

Current Features of LS-PREPOST

5th European LS-DYNA Users' Conference

May 25, 2005



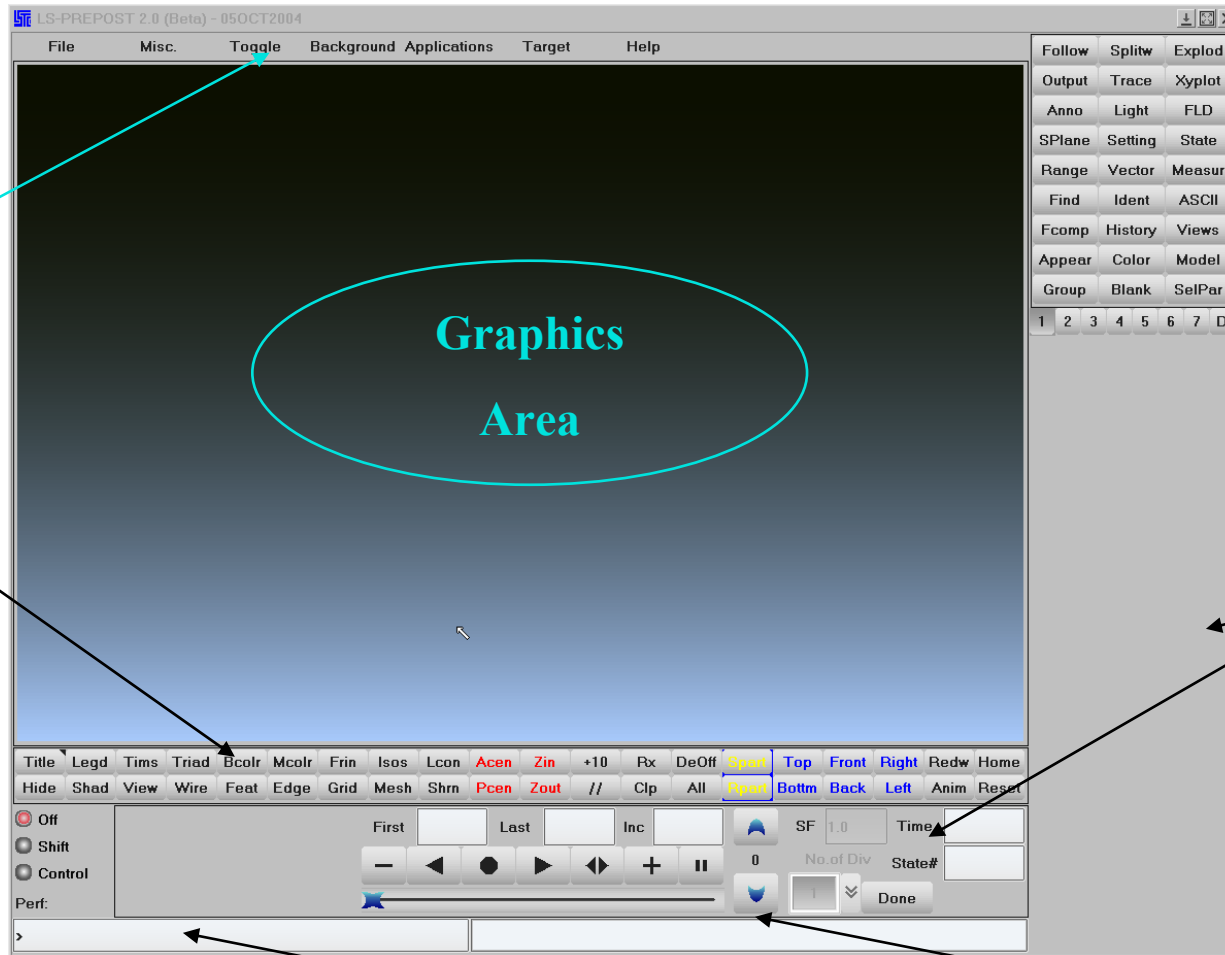
Overview

LS-PREPOST is an advance pre-processor and post-processor for LS-DYNA

- Full support of LS-DYNA keywords
- Full post-processing capabilities for LS-DYNA
- LS-DYNA model Visualization
- Model creation and editing



Interface Layout



Pull Down Menu

Main Button Menu

Page Menu

Hot button Area

Interface Working Areas

Command Area Button Description

Message Dialog





General Operations

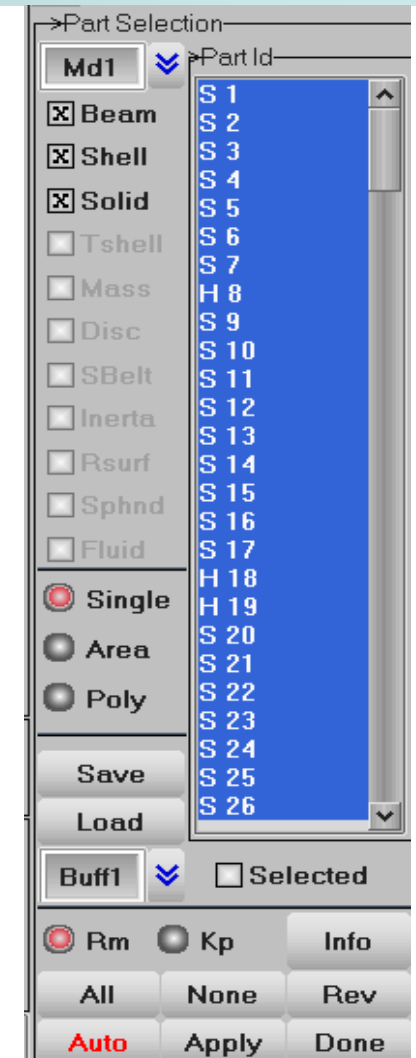
- Part Selection – selection parts on/off
- Identify – identify elements/nodes/parts
- Find – find a specified elements/nodes/parts
- Blank – blank out part of the model
- Appearance – set the appearance (how model is being rendered)
- Lighting – turn on/off lights, set position and RGB values
- Coloring and Transparency – set color and/or transparency of parts
- Grouping – group parts together as a group
- Annotation – annotate on the graphics window (2D only)
- Explode – explode parts to give a better view of the model
- Viewing – Save and retrieve a view (the orientation of the model)
- Macro command – group commands together to become a macro



Part Selection

Purpose: To turn on and off parts

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

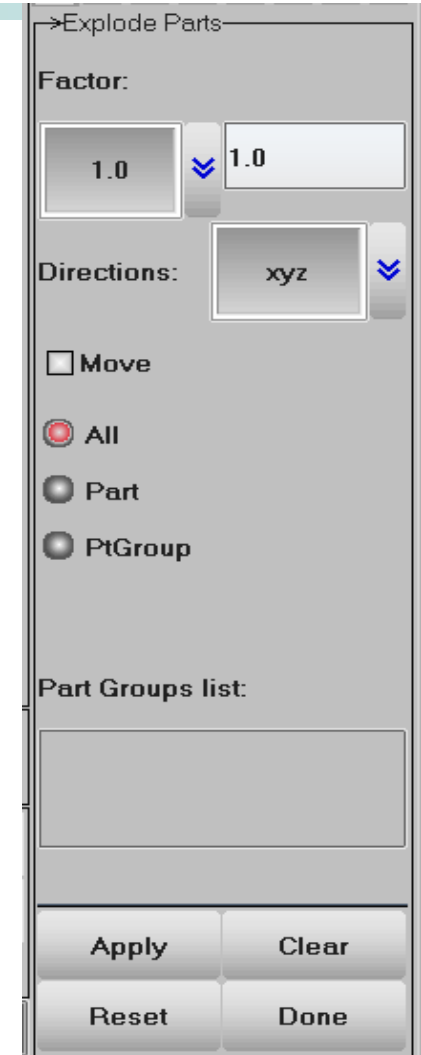




Explode Parts

Explode/separate parts for better visualization

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

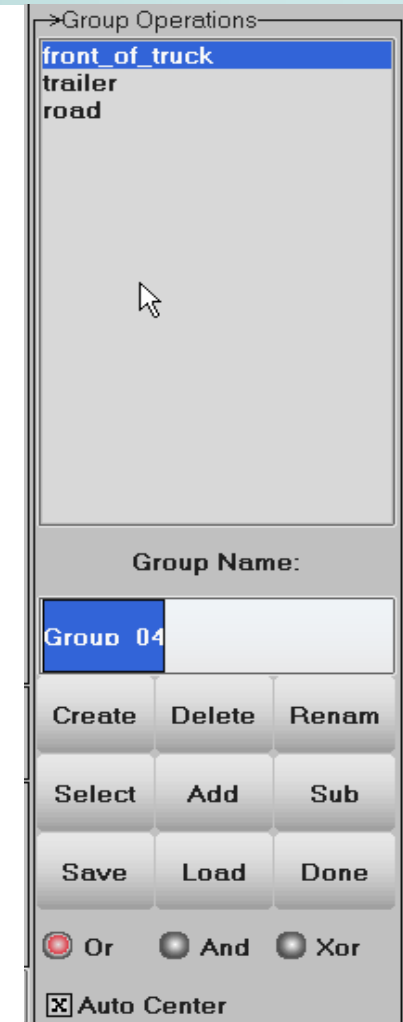




Group

Purpose: Allows users to create groups of parts.

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

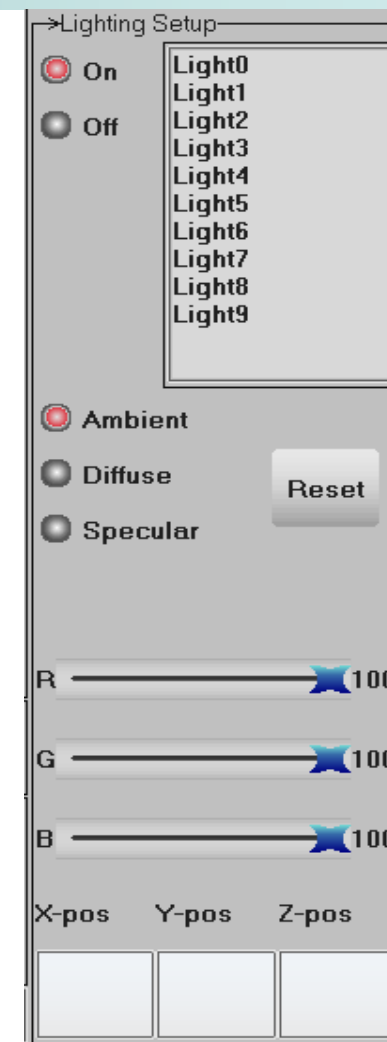




Lighting Setup

Purpose: To setup lighting for the model

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





Color and Transparency

Allows users to change

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

R 26
G 39
B 64

Set Show

Backg Text Sky
Mesh Label Middle
Hilite Outlin Ground

>Set Part Color

Pick Part

Area In

Polygon Out

Color Transpar

>Part Colormap

Red	Blue	Green	Yellow	Brown	Pink
Dark Blue	Light Green	Yellow	Brown	Red	Light Blue
Green	Yellow	Brown	Purple	Orange	Light Red
Light Brown	Light Green	Brown	Purple	Light Green	Light Blue
Pink	Light Green	Blue	Pink	Cyan	White
White	White	White	White	White	White
Grey	Grey	Grey	Grey	Grey	Grey

Editmap

AllVis Amap Rstp
Snap Lmap Done

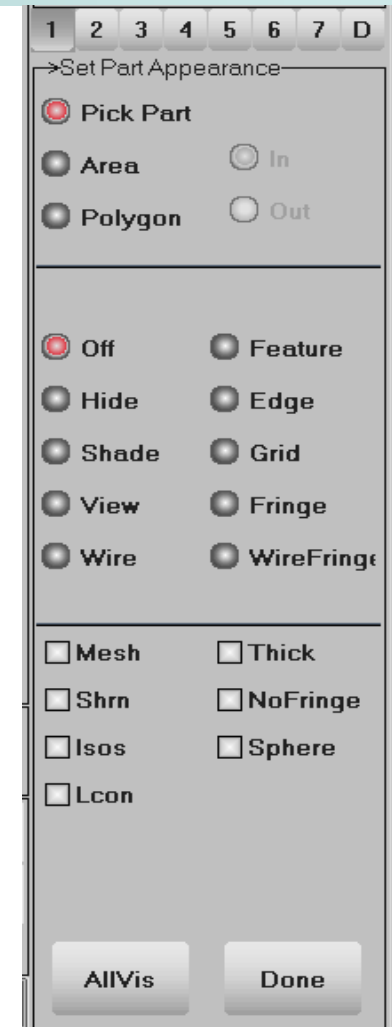




Appearance

Purpose: To set the appearance of each part

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





Identity Nodes/Elements/Parts

Purpose: To identify nodes/elements/parts

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

→Identify Node/Ele/Part

Pick Entity

Area In

Polygon Out

Key in ID:

Key in xyz coord:

Node

Element

Part

Show Results

Show Ele Direction

Show Mat Direction

Show Popup

Unpick Last

Clear Node	AllVis
Clear Ele	Clear All
Clear Part	Done



Orthotropic Material Axes

Orientation Specimen Tension

The image shows a 3D finite element mesh of a specimen, colored red. Three sets of orthogonal axes (A, B, C) are shown at different nodes: H 494, H 529, and H 575. Axis A is red, B is green, and C is blue. A small 3D coordinate system (x, y, z) is visible in the bottom left corner of the plot area.

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

Output Traj
Anno Ligt
SPlane Setti
Range Vec
Find Ide
Fcomp Hist
Appear Col
Group Blar

1 2 3 4
Identify Nod
◆ Pick Entity
◇ Area
◇ Polygon
Key in ID:
Key in xyz coord:
◇ Node
◆ Element
◇ Part
 Show Results
 Show Ele Dir
 Show Mat Dir
 Show Popup





Find Nodes/Elements/Parts

Purpose: To find a node/element/parts

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

The screenshot shows a dialog box titled "Find Node/Element/Part". It contains several radio button options for selection:

- Node
- Element
- Part
- Any
- Beam
- Shell
- Solid
- Tshell
- Sphnode

Below these options are two more radio buttons:

- Show Only
- Highlight

There is a text input field labeled "Node/Elem/Part Id".

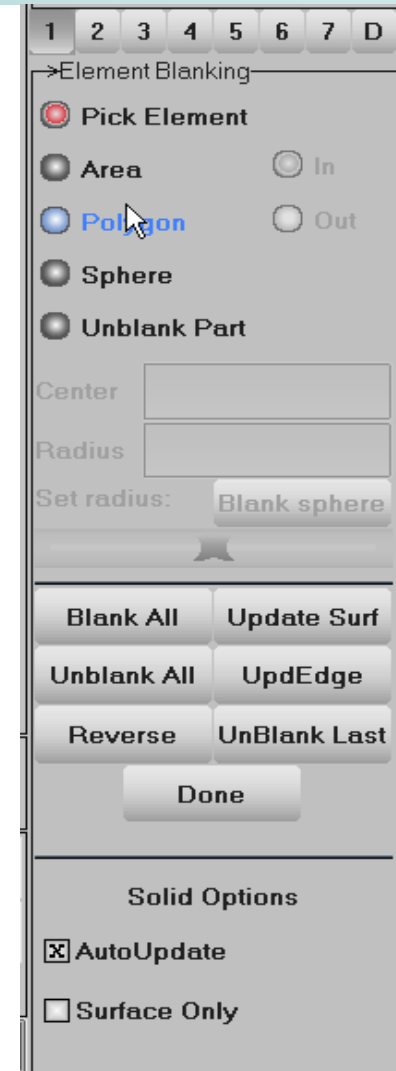
At the bottom, there are several buttons: "Find", "Neighbors", "Done", "UnblankAll", and "UnblankPart".



Blank Elements

Purpose: To blank element from display

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

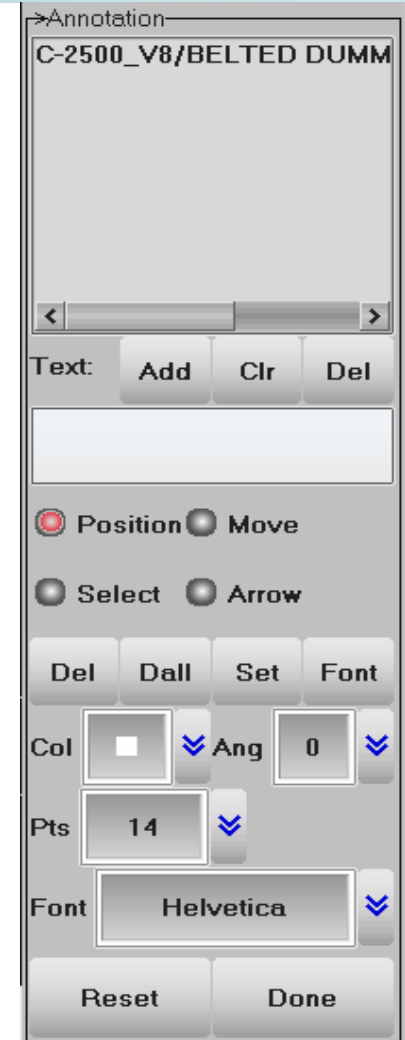




Annotate

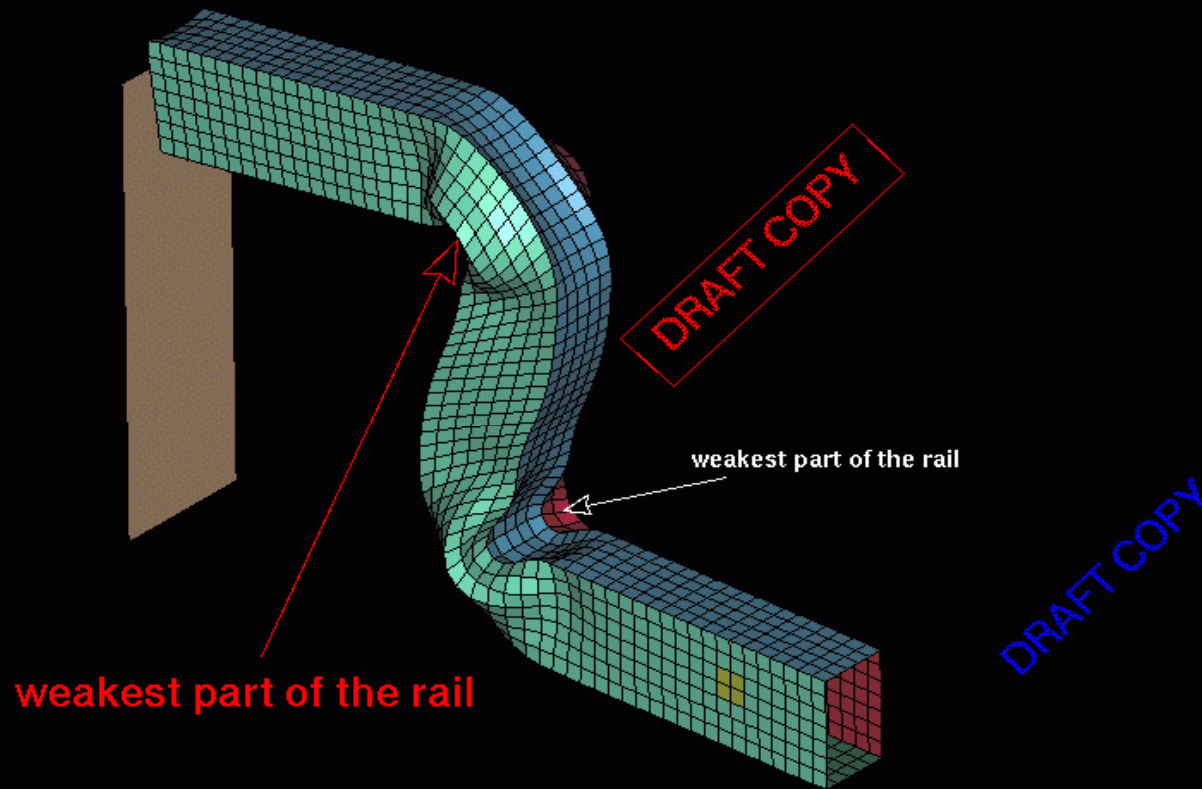
Purpose: Anonate graphics window and xy-plot windows

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



Anno - Annotations graphics window and xy-plots

Time = 20



Follow	Splitw	Explod
Output	Trace	Xyplot
Anno	Light	FLD
SPlane	Setting	State
Range	Vector	Measur
Find	Ident	ASCII
Fcomp	History	Views
Appear	Color	Model
Group	Blank	SelPar

1 2 3 4 5 6 7 D

Annotation

Anno - Annotations graphics ' weakest part of the rail
DRAFT COPY

Text: Add Clr Del

[]

