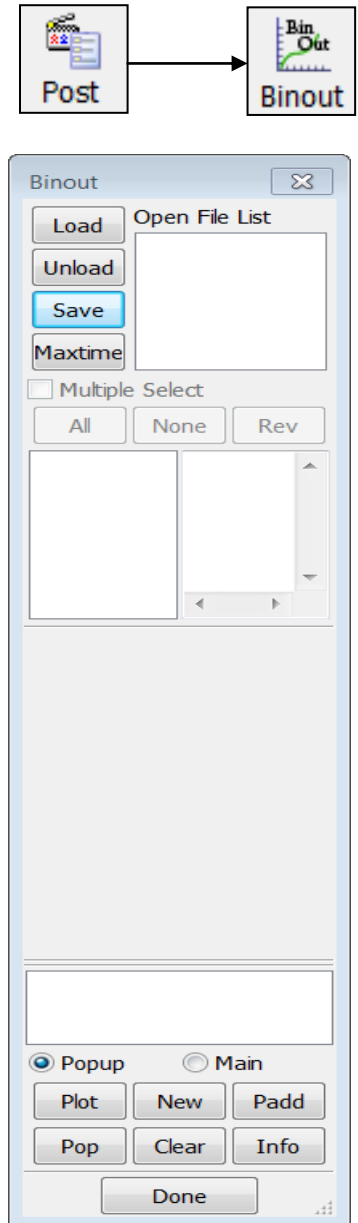
- File – show list of all loaded XY-Plot files
- Window – show list of all XY-Plot windows
- Add – import XY data (use CRV, CSV, XY formats)
- Remove – remove files from list
- Info – display number of data points
- Show – show selected plot
- Cross – create cross plot (e.g., combine Force/Time and Disp/Time to create Force/Disp)
- Curve Clip – clip curve
- Interpolate – modify number of points in curve



Post → Binout

Purpose: plot data from binout files

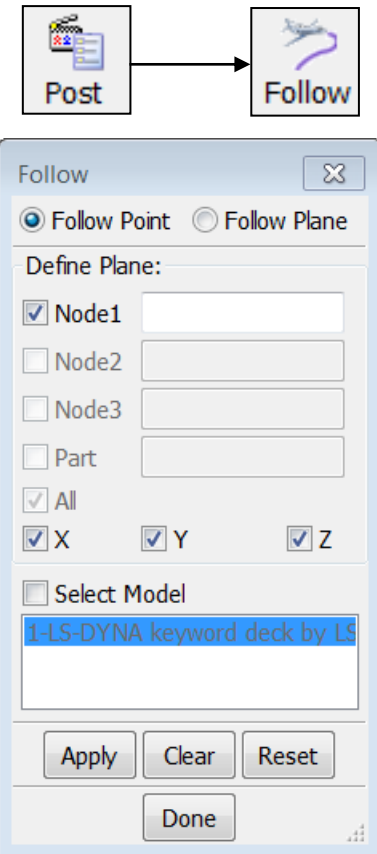
- Binout contains same data as ASCII files but in binary format
- Set BINARY=2 on *DATABASE_{*OPTION*} to generate binout file
- Load – load binout file (multiple binout files can be loaded)
- Unload – unload binout file
- Save – save binout branch in ASCII format
- Done – exit the Binout interface



Post → Follow

Purpose: define a reference point or plane for animation and fringing relative displacement