◇ Position ◇ Move

◇ Select ◇ Arrow

Del Dall Set Font

Col ■ Ang 45

Pts 24

Font Helvetica

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

First 1 Last 21 Inc 1

SF 1.0 Time 20

No. of Div State# 21

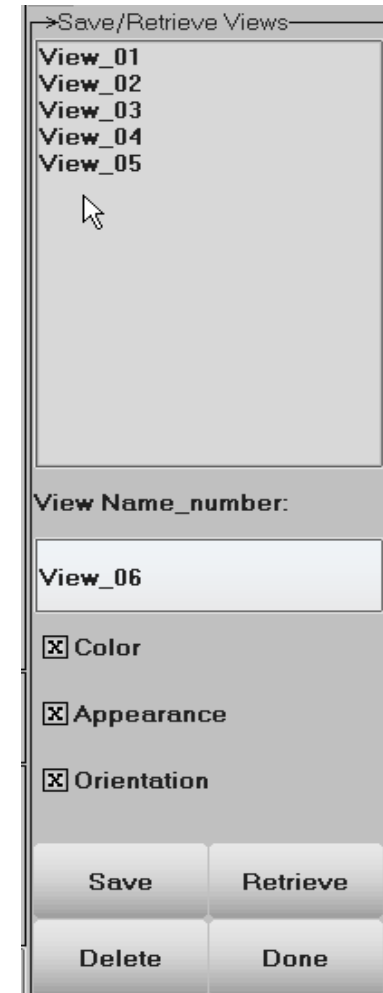




Views Interface

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

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





Macro Command File

- A Macro is a group of commands that bundle together for repeated operation
- A Macro command file can contains many Macros
- Syntax of macro:
 - *macro begin name
 - Command 1
 - Command 2
 - Command 3
 - *macro end
- Parameters can be used in the macro
- Parameters can be pre-defined or define on the fly



Macro Commands

- ❑ Add – add macro from file
- ❑ Remove – remove macro from memory
- ❑ Show – show the content of a selected macro
- ❑ Param – show the parameters that are used in the selected macro
- ❑ Exec – execute the selected macro





Macro Command - Parameters

- ❑ Parameters are defined by ‘&’ before the parameter name
 - ❑ E.g. rotate_model &xangle &yangle &zangle
 - ❑ Where xangle, yangle, zangle are parameter names
 - ❑ Parameters can be predefined in another command file:
 - Parameter xangle 50.0
 - Parameter yangle 120.0
 - Parameter zangle 45.0
 - ❑ If parameters were not pre-defined, they can be defined using interface before execution
 - ❑ Parameters can be redefined at any time





Post-Processing

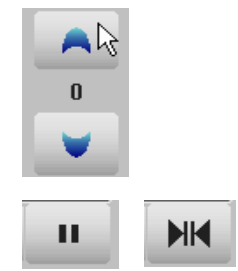
- Animation – animation control including eigenvector animation
- Fringe plots – fringe components selection
- Range setting – set min/max value for fringe plot
- Measure – coordinates, distances, angles, areas, volumes, mass, etc.
- Follow – follow a point or a plane
- State control – select and deselect states
- Trace – trace nodal displacement through all states
- Section plane – section cut with many facilities
- Vector plot – velocity, principal stress/stain
- History plots – from d3plot or d3thdt files
- ASCII – handle all LS-DYNA ascii files
- Binout – binary format of all ASCII files
- Output – output geometry or result for any states



Animation

Purpose: Control Animation state

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

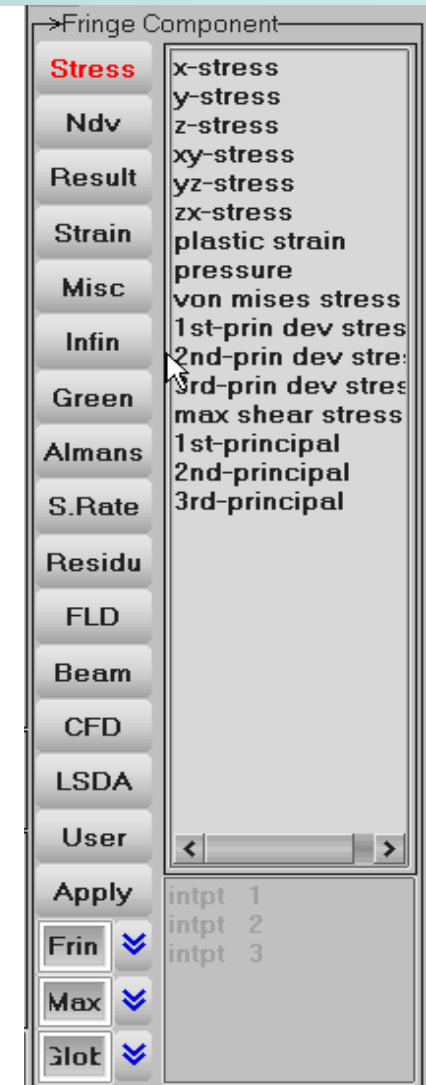




Fcomp – Fringe Component Selection

Purpose: To select fringe components

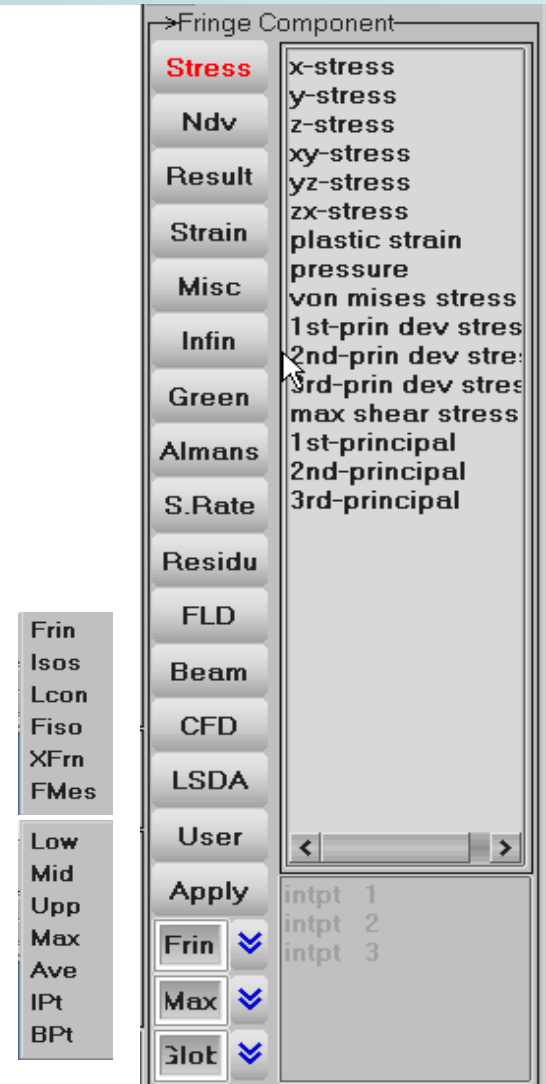
- Stress – regular stress components
- Ndv – nodal displacement and velocity
- Result – stress resultant components
- Strain – regular strain components
- Misc – miscellaneous components like pressure, temperature, internal energy, shell thickness, etc.
- Infin – infinitesimal strains
- Green – Green-St.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





Fcomp – Fringe Component Selection

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

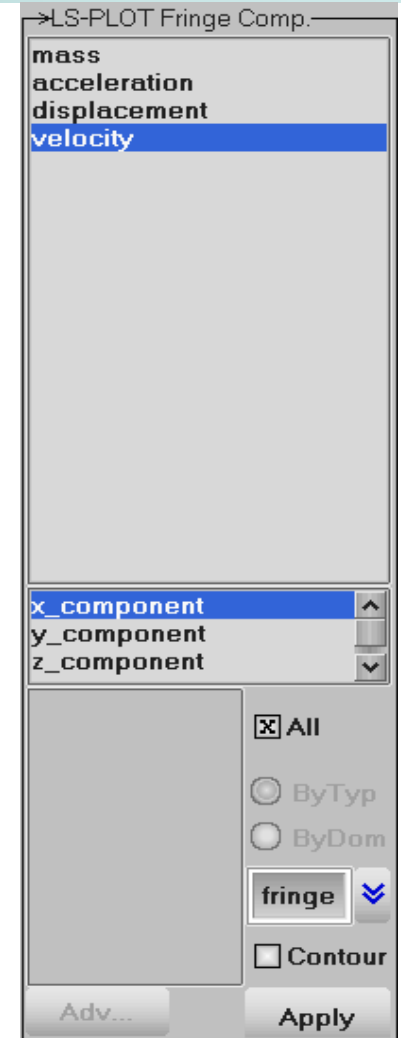




Fcomp - LSPLLOT

• Processing new database from LS-DYNA

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

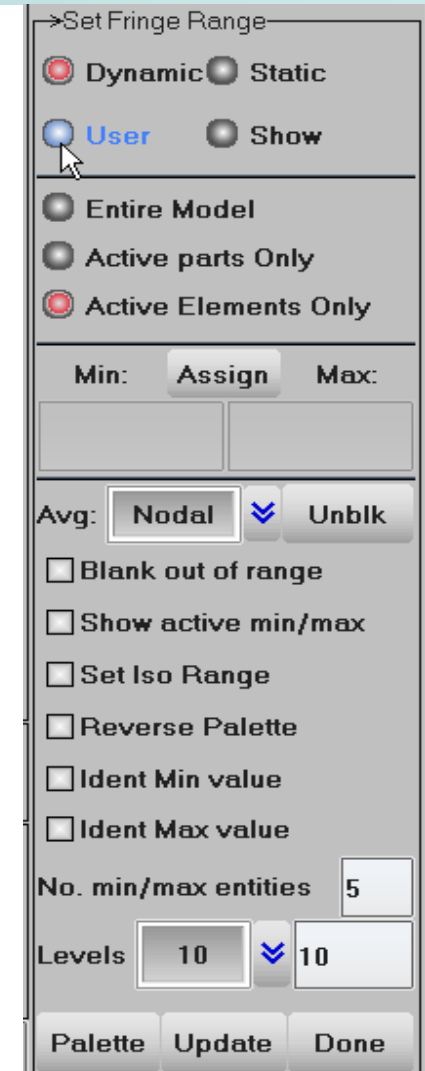
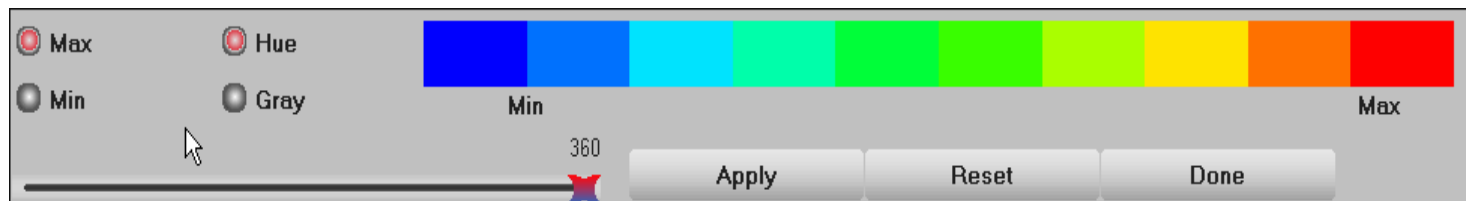




Range

Range allows user set fringe and iso-surface ranges

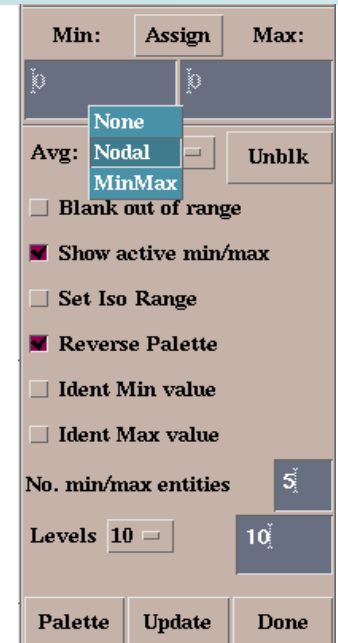
- Dynamic - min/max is different for each time state
- Static sets the same min/max for all the states
- User specifies a defined min/max for all the states
- Show – Shows elements within the specified range
- Choose entire model, active parts only or active elements only
- Color Palette can also be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



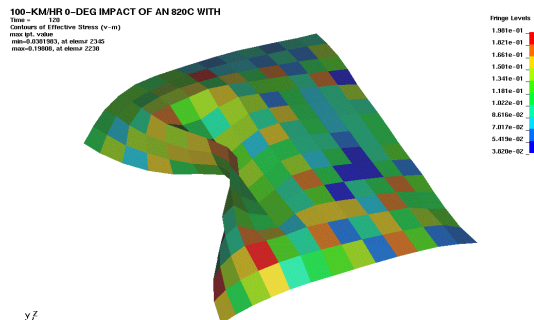


Range

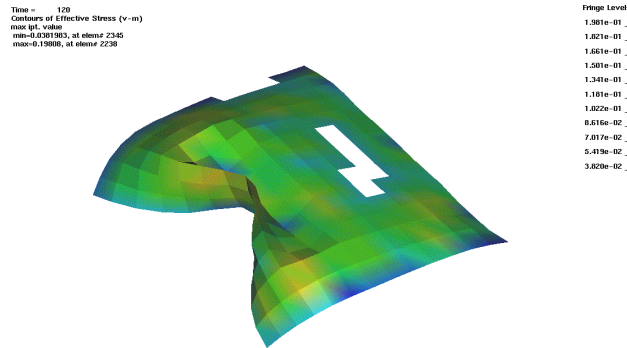
- ❑ Elements out of range can be blanked out
- ❑ The color palette can be reversed with red showing min and blue max
- ❑ Min/Max elements can be identified along with a user defined number of elements to be shown
- ❑ Number of fringe levels can be set to 10, 20 and 30
- ❑ Average can be set to none,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



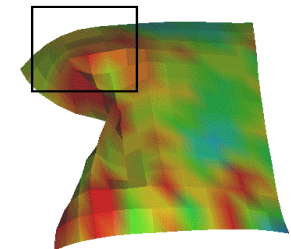
No Average



Out of range Elements blanked



Min/max in selected area shown





Measure

Purpose: To make measurements of various items

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



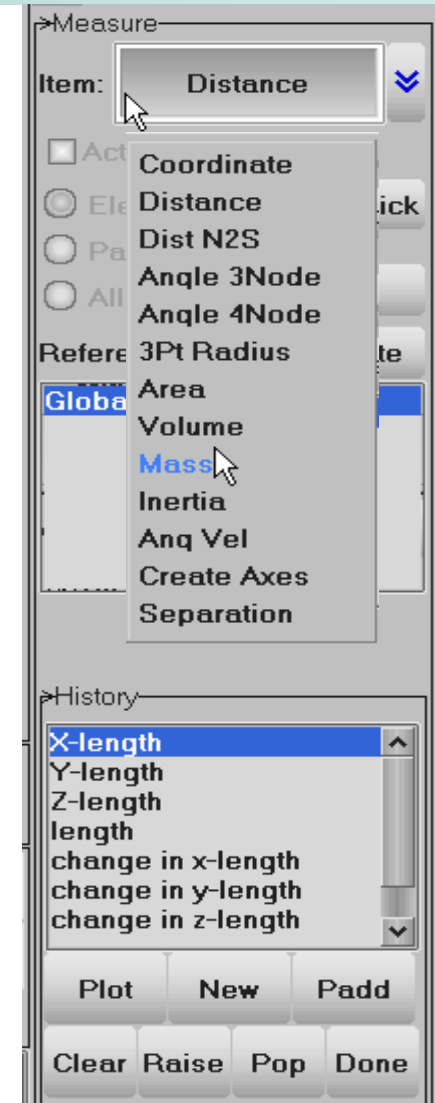


Measure

Purpose: To make measurements of various items

☐ Items to be measured

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

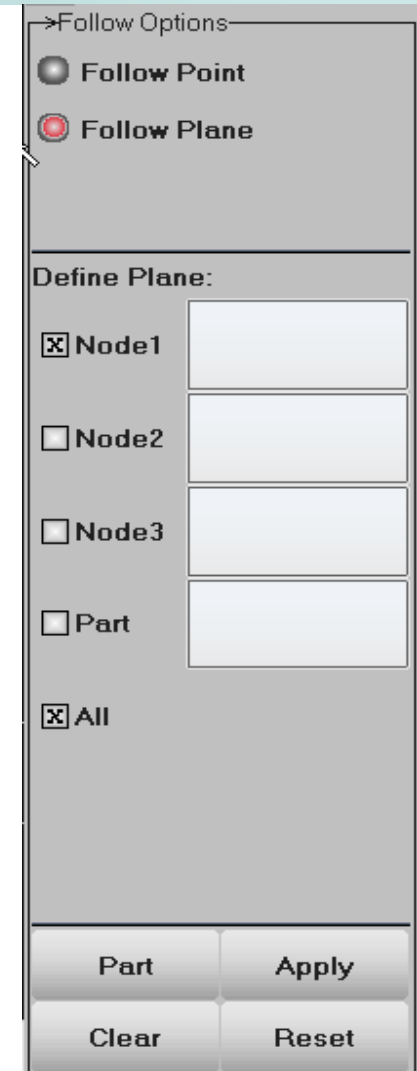




Follow

This interface is used for model animation and post processing

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



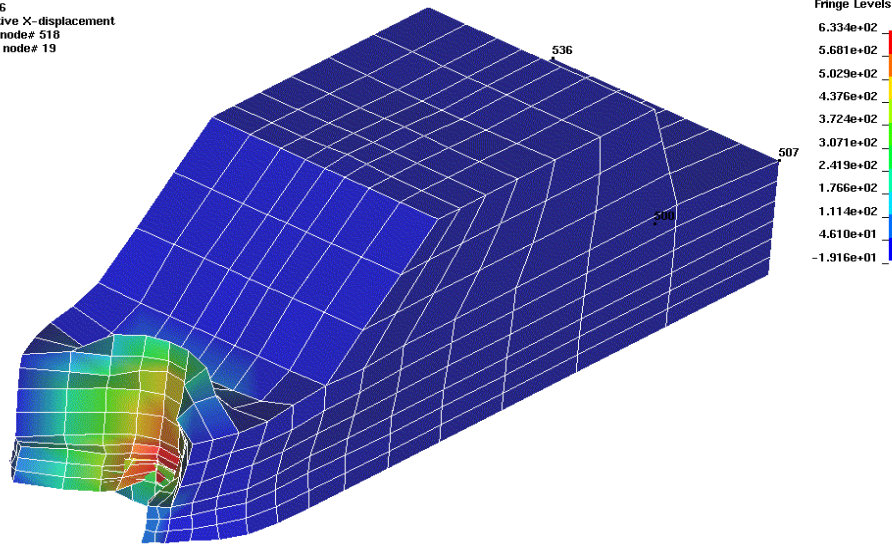


Follow

Relative displacement contour plot

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

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



→Fringe Component

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

Relative displacement fringes





State Control

- Direct access to a single state

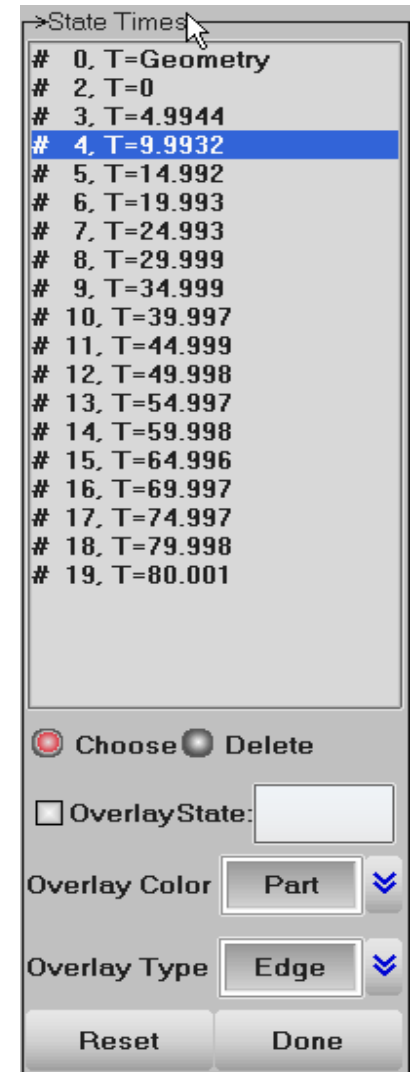
- User may directly select a state in the state list, LSPREPOST will bring the state of the model requested.