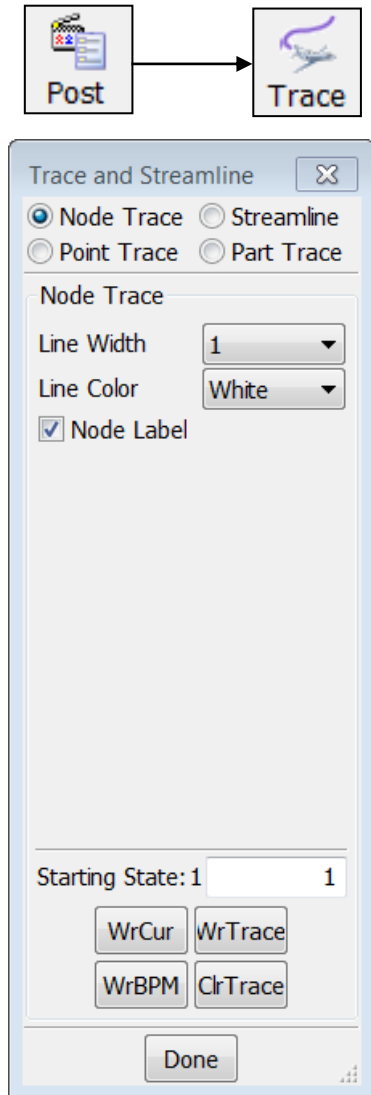
- Animation usage...
 - Pick node(s) to define reference point or plane
 - Click Apply
 - Play animation
- To fringe relative displacement...
 - Pick node(s) to define reference point or plane
 - Click Apply
 - Go to Page 1: Fcomp
 - Select Ndv
 - Select “rx-disp”, “ry-disp”, “rz-disp”, or “r-result disp”



Post → Trace

Purpose: trace nodal displacements

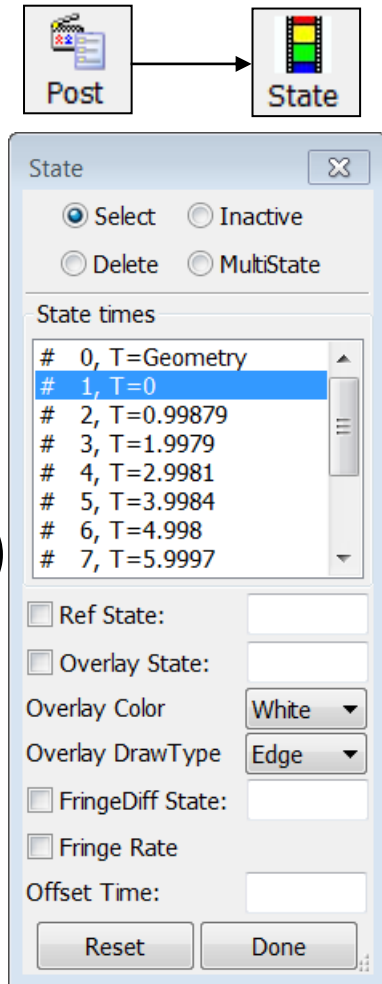
- Use General Selection interface to select nodes
- Set width and color of trace line
- Select state to begin trace
- Output trace in several formats...
 - *DEFINE_CURVE
 - Coordinate history
 - *BOUNDARY_PRESCRIBED_MOTION_NODE



Post → State

Purpose: select/overlay animation states or display multiple states in a grid

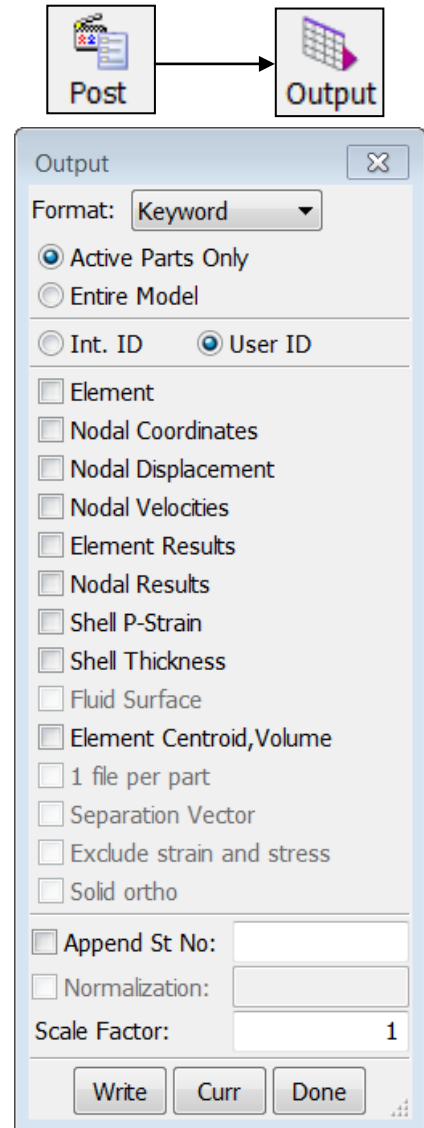
- Select – display the selected state or create state overlays (with customized rendering type)
- Inactive – make states inactive (remove from animation playback)
- Delete – delete states (delete d3plot files if possible)
- Multistate – display multiple states in a grid



Post → Output

Purpose: output post-processing data

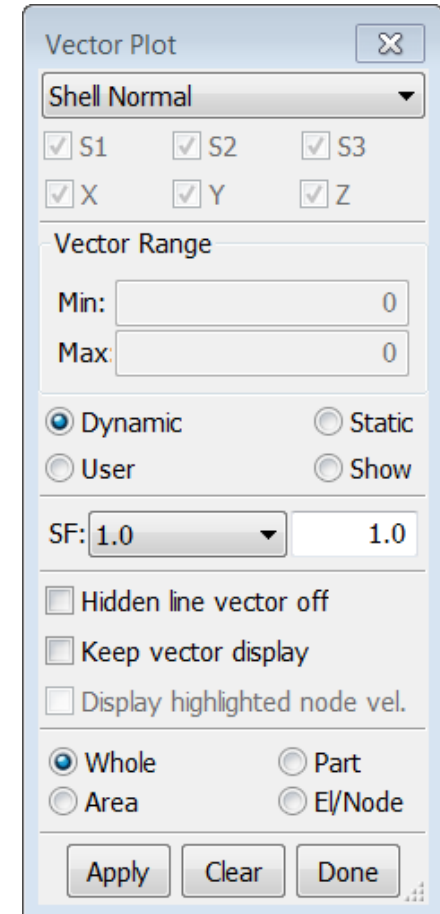
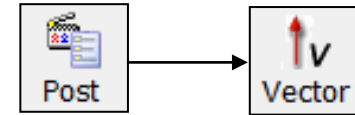
- Some available formats...
 - LS-DYNA Keyword
 - Nastran
 - Dynain ASCII
 - ASCII and Binary STL
- Active Parts Only – write only displayed parts
- Entire Model – write entire model (use check boxes to control exactly which entities are written)
- St No – Controls which state is written



Post → Vector

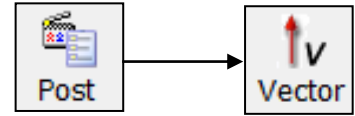
Purpose: create vector plots (similar to fringing)

- d3plot (binary plot) options...
 - Shell Normal
 - Displacement
 - Velocity
 - Principal Stress
 - Principle Strain
- intfor (interface force file) options...
 - Force
 - Pressure
 - Surface Shear