- Turn off un-wanted states

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

- Overlay different states in various ways

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





State Control with Overlay

TERMINATION TIME = 80 MS
Time = 80.001

>State Times

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

Choose Delete

OverlayState: 1

Overlay Color: White

Overlay Type: Feat

Reset Done



Trace Node plot

Purpose: Plot trace of nodes

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

Node Trace

Pick Node

Area In

Polygon Out

Key in Node IDs

Node Label

Line Width

Line Color

Starting State:

Write Curve Write Trace

Clear Trace Done



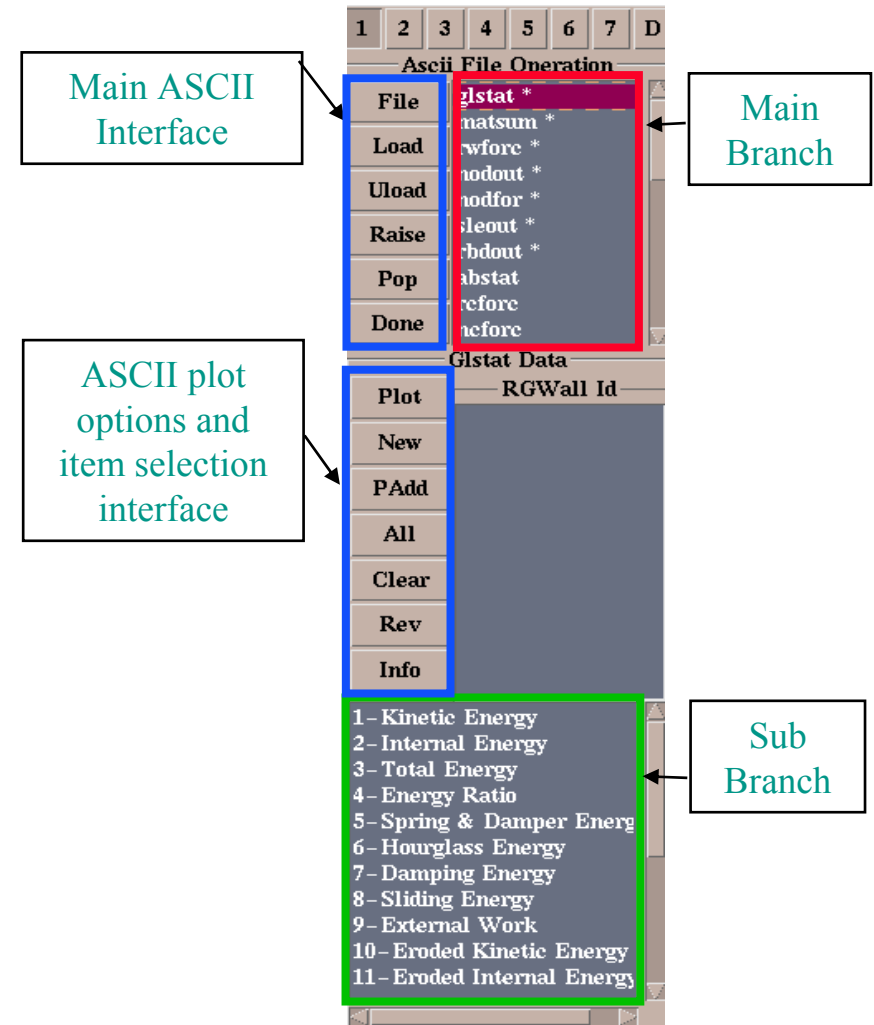
ASCII files

XY-plotting of various LS-DYNA ASCII files

Support for loading multiple ASCII files from different runs

❑ Main ASCII Interface

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





ASCII files

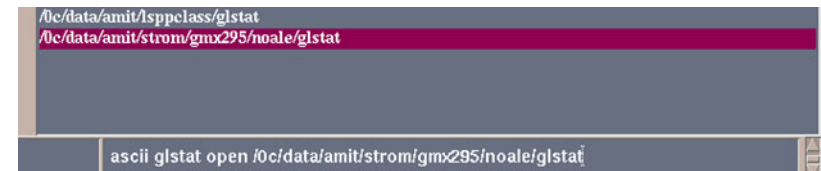
□ ASCII XY Plot Interface

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



□ Multiple ASCII files can be loaded from alternate subdirectories

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



Multiple ASCII file loading with full path of file shown

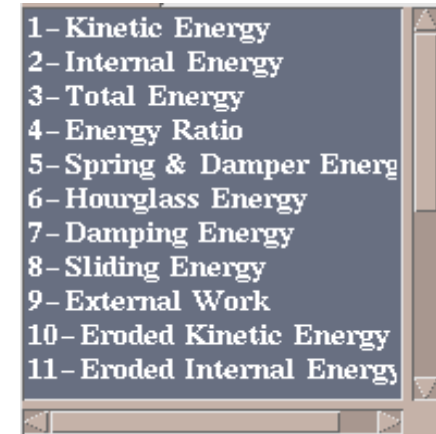




ASCII files

□ ASCII Interface SUB Branch

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



Component list of GLSTAT file

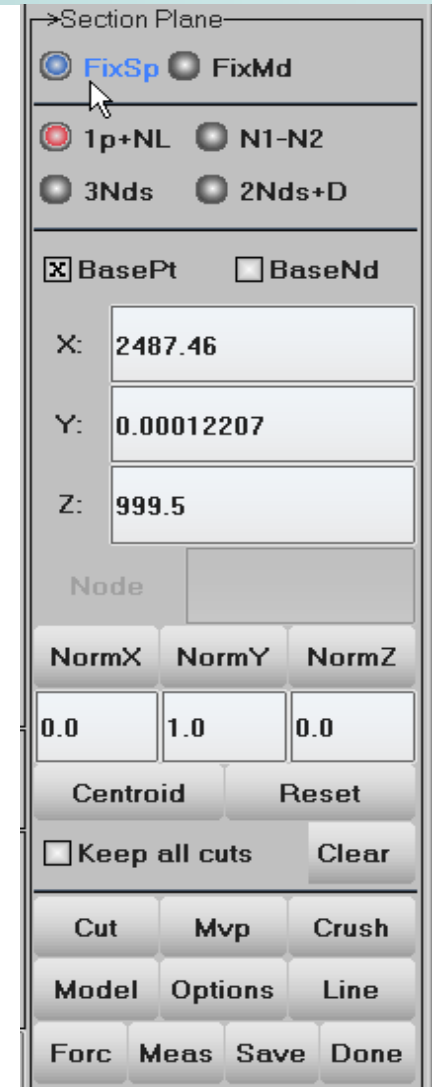


Section Plane

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

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

NormX	NormY	NormZ
0.0	1.0	0.0





Section Plane



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

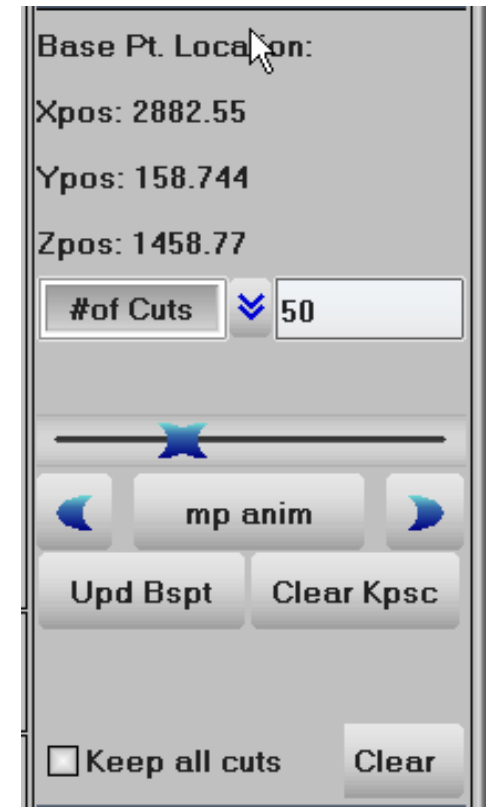




Section Plane

Mvp – Moving plane interface

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

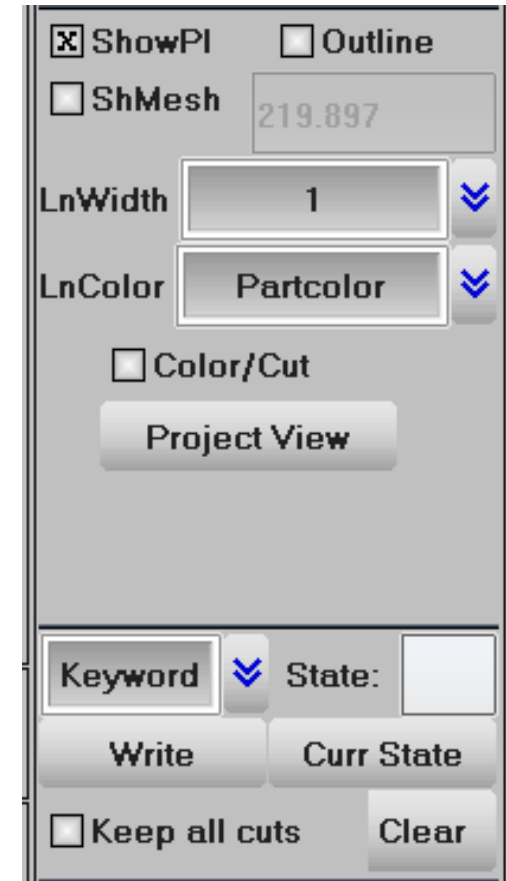




Section Plane

Options – Misc. options for section plane

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

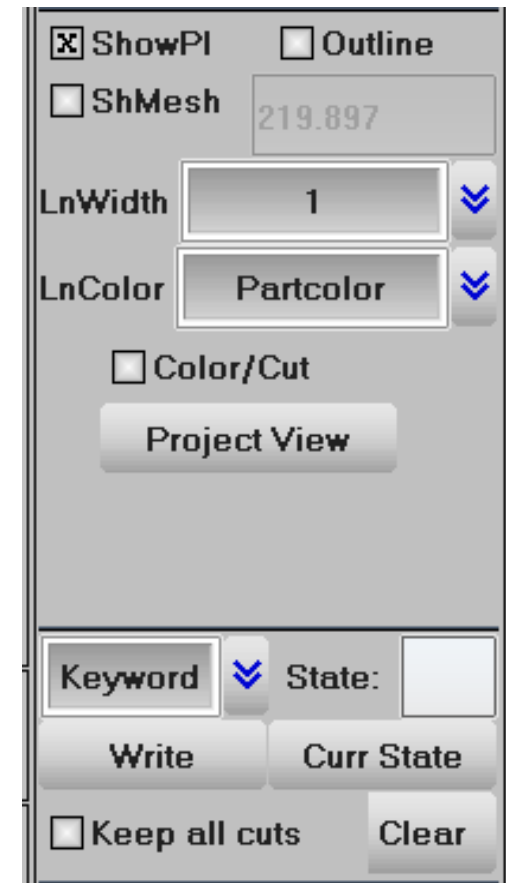




Section Plane

Options – Misc. options for section plane

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





Section Plane

Cursh – Intrusion plot – measure distance from node to plane

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

Selected PartID: 7
Coord at X = 0:
X= 849.613362
Y= 0.000123
Z= 808.303894
Coord at X = End:
X= 1625.957642
Y= 0.000122
Z= 990.140747

Plot New Padd
Flip Raise Pop

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

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

Intrusion Plotting:

Line Width 1

Line Color White

PlotType Crush

Node ID:

Plot New Padd
Clear Raise Pop





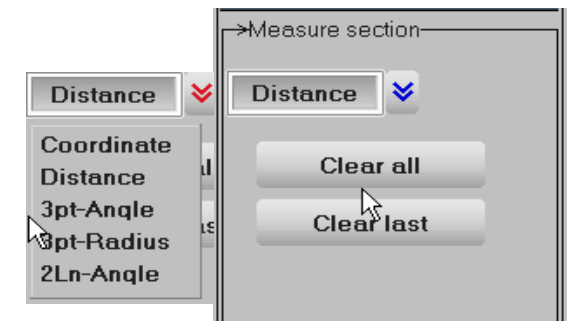
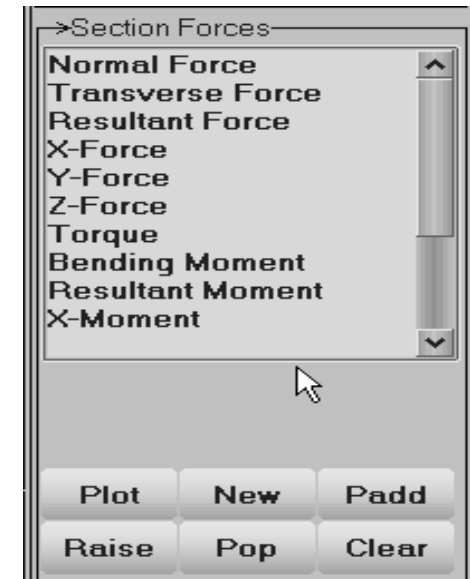
Section Plane

Force – Calculate and plot the section forces

- Force and moment components
- Area can also be plotted

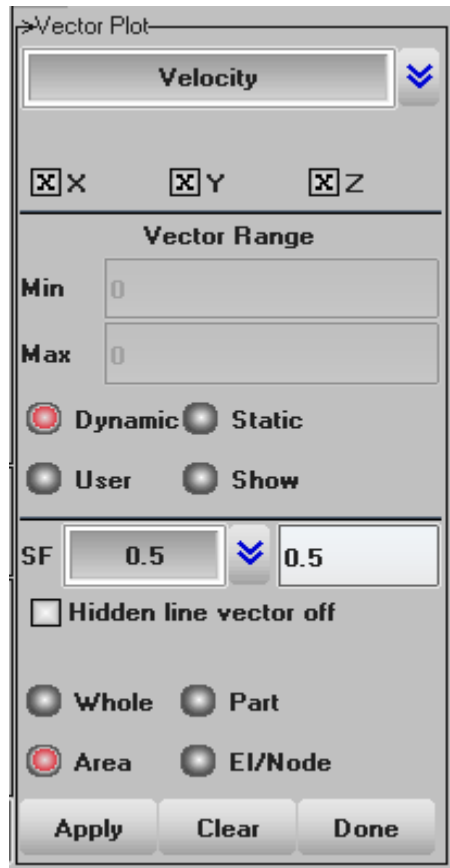
Measure – Section cut measure interface

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





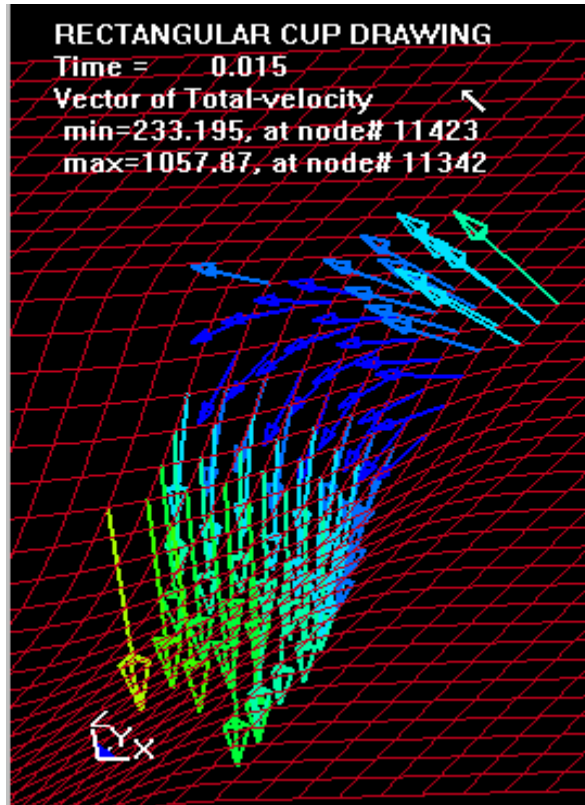
Vector Plot Menu



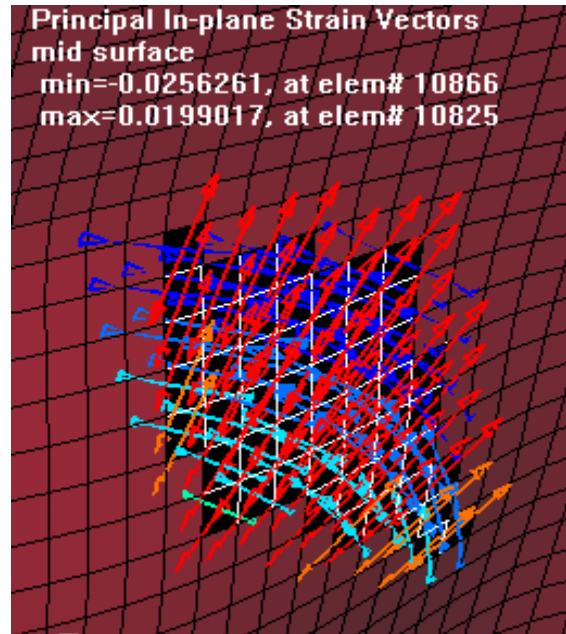
- Vector display – Shell Normal, Displacements, Velocities and Principal stress and strain directions for the d3plot file
- Additionally, Forces, Pressures and Surface shear for the Interface Force file.
- Components for displacement, velocity and force vectors can be selected
- Dynamic – sets range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