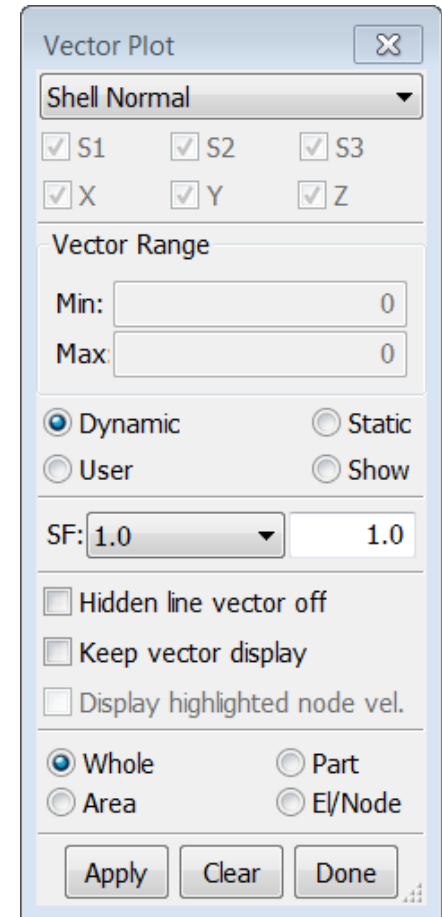


Post → Vector *(continued...)*

- X, Y, and Z components for displacement, velocity and force can be selected



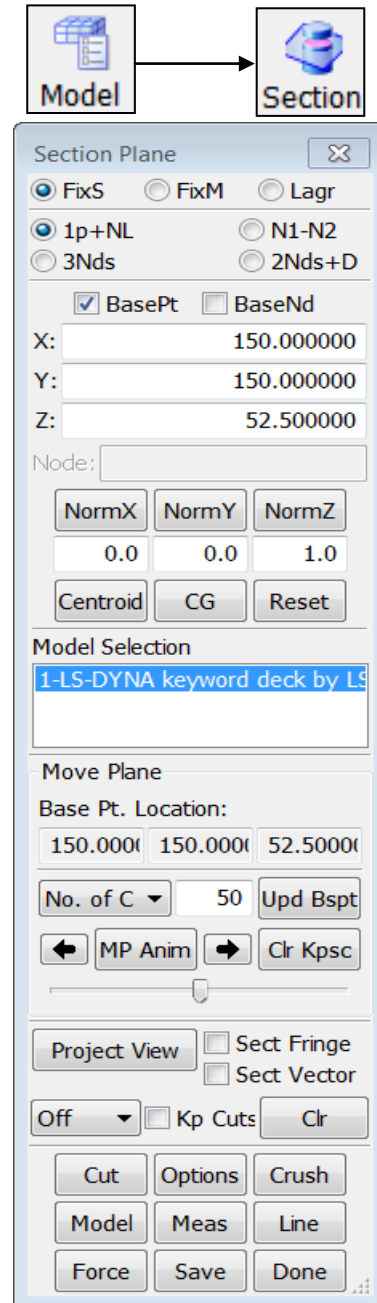
- Range settings...
 - Dynamic – min/max adjusted for each time state
 - Static – same min/max for all the states
 - User – custom min/max for all the states
 - Show – show vectors within the specified range
- Display settings...
 - SF – set vector size scale factor
 - Whole – display vectors for the whole model
 - Part – display vectors for a specific part
 - Area – display vectors for a user defined area
 - El/Node – display vectors for specific elements/nodes



Model → Section Plane

Purpose: visualize section planes or take measurements

- Section plane options...
 - FixS – section plane is fixed in space
 - FixM – section plane is fixed to the model (3Nds only)
 - Lagr – section plane is lagrangian type
- Plane definition options...
 - 1p+NL – base point + normal direction
 - N1-N2 – n1 defines base point, n1 to n2 defines direction
 - 3Nds – 3 nodes define the plane
 - 2Nds+D – 2 nodes + direction define the plane
 - BasePt/BaseNd – enter XYZ or select node as base point
 - NormX/NormY/NormZ – set direction cosines
 - Centroid – make centroid of model the base point
 - Reset – reset and clear plane definition



Model → Section Plane *(continued...)*

- Additional options...
 - No. of Cut/Distance - divide model along the plane normal direction
 - Right/Left Arrows – move the plane forward/backward
 - MP Anim – animate the section
 - Upd Bspt – update base point with current position
 - Clear Kpsc – clear all kept section cuts from memory

The screenshot shows the 'Section Plane' dialog box with the following settings:

- FixS** (selected), **FixM**, **Lagr**
- 1p+NL** (selected), **N1-N2**, **3Nds**, **2Nds+D**
- BasePt** (checked), **BaseNd** (unchecked)
- X:** 150.000000, **Y:** 150.000000, **Z:** 52.500000
- Node:** (empty)
- NormX**, **NormY**, **NormZ** (0.0, 0.0, 1.0)
- Centroid**, **CG**, **Reset**
- Model Selection:** 1-LS-DYNA keyword deck by LS
- Move Plane:**
 - Base Pt. Location:** 150.0000, 150.0000, 52.5000
 - No. of C** (dropdown): 50, **Upd Bspt**
 - MP Anim** (left arrow), **Clr Kpsc** (right arrow)
- Project View** (checkbox), **Sect Fringe** (checkbox), **Sect Vector** (checkbox)
- Off** (dropdown), **Kp Cuts** (checkbox), **Clr**
- Cut**, **Options**, **Crush**
- Model**, **Meas**, **Line**
- Force**, **Save**, **Done**

Model → Section Plane *(continued...)*

■ Additional options...