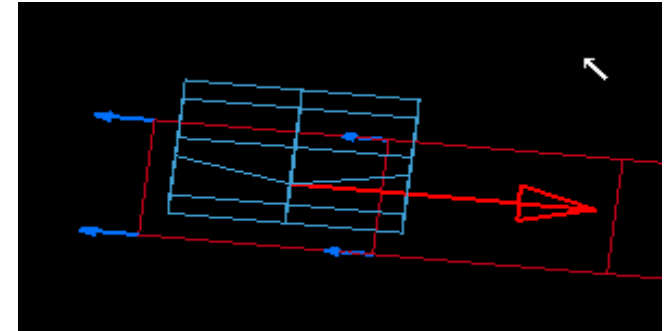
Vector Menu



Velocity



Principal Strain



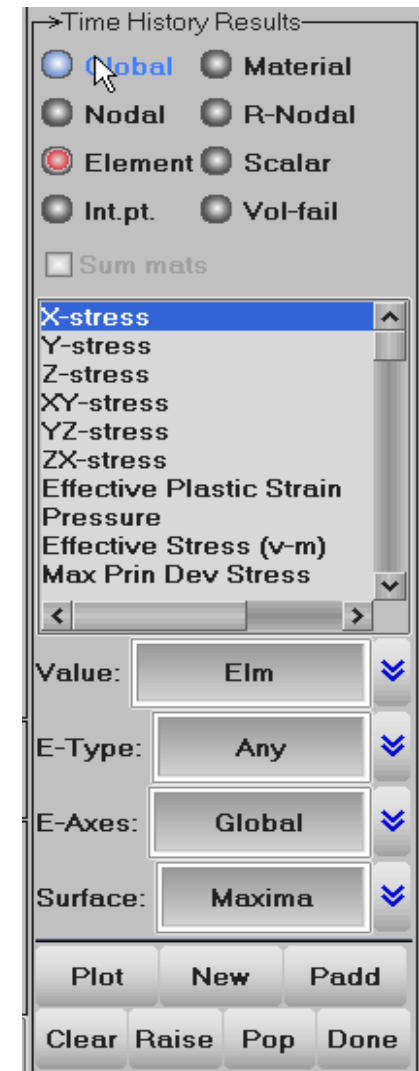
Nodal Surface Shear



Time History Interface

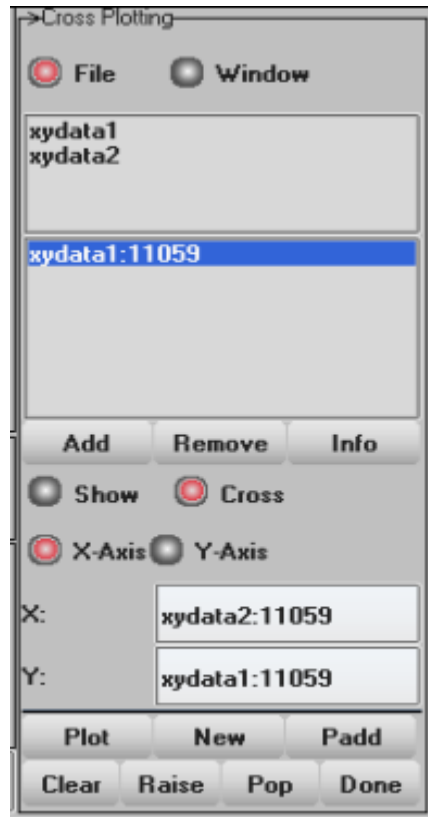
History – to plot time history data

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





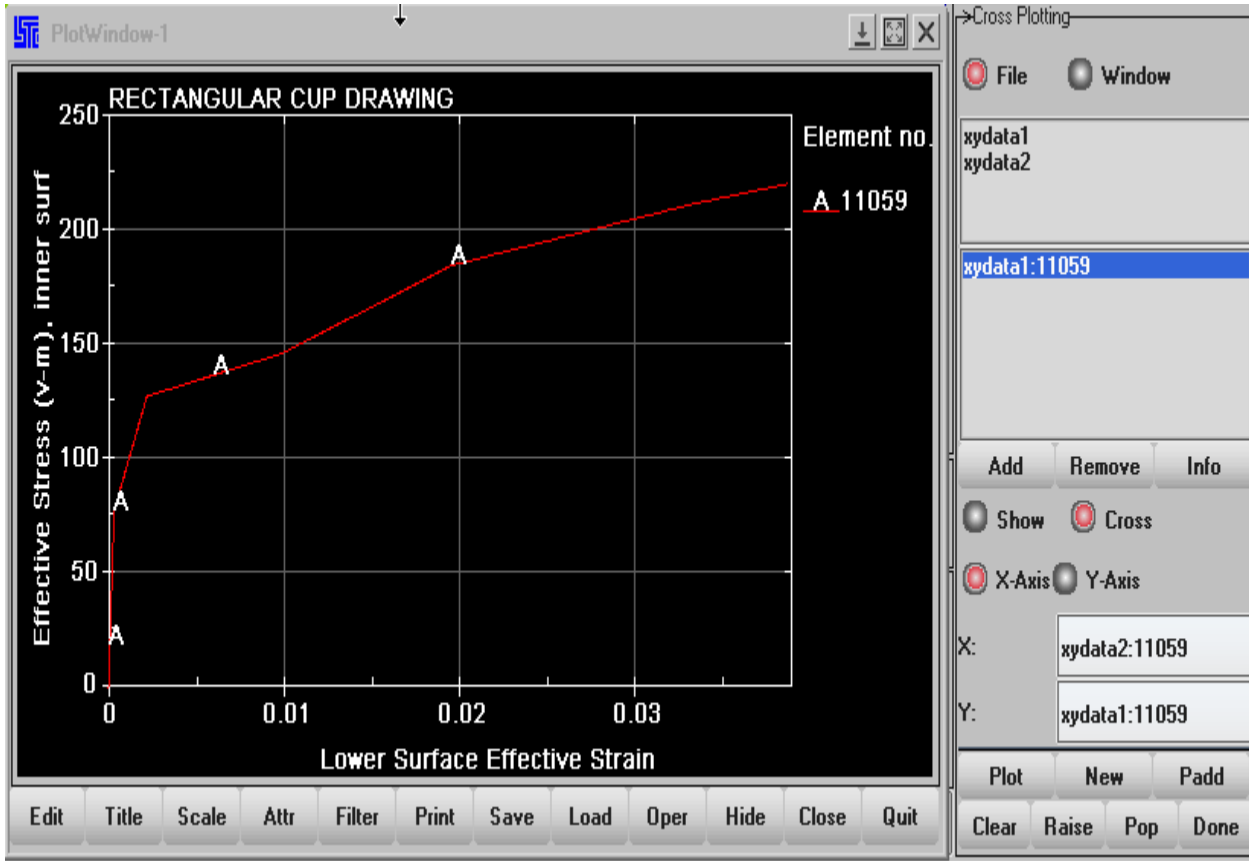
XYPlot – show and cross plotting



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

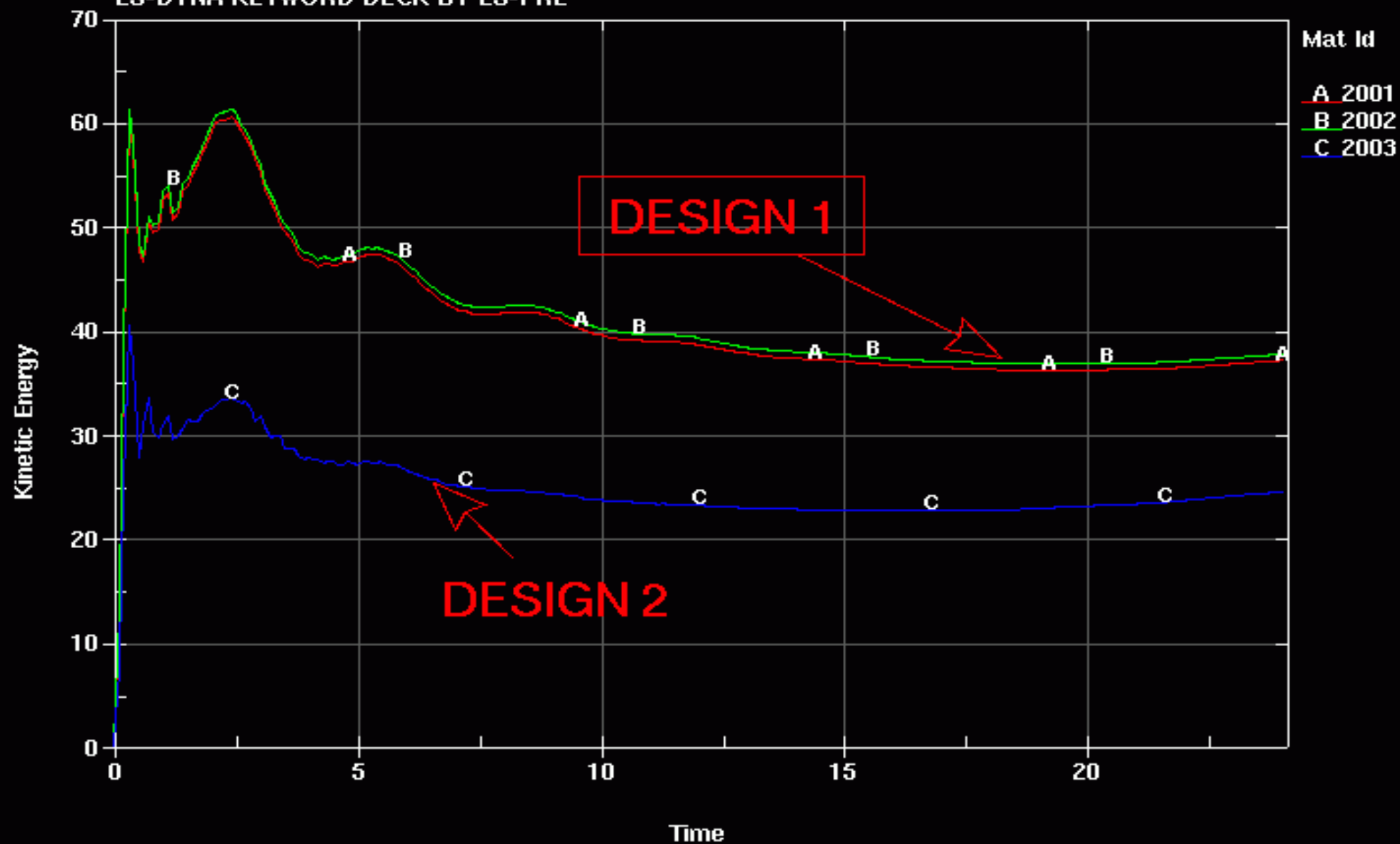


XY plot – cross plot



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

LS-DYNA KEYWORD DECK BY LS-PRE



Edit

Title

Scale

Attr

Filter

Print

Save

Load

Oper

Hide

Close

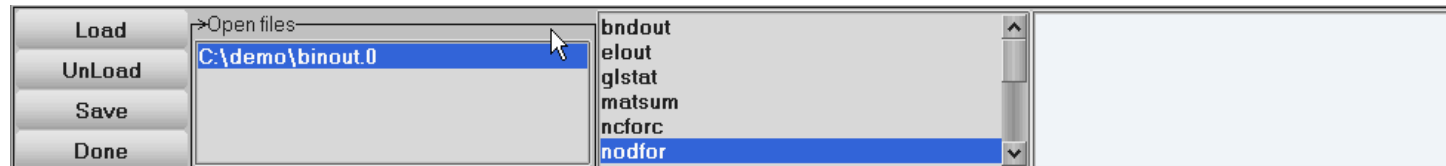
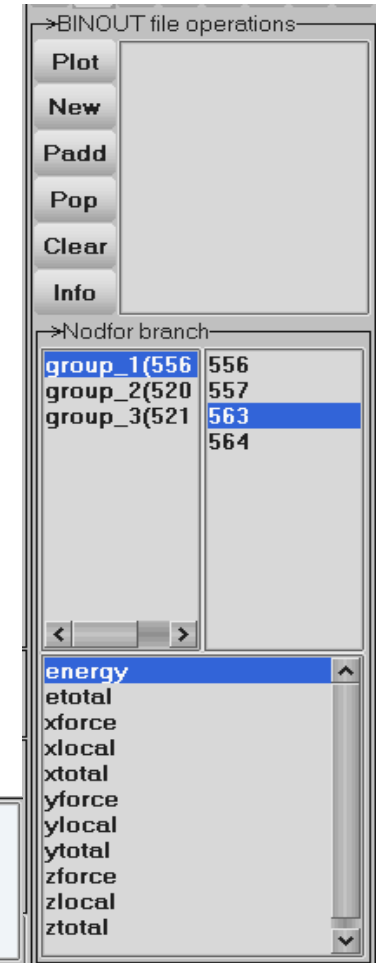
Quit



Binout

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

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

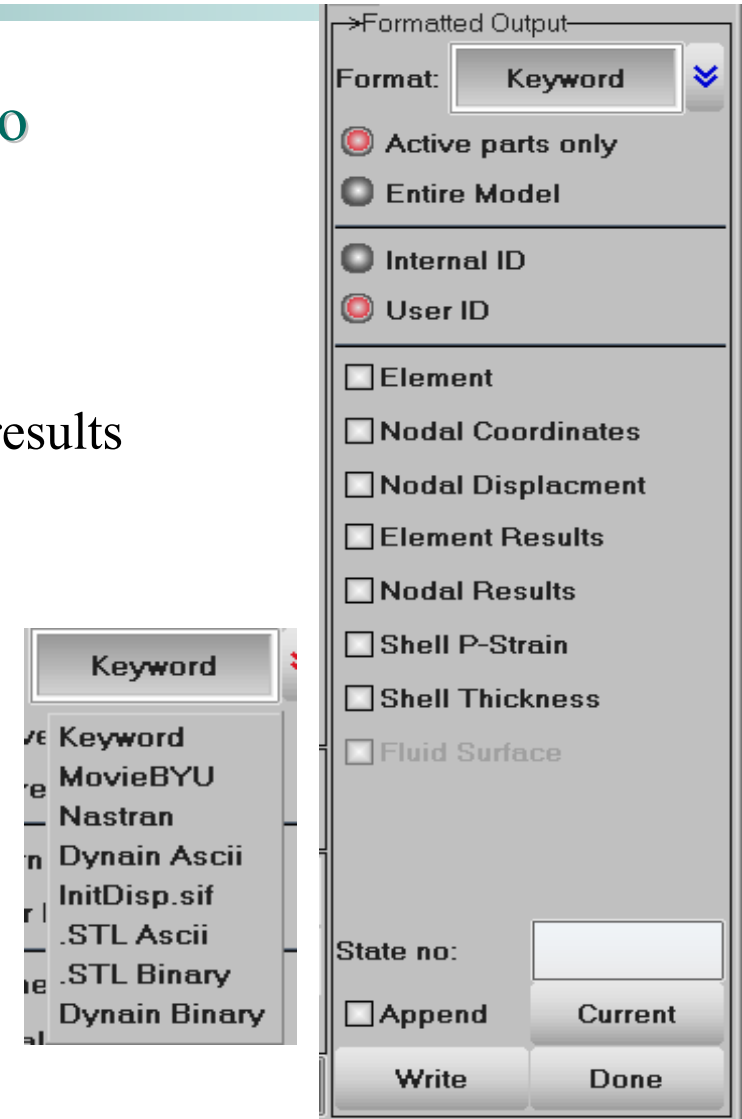




Formatted Output

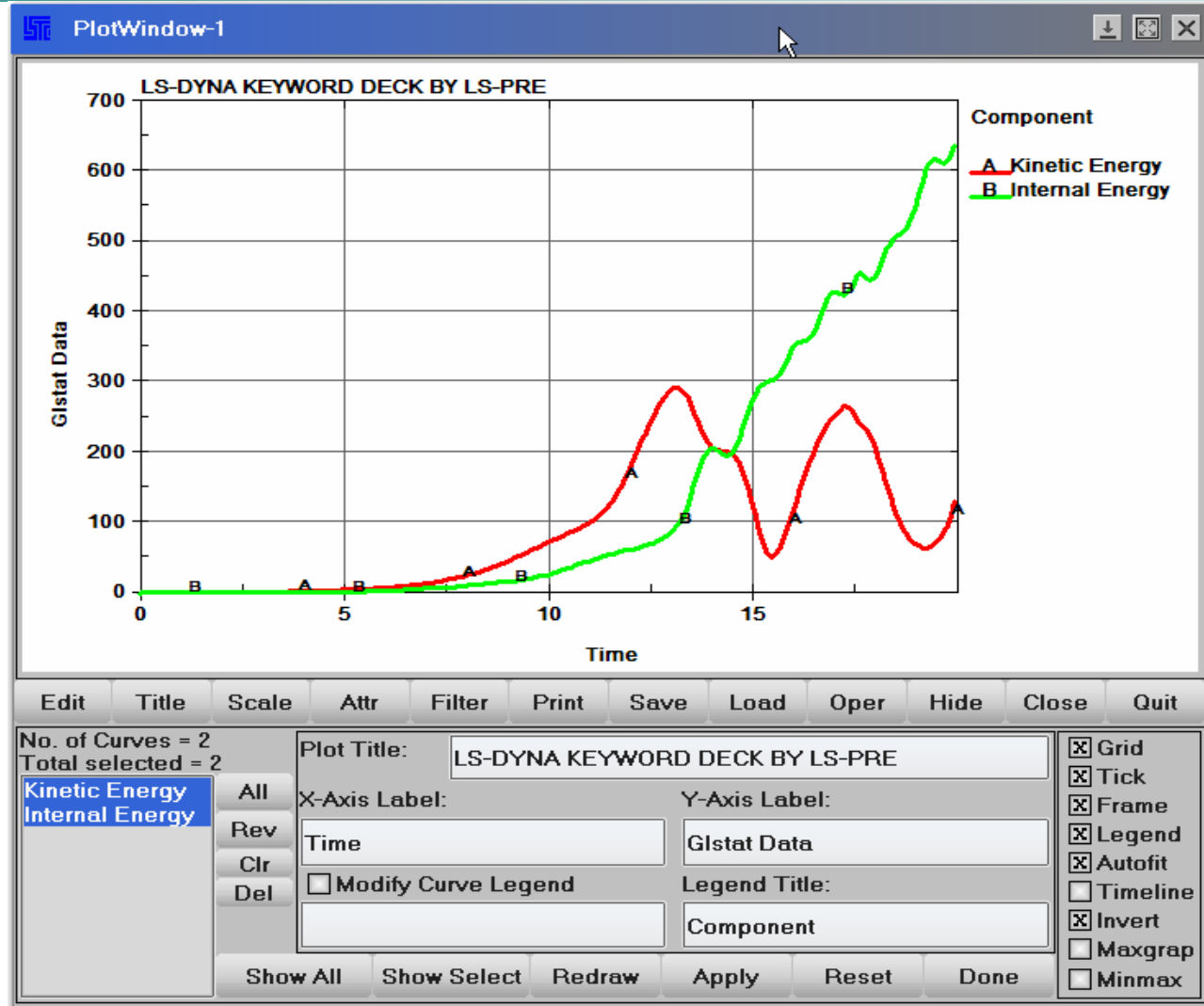
Purpose: To output post-processing data into various formatted file

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



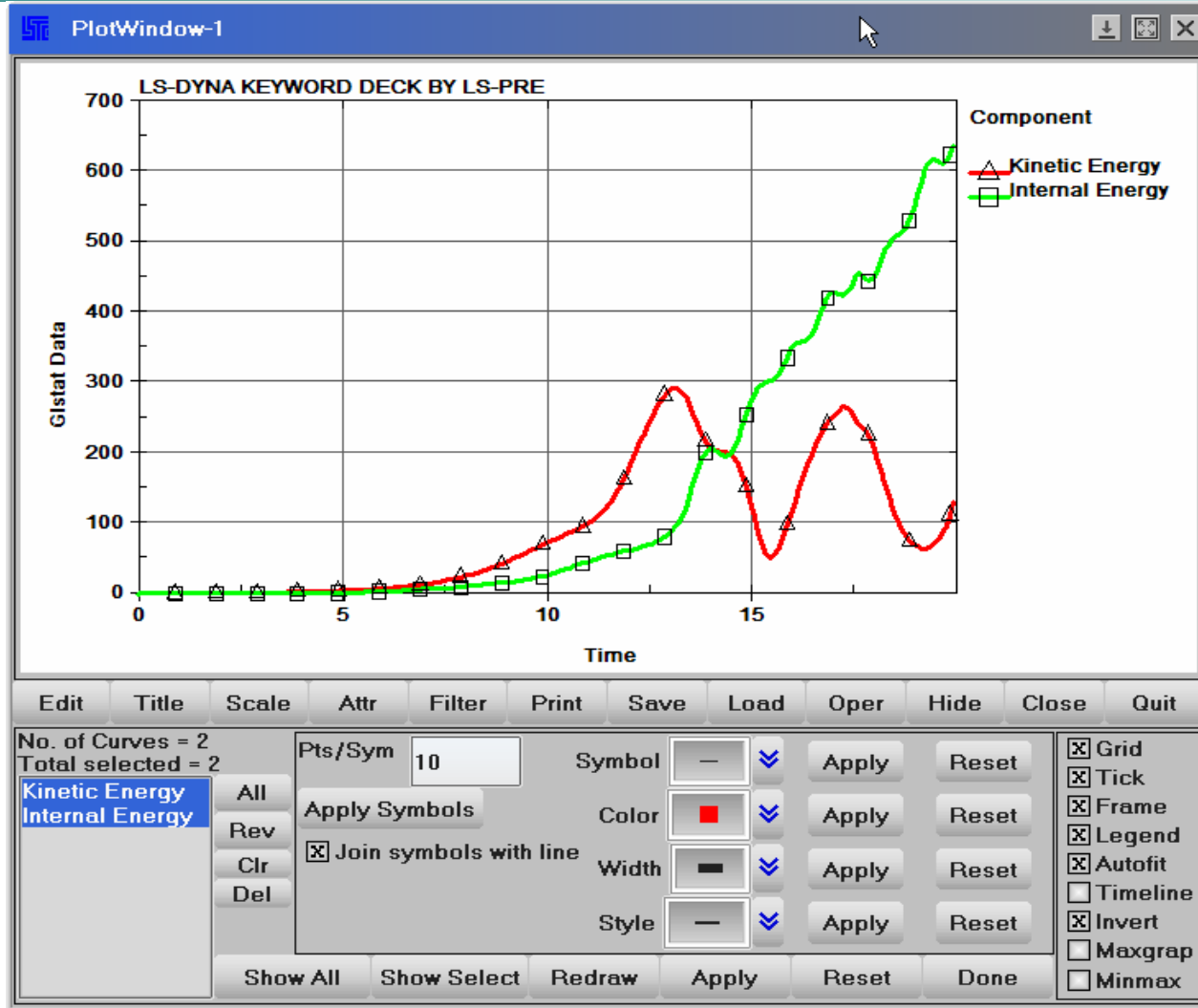


XYPlot Window





XYPLOT Window





Alemt – Processing ALE fluid

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

1 2 3 4 5 6 7 D

Fluid Volume

Pick fluid mesh

H 3
H 4

Tie to fluid

Inside Tie to fluid

p#1: box - fgid=1, inside

Rectangle Box

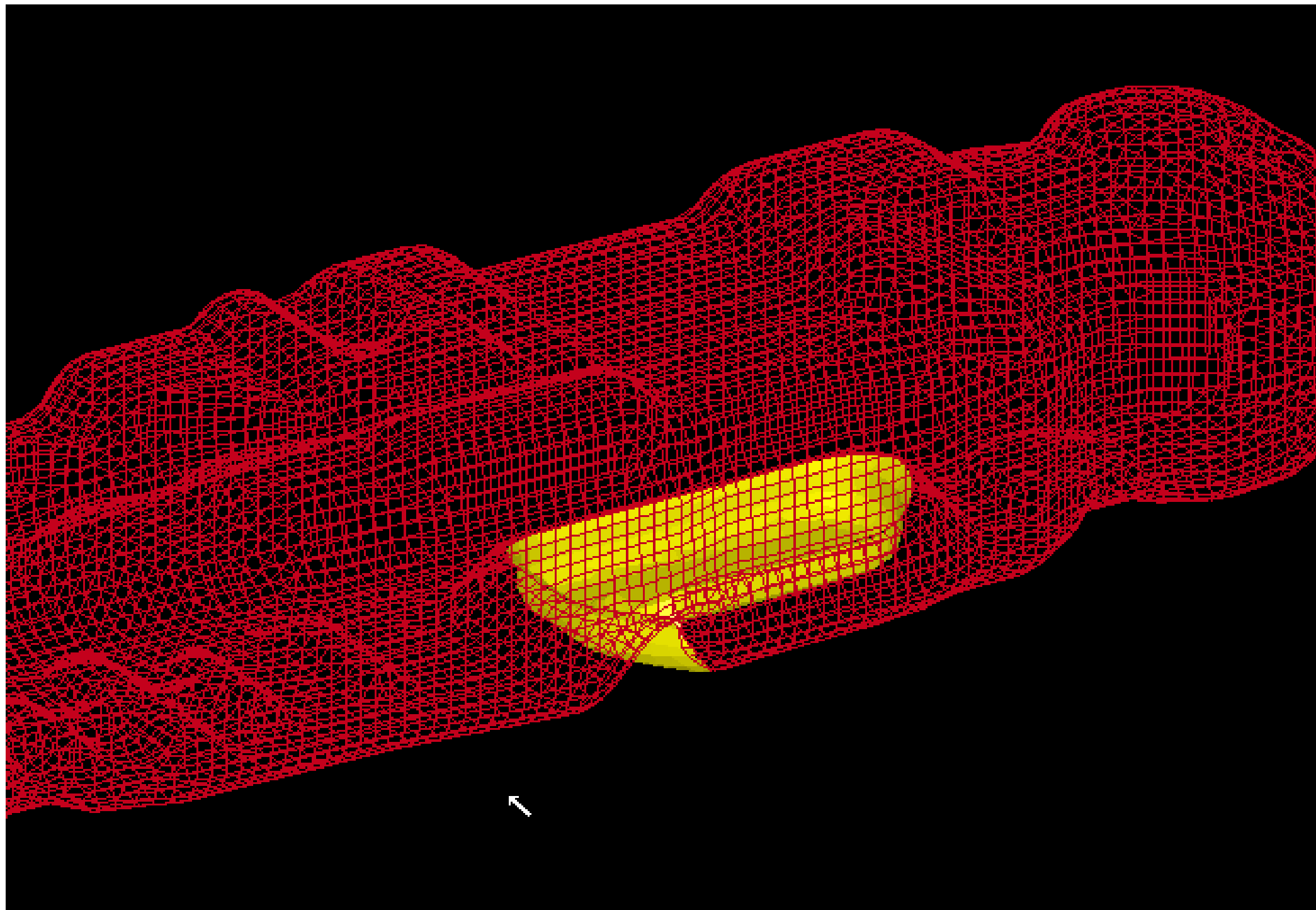
Min Max

X:

Y:

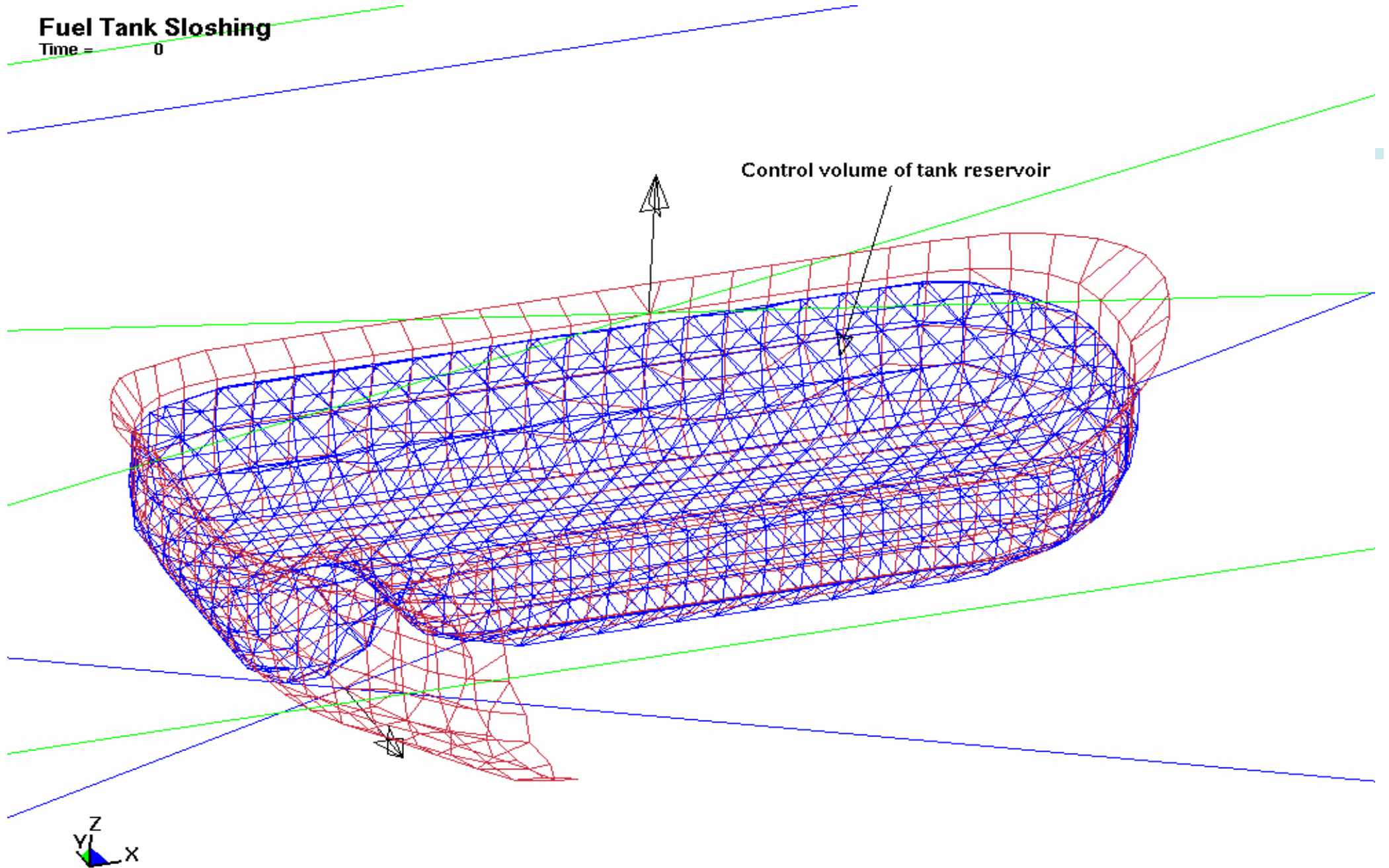
Z:

Show

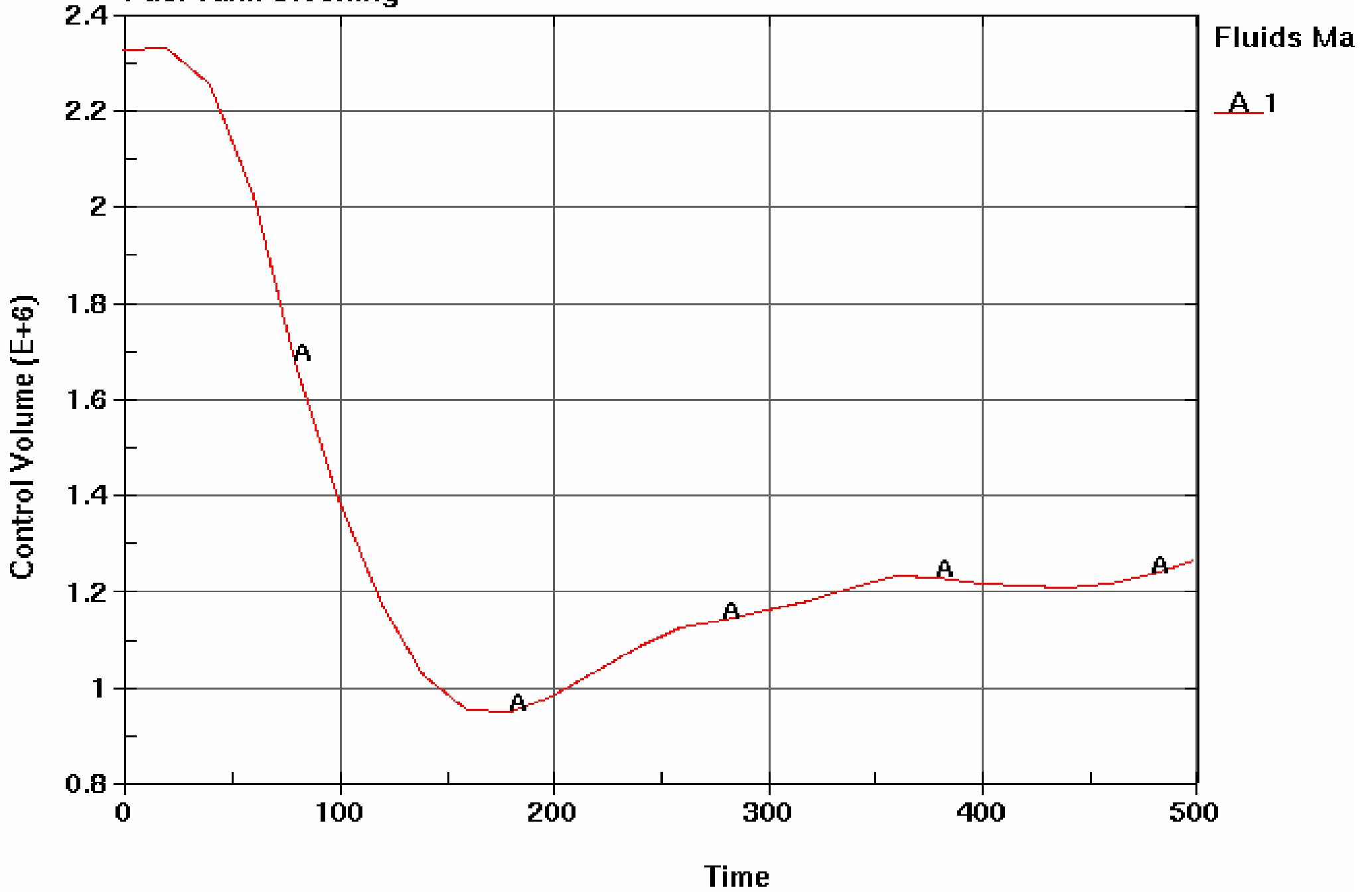


Fuel Tank Sloshing

Time = 0

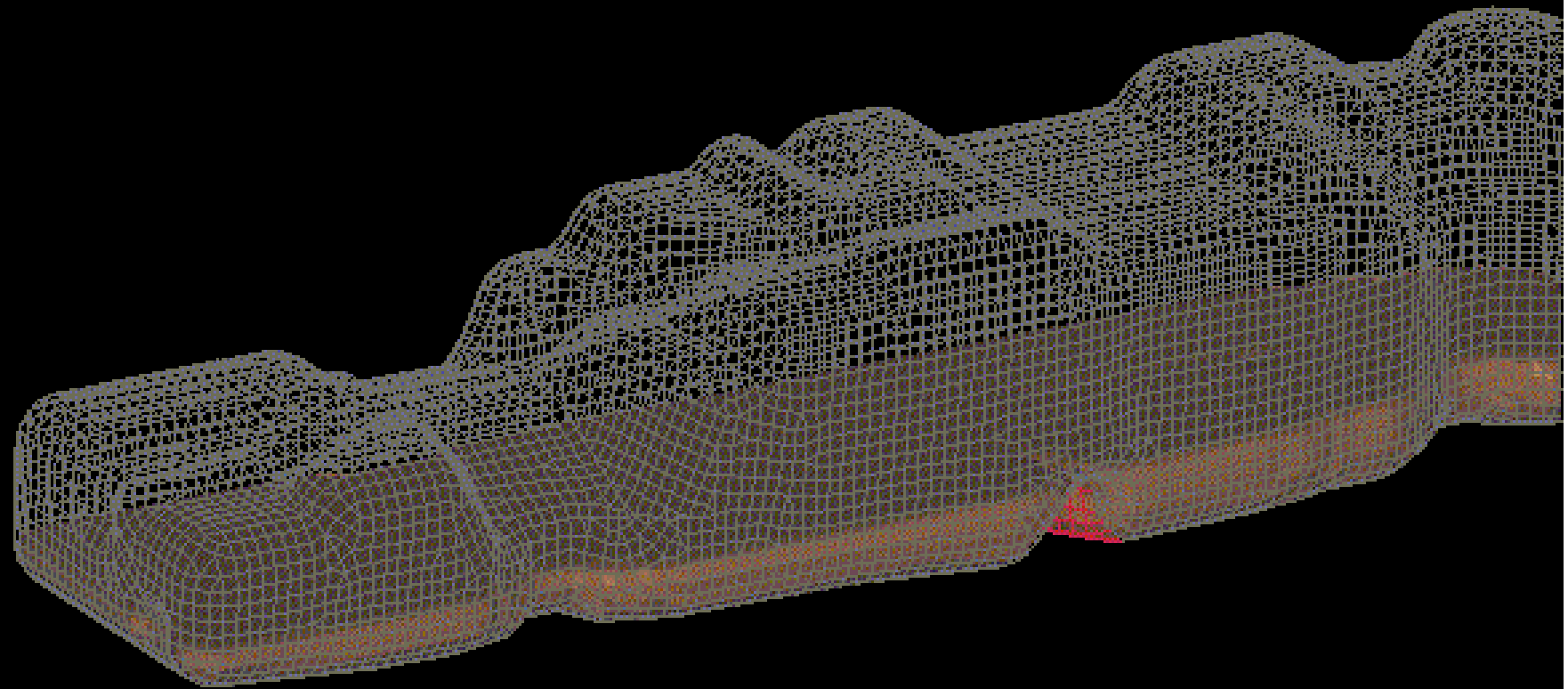


Fuel Tank Sloshing



Fuel Tank Sloshing

Time = 0





LS-Prepost_VR

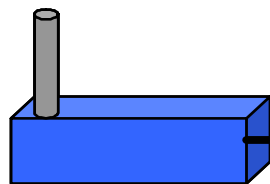
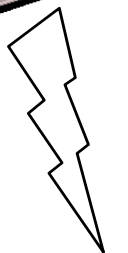
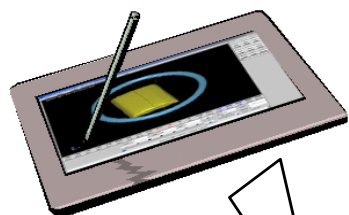
LS-Prepost_VR: A special version of Isprepost to run on large screen multi-wall system

- ❑ It is built on VR Juggler (<http://www.vrjuggler.org>), an open source framework for build virtual reality applications
- ❑ Wand-based interaction – LS-Prepost_VR supports interfaction via a 6-dof wand. The wand controls the model manipulation (rotate, translate, scale), animation and selection, also an interactive clipping plane to explore the dataset
- ❑ LS-Prepost_VR is free just like LS-Prepost, but requires installation Fee. (cost is systems dependent, contact LSTC for the cost)



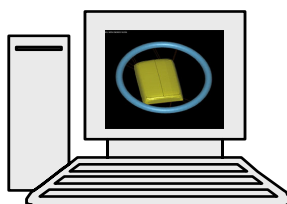
LS-Prepost_VR

LS-PrePost-Remote
(client)



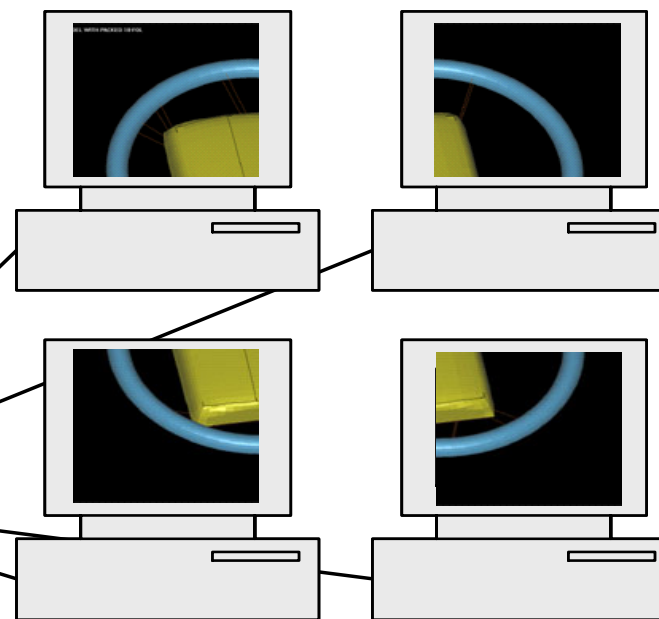
Wireless
Access
Point

LS-PrePost-VR
(server)



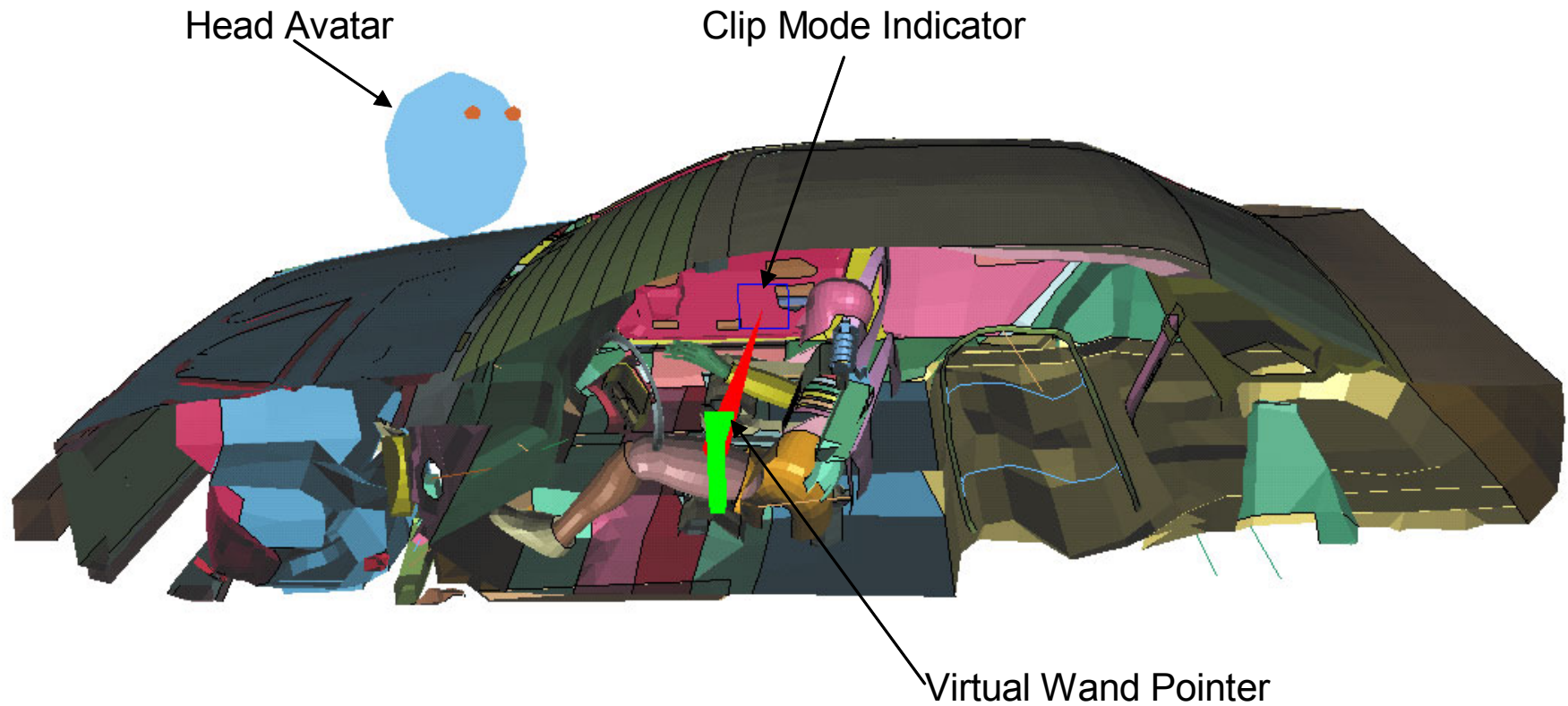
10/100
Network
Switch

LS-PrePost-VR
Render Nodes





LS-Prepost_VR





Pre-Processing

- Meshing
- Mesh data manipulation
- Model data manipulation
- LS-DYNA data creation and modification
- LS-DYNA keyword data import, export, creation and modification
- Other applications



Meshing

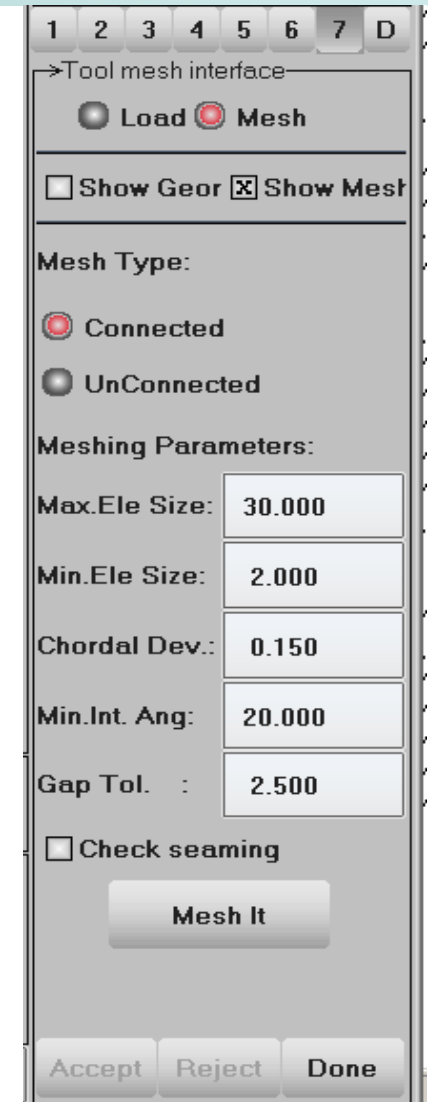
- ❑ Surface automatic shell element meshing
 - ❖ Tool mesher for metal stamping application
 - ❖ Topology mesher for general applications
- ❑ 2d Maze Magic meshing
- ❑ 3D simple geometry meshing