- Project View – view section normal to section plane
- Off/Clip – turn clipping planes off/on
- Kp Cuts – keep and display all section cuts
- Clr – clear all section cuts (keep only the last cut)
- Cut – perform section cut
- Options – display additional section plane options
- Crush – open crush/intrusion interface
- Model – reset display to full model view
- Meas – open section cut measurement interface
- Line – open line plot interface
- Force – open section force interface
- Save – open interface to save/load planes (to file)
- Done – exit the Section Plane interface

The screenshot shows the 'Section Plane' dialog box with the following settings:

- FixS** (selected), **FixM**, **Lagr**
- 1p+NL** (selected), **N1-N2**, **3Nds**, **2Nds+D**
- BasePt** (checked), **BaseNd** (unchecked)
- X:** 150.000000, **Y:** 150.000000, **Z:** 52.500000
- Node:** (empty)
- NormX** (0.0), **NormY** (0.0), **NormZ** (1.0)
- Centroid**, **CG**, **Reset**
- Model Selection:** 1-LS-DYNA keyword deck by LS
- Move Plane:**
 - Base Pt. Location:** 150.0000, 150.0000, 52.50000
 - No. of C:** 50, **Upd Bspt**
 - MP Anim** (left arrow), **MP Anim** (right arrow), **Clr Kpsc**
- Project View** (checked), **Sect Fringe** (unchecked), **Sect Vector** (unchecked)
- Off** (selected), **Kp Cuts** (unchecked), **Clr**
- Cut**, **Options**, **Crush**
- Model**, **Meas**, **Line**
- Force**, **Save**, **Done**

Model → Section Plane *(continued...)*

- “Options” Interface
 - ShowPl – show section plane on/off
 - 3DOutline – show solid part outline after cut
 - ShMesh – show section plane as mesh
 - Line Width – set section cut line width
 - Line Color – set section cut line color
 - Color/Cut – use different color for each cut instead of different color for each part
 - Thickness – draw section cut with thickness
 - VP – draw vector on section cut
 - Write – write section cut to file in selected format (Keyword, VGA, or IGES)
 - Curr State – set current state for writing

The screenshot shows the 'Section Plane' dialog box with the following settings:

- FixS** (selected), **FixM**, **Lagr**
- 1p+NL** (selected), **N1-N2**, **3Nds**, **2Nds+D**
- ☒ **BasePt**, ☐ **BaseNd**
- X:** 150.000000, **Y:** 150.000000, **Z:** 52.500000
- Node:** (empty)
- NormX**, **NormY**, **NormZ** (0.0, 0.0, 1.0)
- Centroid**, **CG**, **Reset**
- Model Selection:** 1-LS-DYNA keyword deck by LS
- Move Plane:**
 - Base Pt. Location:** 150.0000, 150.0000, 52.5000
 - No. of C** (dropdown): 50, **Upd Bspt**
 - MP Anim** (left arrow), **Clr Kpsc** (right arrow)
- Project View** (selected), ☐ **Sect Fringe**, ☐ **Sect Vector**
- Off** (dropdown), ☐ **Kp Cuts**, **Clr**
- Cut**, **Options**, **Crush**
- Model**, **Meas**, **Line**
- Force**, **Save**, **Done**

Model → Section Plane *(continued...)*

- “Crush” Interface – measure distance from node to plane (intrusion plotting)
 - Line Width – Intrusion plot line width
 - Line Color – Intrusion plot line color
 - PlotType – select history plot type
 - Node ID – can be picked or keyed in
- “Line” Interface – plot fringe values at section cut along the length of the cut
 - Mainly used for metal stamping
 - Must load fringe value (using Fcomp) and perform cut first
 - Pick a part for the line plot

The screenshot shows the 'Section Plane' dialog box with the following settings:

- FixS** (selected), **FixM**, **Lagr**
- 1p+NL** (selected), **N1-N2**, **3Nds**, **2Nds+D**
- BasePt** (checked), **BaseNd** (unchecked)
- X:** 150.000000, **Y:** 150.000000, **Z:** 52.500000
- Node:** (empty field)
- NormX** (0.0), **NormY** (0.0), **NormZ** (1.0)
- Centroid**, **CG**, **Reset**
- Model Selection:** 1-LS-DYNA keyword deck by LS
- Move Plane:**
 - Base Pt. Location:** 150.0000, 150.0000, 52.5000
 - No. of C** (dropdown): 50, **Upd Bspt**
 - MP Anim** (left arrow), **Clr Kpsc** (right arrow)
- Project View** (checkbox), **Sect Fringe** (checkbox), **Sect Vector** (checkbox)
- Off** (dropdown), **Kp Cuts** (checkbox), **Clr**
- Cut**, **Options**, **Crush**
- Model**, **Meas**, **Line**
- Force**, **Save**, **Done**