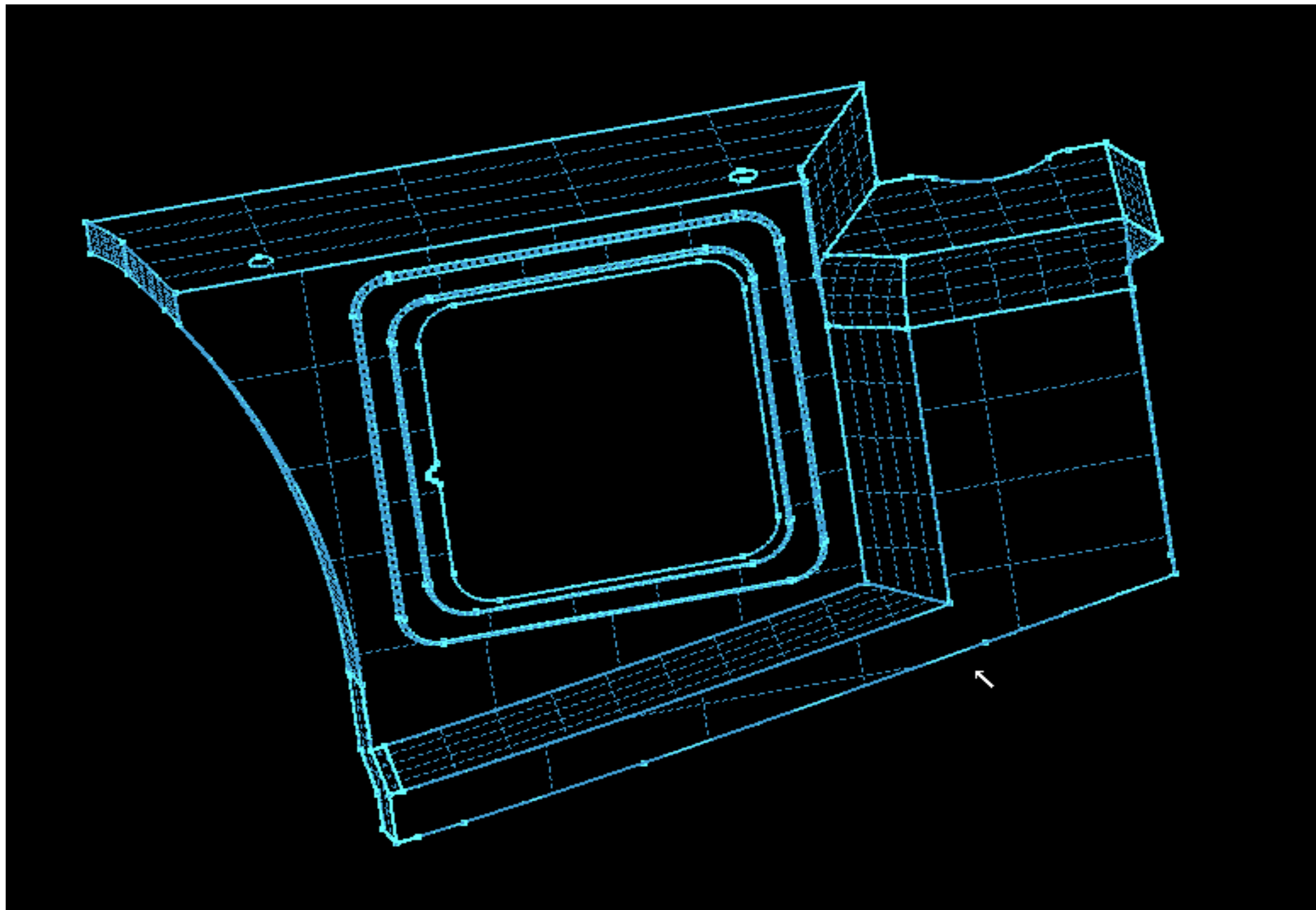


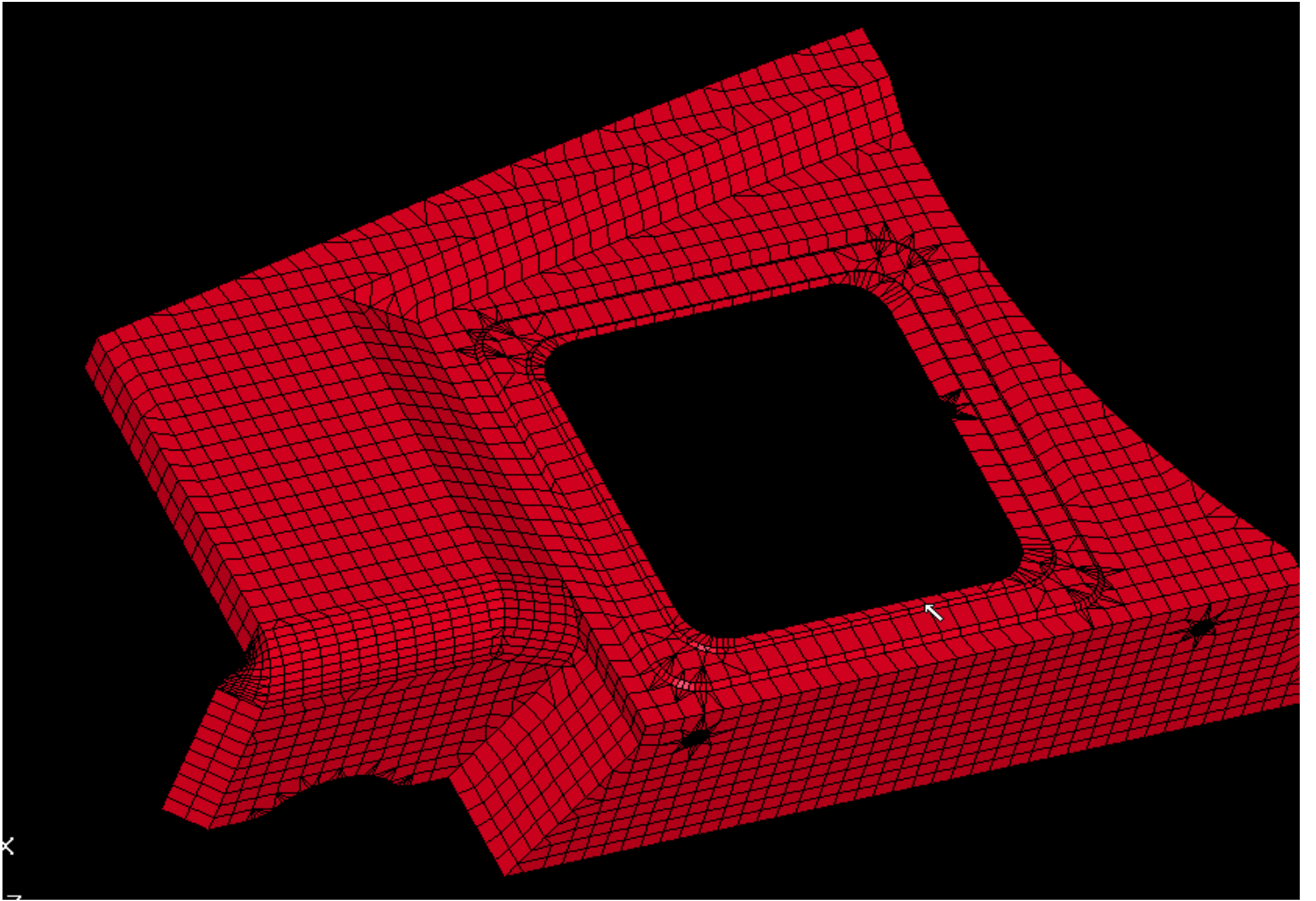
Meshing – TMesh and AMesh

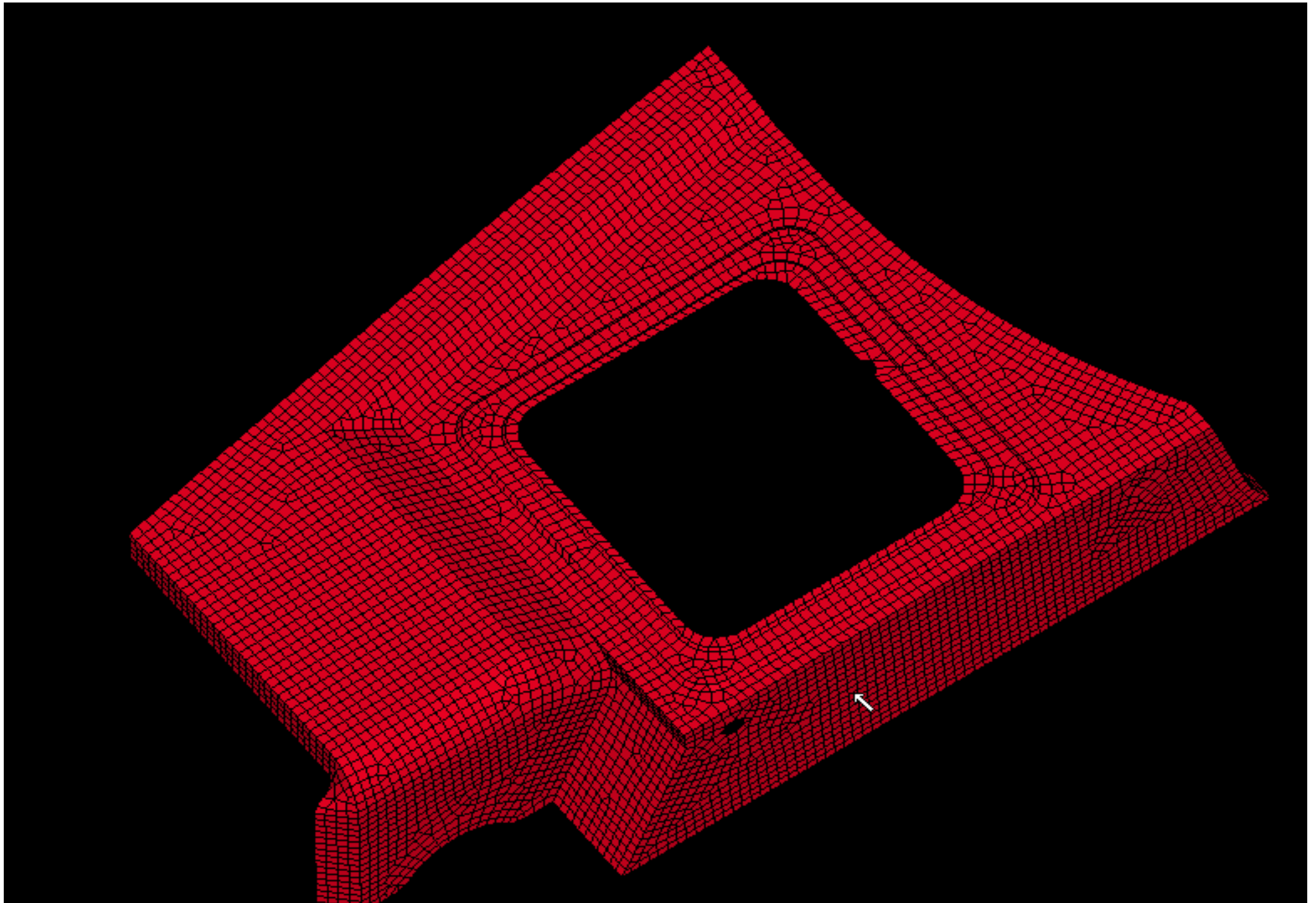
LSTC licensed ETA tool mesher and Topology mesher to incorporate into Isprepost

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







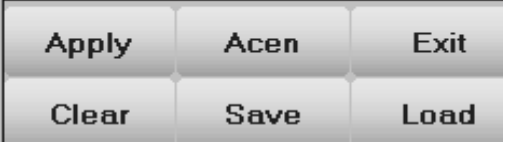
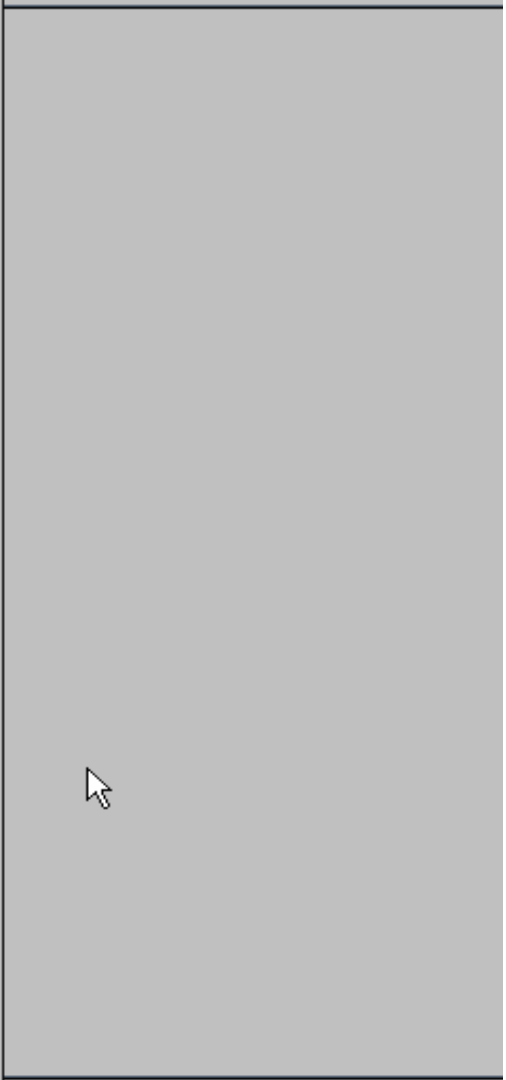
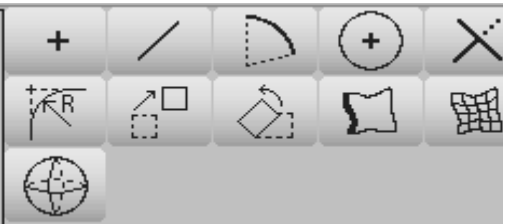
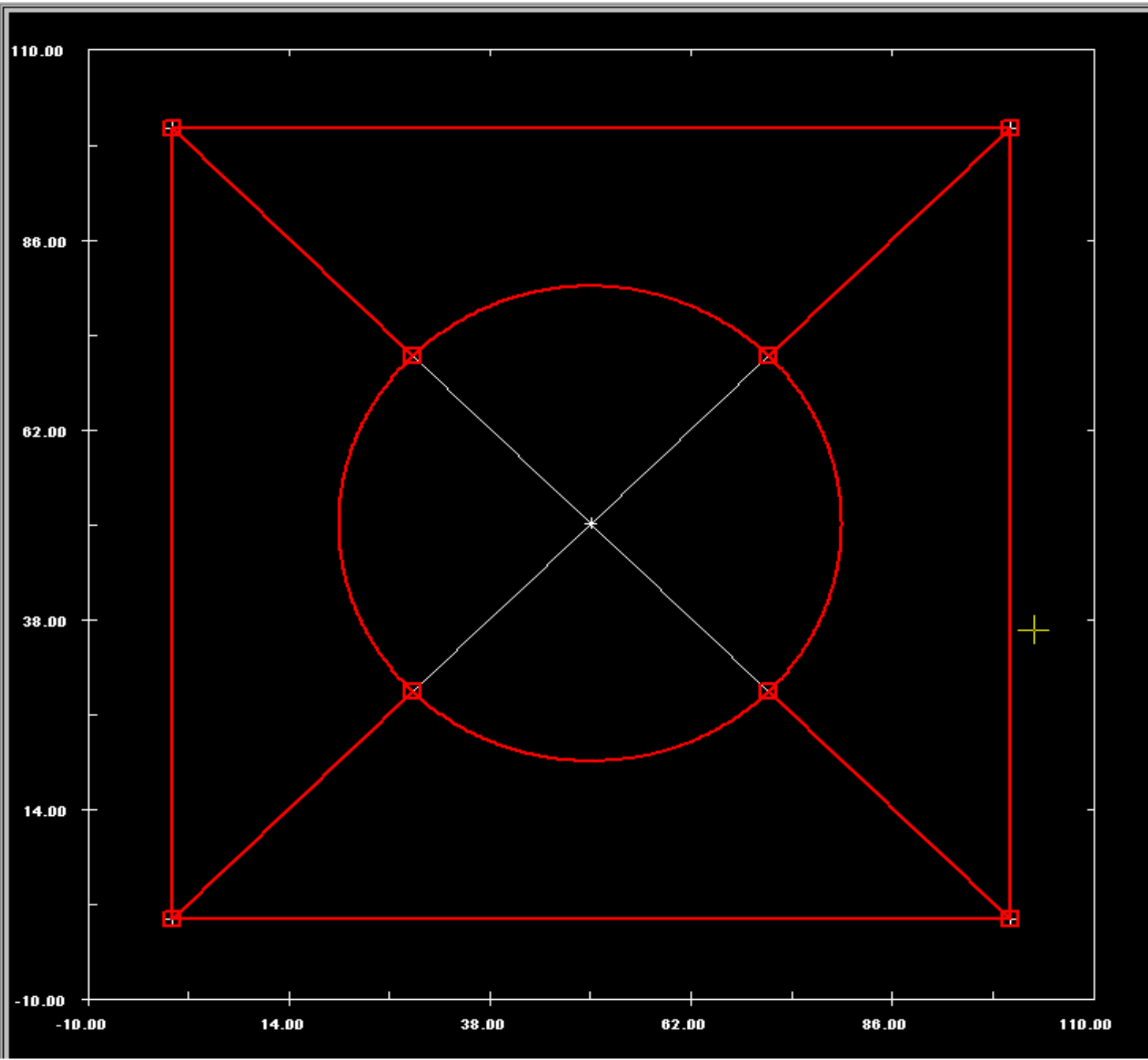


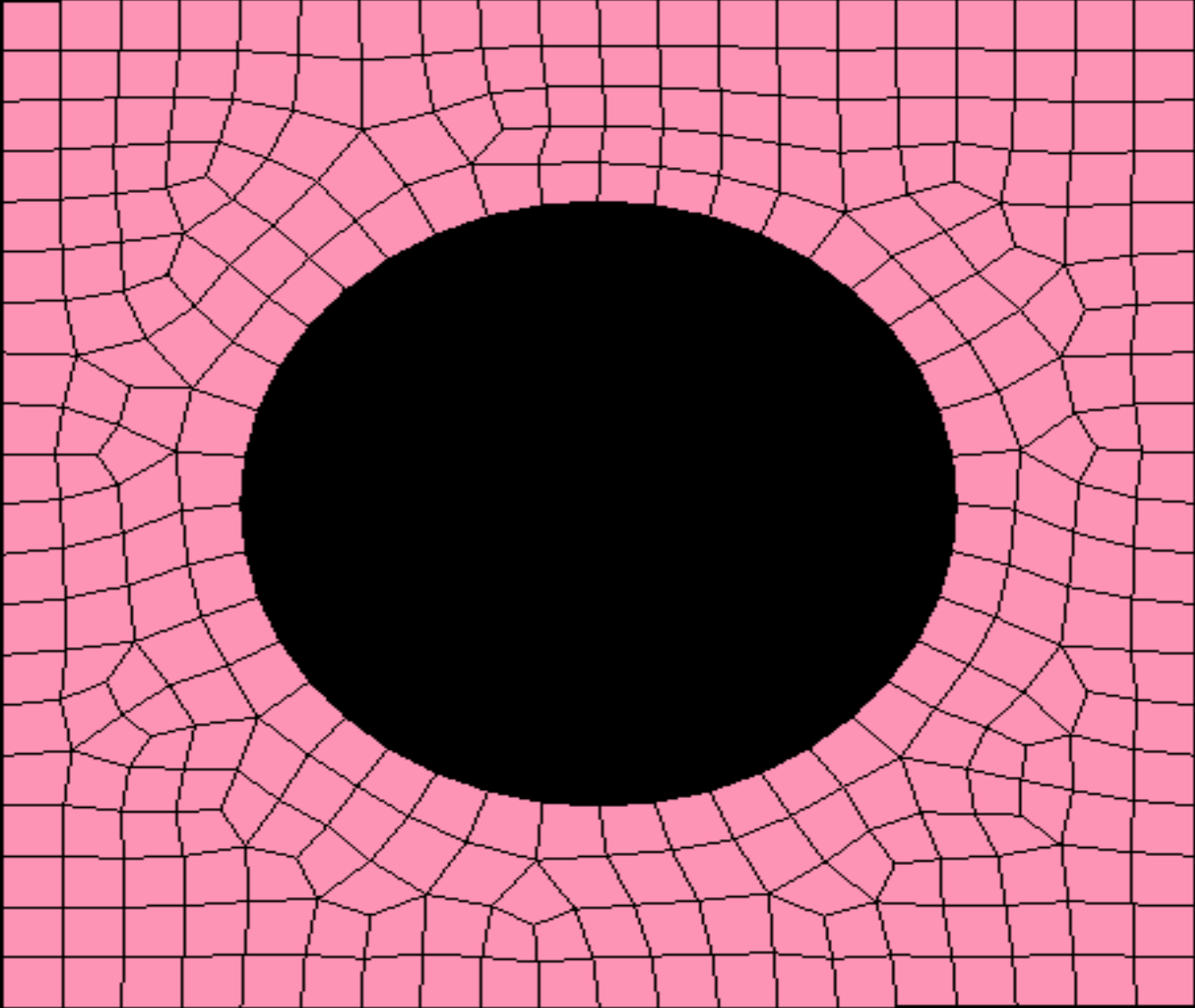


Meshing – 2d Mesher with sketch board

2dmesh – is the magic mesher from Maze

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



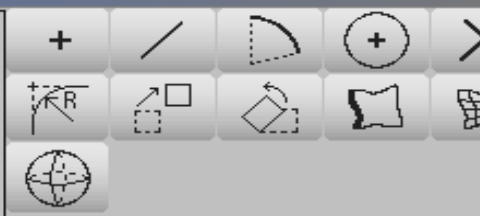
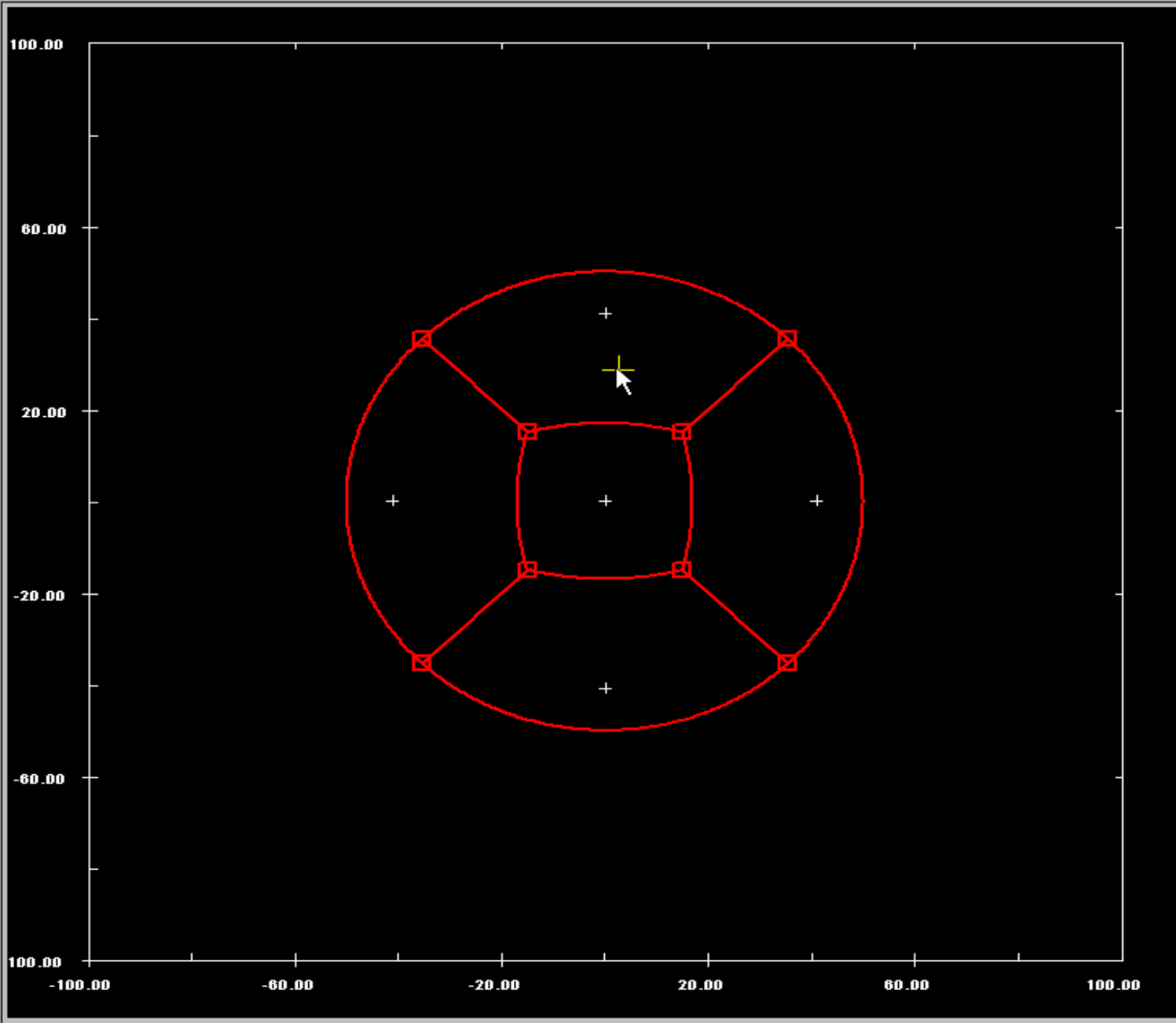