Model → Section Plane *(continued...)*

- “Force” Interface – calculate and plot section values
 - Forces
 - Moments
 - Area
 - Centroid
- “Meas” Interface – section plane measurements
 - Coordinate
 - Distance
 - 3Pt-Angle
 - 3Pt-Radius
 - 2Ln-Angle
 - Measure can be done between any two cuts

The screenshot shows the 'Section Plane' dialog box with the following settings:

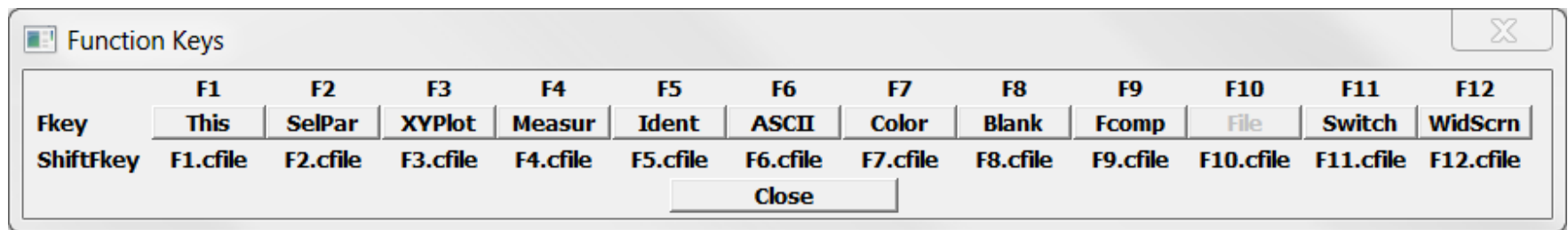
- FixS** (selected), **FixM**, **Lagr**
- 1p+NL** (selected), **N1-N2**, **3Nds**, **2Nds+D**
- BasePt** (checked), **BaseNd** (unchecked)
- X:** 150.000000, **Y:** 150.000000, **Z:** 52.500000
- Node:** (empty)
- NormX** (0.0), **NormY** (0.0), **NormZ** (1.0)
- Centroid**, **CG**, **Reset**
- Model Selection:** 1-LS-DYNA keyword deck by LS
- Move Plane:**
 - Base Pt. Location:** 150.0000, 150.0000, 52.5000
 - No. of C** (dropdown): 50, **Upd Bspt**
 - MP Anim** (left arrow), **Clr Kpsc** (right arrow)
- Project View** (checked), **Sect Fringe** (unchecked), **Sect Vector** (unchecked)
- Off** (dropdown), **Kp Cuts** (unchecked), **Clr**
- Cut**, **Options**, **Crush**
- Model**, **Meas**, **Line**
- Force**, **Save**, **Done**

Configuration

- Configuration files
 - Lsppconf – record general configuration parameters, user the pull down menu “Setting”->“Configuration Settings” to set parameters
 - Lspplasttouch – record last session windows size, dialog location, file path, etc.
 - Configure_Toolbar.cfg – record toolbar configurations
 - .lspp_recent – record recently used files and their locations
- Configuration file location (each version of LS-PrePost has its own directory)
 - Windows – C:\Users\uname\AppData\Roaming\LSTC\LS-PrePostx.x
 - Linux - \$HOME/LSTC/LS-Prepostx.x

Function Keys

- F1 launches Function Key interface shown below (shows layout of the function keys)
- Except for F1 and F10, all other function keys can be programmed to act as a button
- Also, a command file with the name F#.cfile can be called using F# (place file in current working directory)
- Function keys can be customized in the configuration file (lsppconf)



Workshop 6

Post-Processing

- ❖ Animation interface
- ❖ Fringe (contour) plotting
- ❖ Time history/ASCII data plotting
- ❖ Section cut
- ❖ Cross plotting
- ❖ Vector draw

Thank You!