Meshing – Simple geometry mesh creation

Purpose: To create simple geometry, Box, Plate , Sphere, Cylinder/Cone

- Create solid box by giving box value (min/max values)
- Create solid Sphere by a point and radius, and meshing density
- Create solid cylinder/cone by giving 2 radii, length of cylinder, and meshing density
- Create Shell by give four positions.
- Create Cylinder shell by give Radius, Length and direction.



3D primitive

Cone Sphere

x: 2.515723

y: 28.725039

radius: 50.00

width: 30.0

Nelem X: 8

Nelem Y: 8

Nelem Z: 8

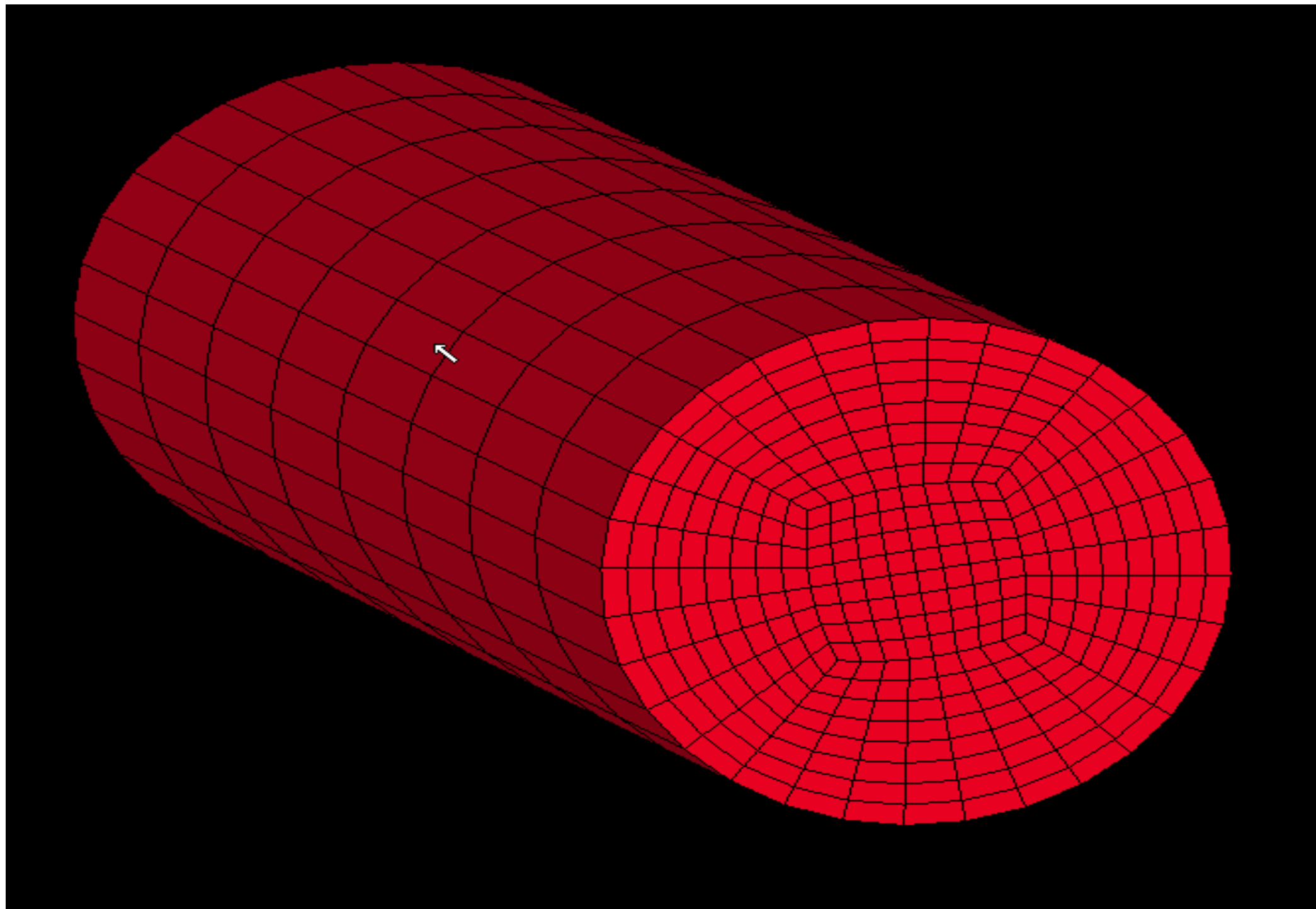
Nelem X-Y: 8

r2: 50.000000

L: 400

Create







Mesh data Manipulation

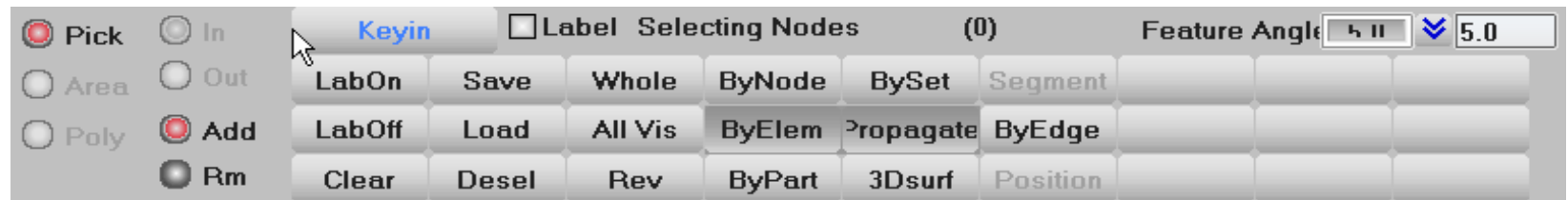
- General selection
- Rotate, Translate, Offset, Scale, Transform, Reflect mesh data
- Move/copy elements from 1 part to a different part
- Mesh repair, check, and edit
- Element generation from other elements
- Node creation, edit



General Selection

Many interfaces use General Selection to select nodes/elements/parts/segments

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





Translate Mesh

Translate a group of nodes in a specified direction

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





Translate Mesh - Copy Option

Nodes or Element can be created with CpEl or CpNd options

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

CpEl CpNd Tran

No. of Copies: 1

PickPart

Starting EID: 1008136

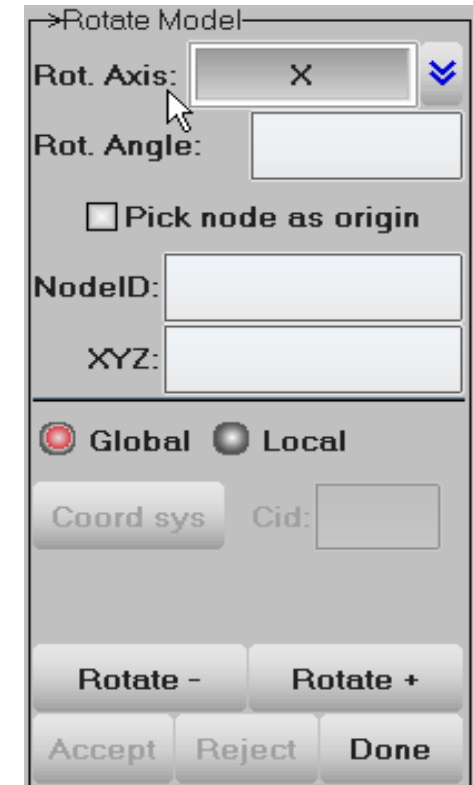
Starting NID: 1007117



Rotate Mesh

Rotate the selected mesh about an axis

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





Scale Mesh

Scale the selected mesh

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

Scale Model

Scale Dir: Global X

Scale Factor:

Pick node as origin

NodeID:

XYZ:

Node1

Node2

Node3

Scale Down Scale Up

Accept Reject Done



Shell offset

Offset shell mesh in the direction of its normal

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

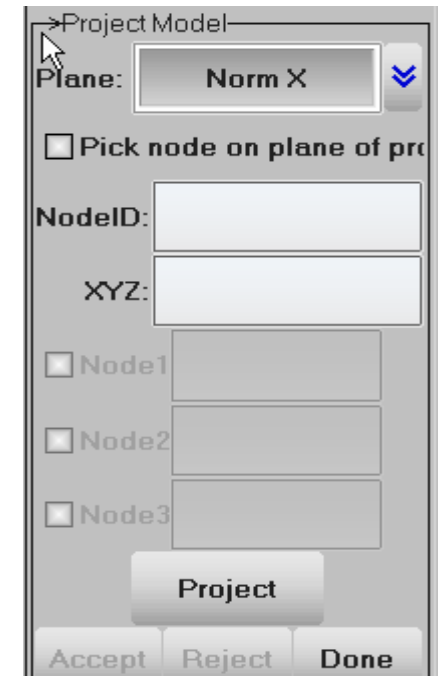




Project Mesh

Project mesh onto a plane

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





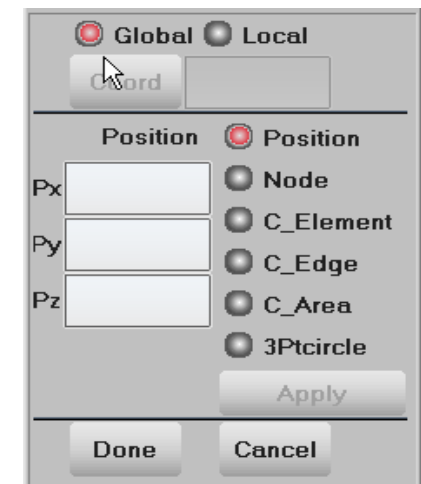
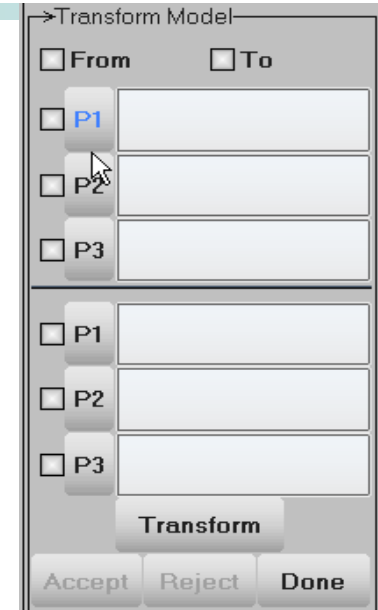
Transform Mesh

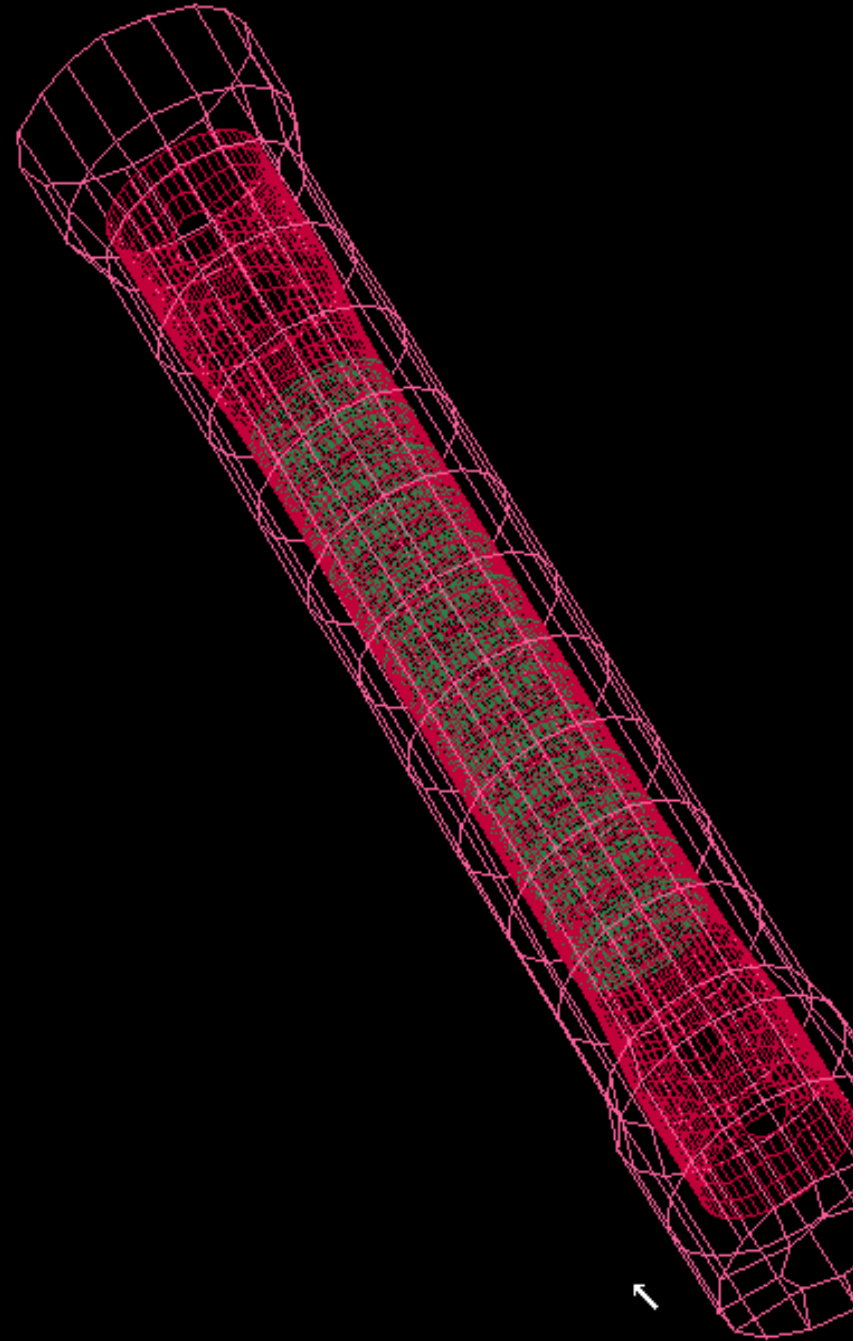
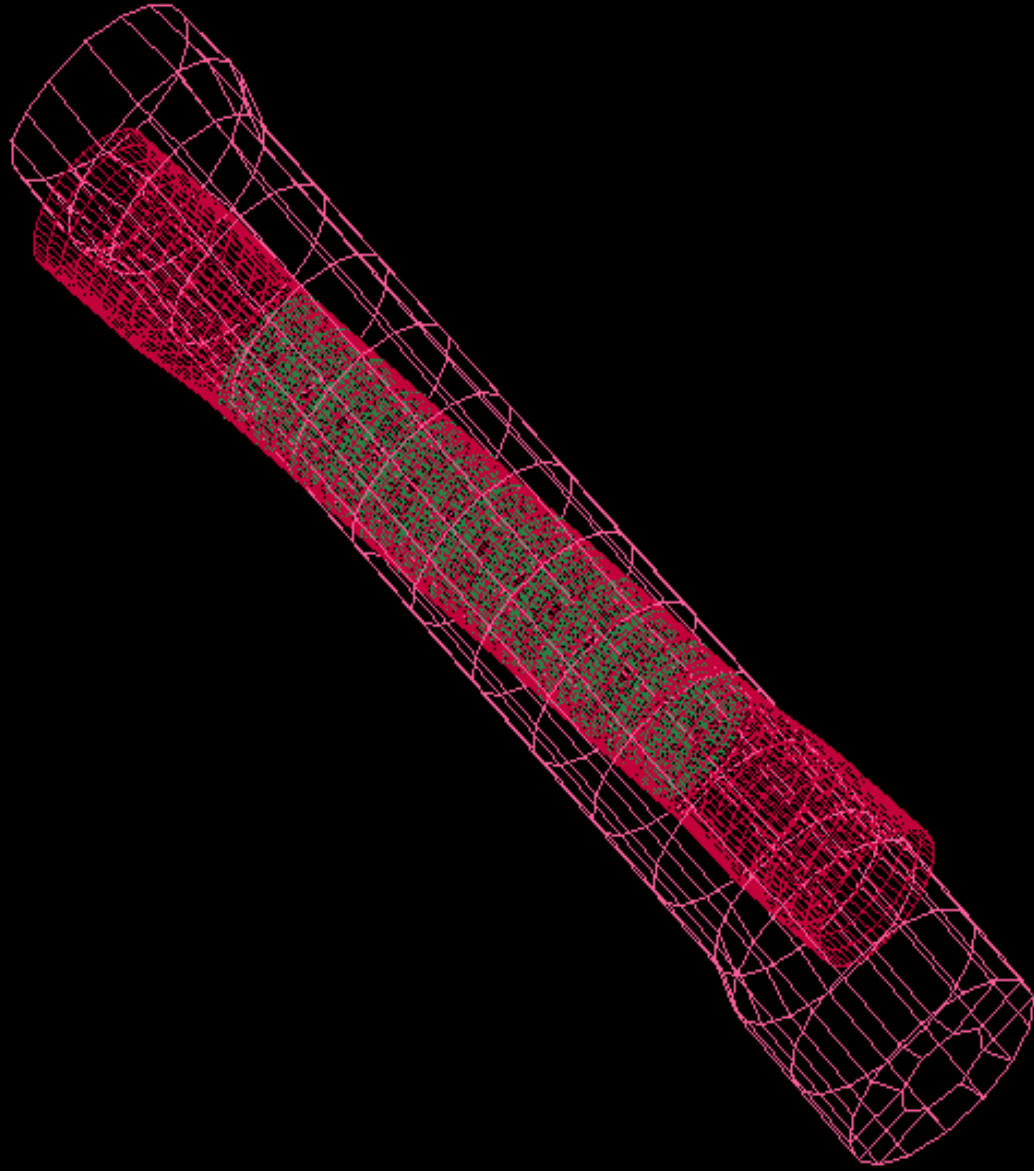
Transform mesh from one coordinate system to another coordinate system

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

Popup position interface:

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



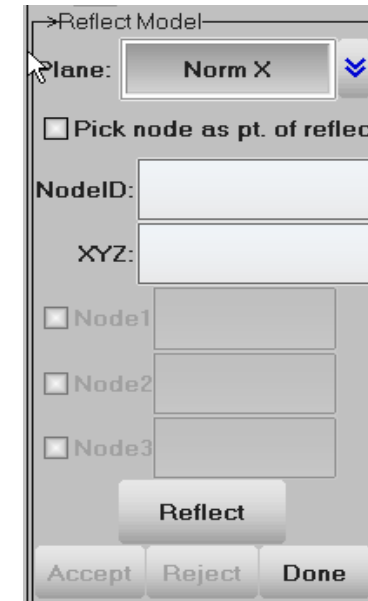




Reflect Mesh

Reflect mesh about a symmetry plane

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

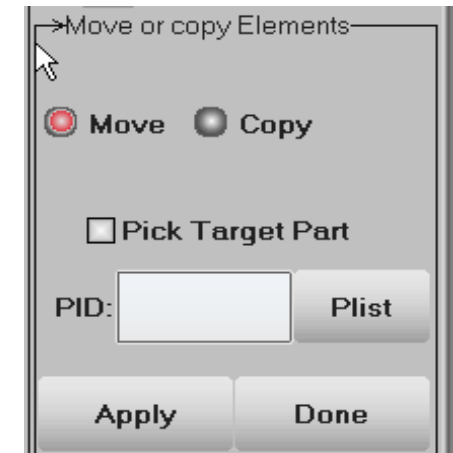




Move and Copy

Move or copy elements from one part to another part

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





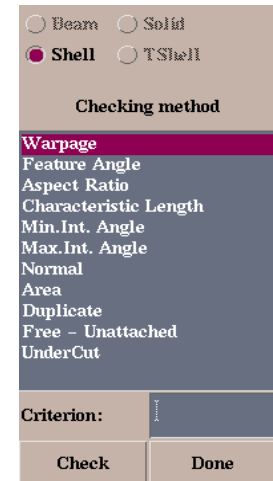
Element Edit

Node and Element creation and editing interface

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



ELEDIT interface



Element quality check





Node Edit - Node modification and creation

<input type="checkbox"/> NodePick	X	<input type="text"/>	Node Id: <input type="text" value="6182"/> NewID	<input checked="" type="radio"/> ByOne	Undo
<input type="checkbox"/> PtonElem	Y	<input type="text"/>		<input type="radio"/> ByTwo	Create
Position	Z	<input type="text"/>		<input type="checkbox"/> Translation	Reject
					Accept

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

Global Local

Coord

Position Position

Px Node

Py C_Element

Pz C_Edge

C_Area

3Ptcircle

Apply

Done Cancel

Positioning nodes

<input checked="" type="checkbox"/> NodePick	<input checked="" type="checkbox"/> PT1	<input type="text"/>	Begin	End	Node Id: <input type="text" value="6182"/> NewID	<input type="radio"/> ByOne	Undo
<input type="checkbox"/> PtonElem	<input type="checkbox"/> PT2	<input type="text"/>				X	<input type="text"/>
<input type="checkbox"/> TwoNode	Num 1	<input type="text"/>	Y	<input type="text"/>	Translation	Reject	Accept
		<input type="text"/>	Z	<input type="text"/>			

Create nodes between two points

Node PtOnElem

Point1 Point2

X

Y

Z

Clear

Align

Reject

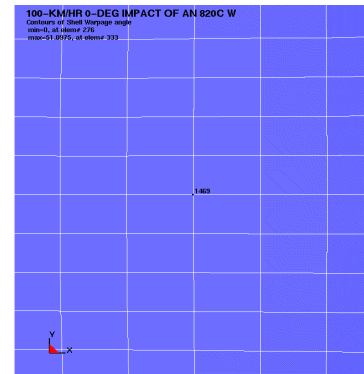
Accept

Done

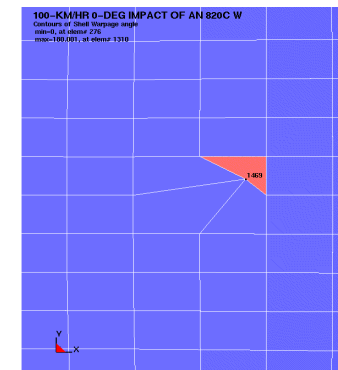
Align nodes between two points

Pick Node | Position |
| X |
| Y |
| Z |
| Quality Check |
| Reject |
| Accept |
| Done |

Modify position of nodes



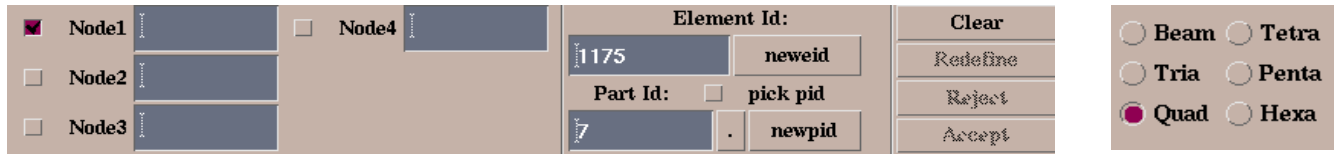
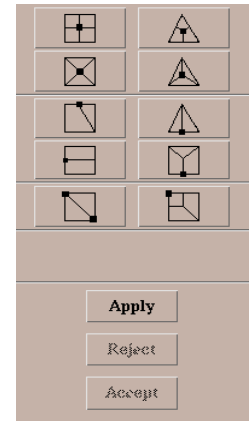
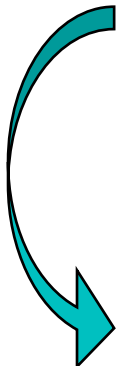
Modifying nodes with element quality check





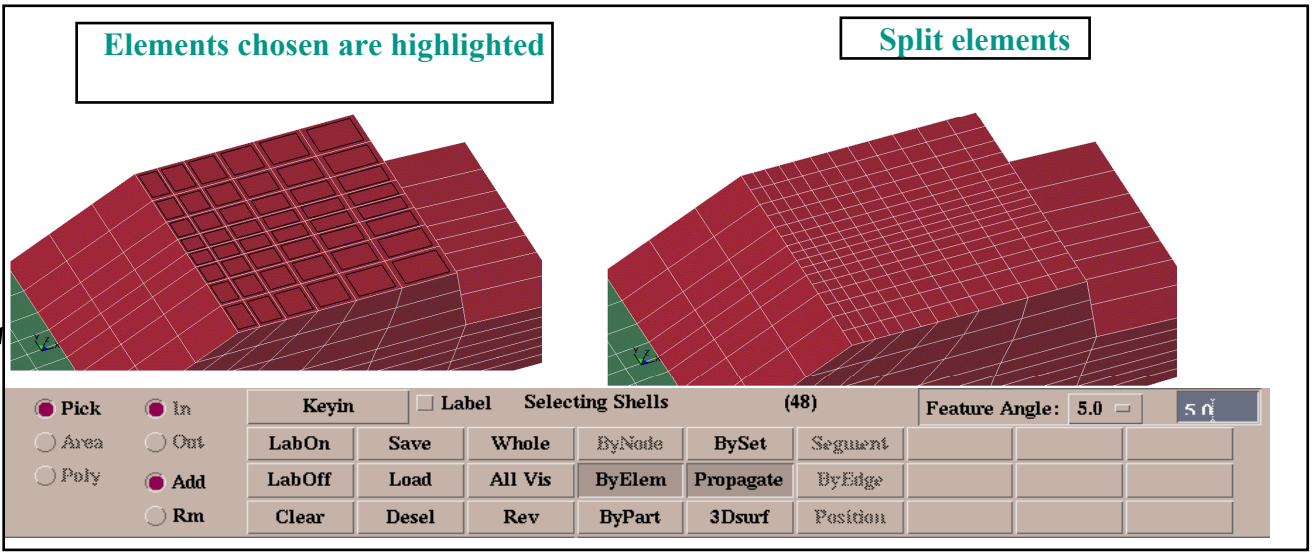
Element Edit - Element modification and creation

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



Split elements methods

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

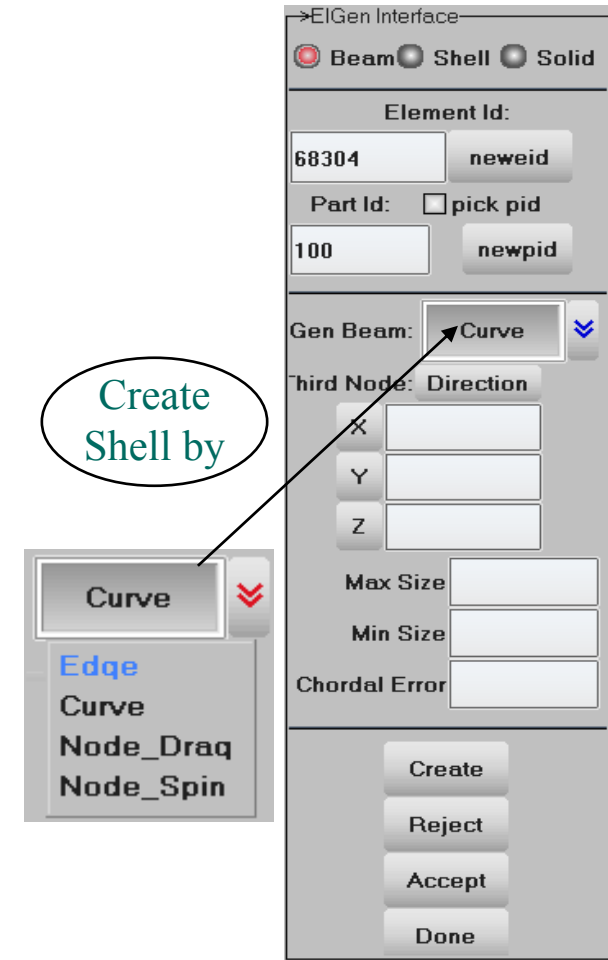




Element Generation - Beam

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

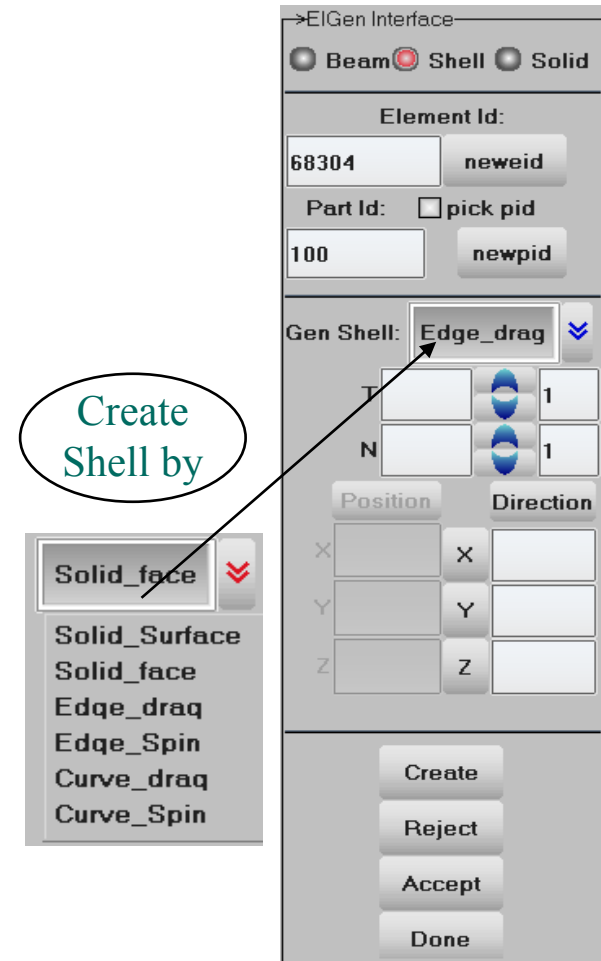
- ❑ Beam elements can be created by selecting an edge, a curve or create nodes using either drag or spin feature
 - ❖ Select an existing node or create a node using the ELEDIT feature
 - ❖ T- Element length, N- number of elements in given length
 - ❖ Select the direction in either global or local system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





Element Generation - Shell

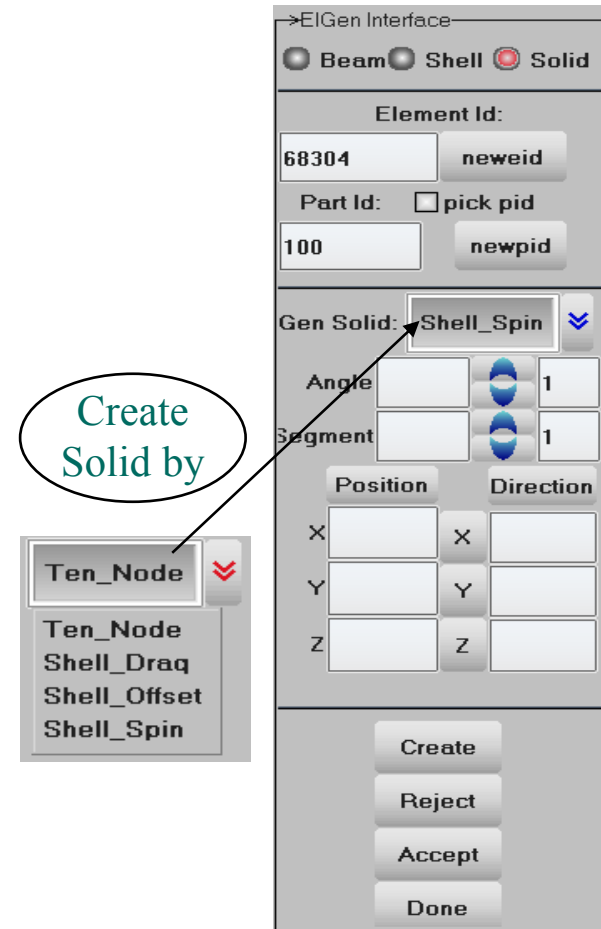
- ❑ Shell elements can be created by selecting an edge, a curve, solid element faces or surfaces
 - ❖ By selecting solid surfaces the entire solid surface can be chosen and shell elements created
 - ❖ By selecting solid faces, only a single face of the solid entity is selected
 - ❖ By selecting an edge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





Element Generation - Solid

- ❑ Solid elements can be created by shell drag, offset and spin from
 - ❖ Create ten node tetrahedron by selecting a 4 node tetrahedron.
 - ❖ By selecting a shell surface the elements can be dragged in a local or global direction
 - ❖ Solids can also be created using offset in shells
 - ❖ Solids can be created using a spin axis and direction (axi-symmetric)





Model data Manipulation

- Sub-systems – allow sub assemblies to be formed and manipulated
- Part data – creation, deletion and manipulation
- Renumbering – nodes, elements, parts, sets, curves, constrained data, and any data using ID
- Normal – shell element and segment data normal manipulation
- Dupgrid – eliminate duplicated nodes
- Detach – make a group of elements separate from each other
- Refcheck – reference check, and attachment check



Subsystem

SUBSYSTEM Creation and modification

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

CFD	Cgat	Stereo
Binout	Skid	AleMat
Subsys	PTravel	PTrim
Renum	RefCheck	MatDB
ElGen	Detach	Reorg
ElEdit	Curves	Offset
Movcopy	Dupgrid	Normals
Trnsfrm	Reflect	Project
Translt	Rotate	Scale

1 2 3 4 5 6 7 D

SubSystem

Sele Crea Modi
 Dele Move Tree

All None Rev Done
 Apply Cancel Write Manage

1 example.k

SubSystem

Sele Crea Modi
 Dele Move Tree

All None Rev Done
 Apply Cancel Write Manage

1 example.k
 2 Test(empty)

Create Subsystem

Subsystem

Sele Crea Modi
 Dele Move Tree

All None Rev Done
 Apply Cancel Write Manage

1 example.k

Subsystem

Name: Test

ParentId: 1

Filename:

Test





Subsystem

❑ Modifying a Subsystem

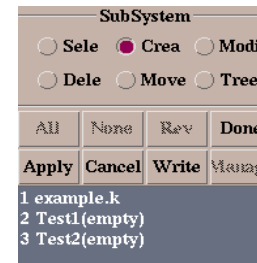
- ❖ The parent ID and the name of the subsystem can be modified using this feature
- ❖ Create a new subsystem “Test2”
- ❖ After Creation select “Modi” and choose “Test2”
- ❖ Change Parent ID from 1 to 2

❑ Delete Subsystem

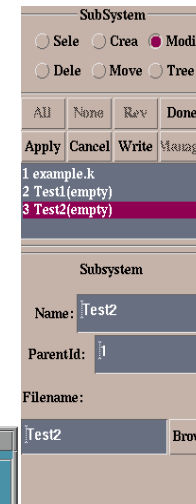
- ❖ Used to delete subsystems

❑ Move – Subsystem Management

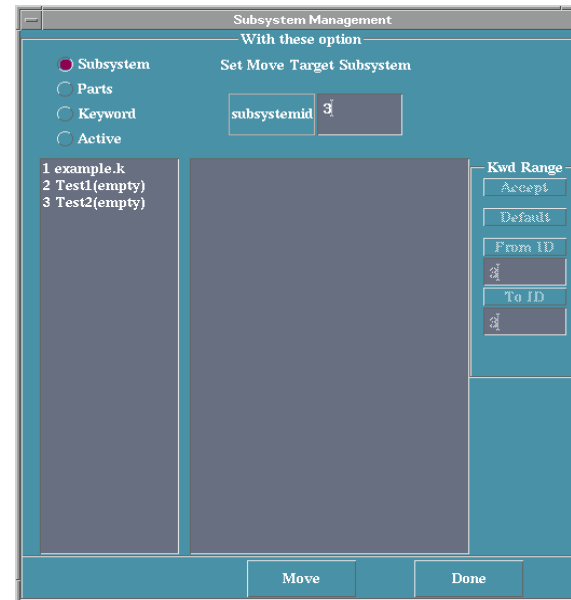
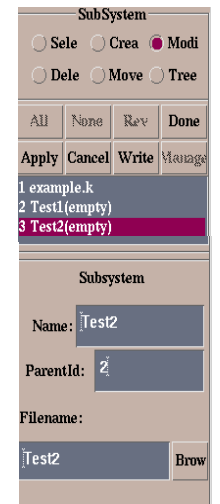
- ❖ This feature can be used to move parts, active parts (parts selected using “selpar” on page1) to alternate subsystems



Create Test2



Modify Parent ID



Subsystem management





Subsystem

The image shows four instances of the 'Set Move Target Subsystem' dialog box. Each instance has a title bar 'With these option' and a subtitle 'Set Move Target Subsystem'. The dialog contains radio buttons for 'Subsystem', 'Parts', 'Keyword', and 'Active'. A 'subsystemid' field contains the value '3'. Below the dialog, there are four labels: 'By part', 'By keyword', 'By Active Part', and 'Moving part 1,2 & 3 to subsystem 3'. The 'By Active Part' screenshot shows a list of parts: CONTACT(2), CONTROL(1), DATABASE(10), ELEMENT(1174), HOURGLASS(6), INITIAL(576), MAT(6), NODE(1236), PART(6), RIGIDWALL(2), SECTION(6), and SET(3).

Move – Subsystem Management

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

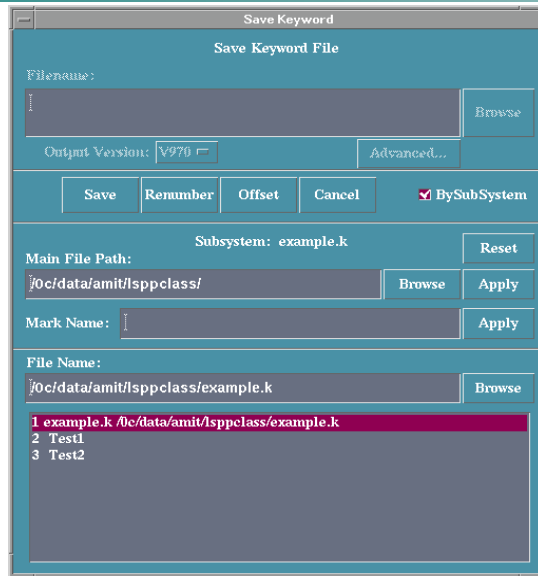
The image shows a 'SubSystem' menu with the following options: Sele, Crea, Modi, Dele, Move (selected), and Tree. Below the menu are buttons for 'All', 'None', 'Rev', 'Done', 'Apply', 'Cancel', 'Write', and 'Manage'. A list of items is shown below the buttons: 1 example.k, 2 Test1, 3 Test2.

All subsystems now have some parts in them

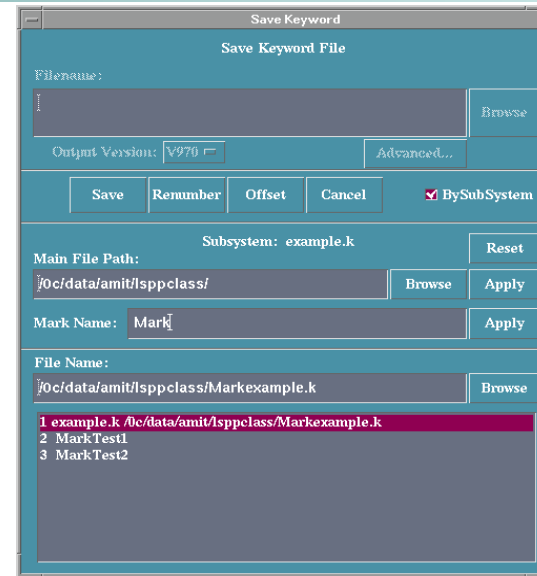




Subsystem



Saving subsystems



Save subsystems by marks

❑ Save file by selecting “by Subsystem”

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



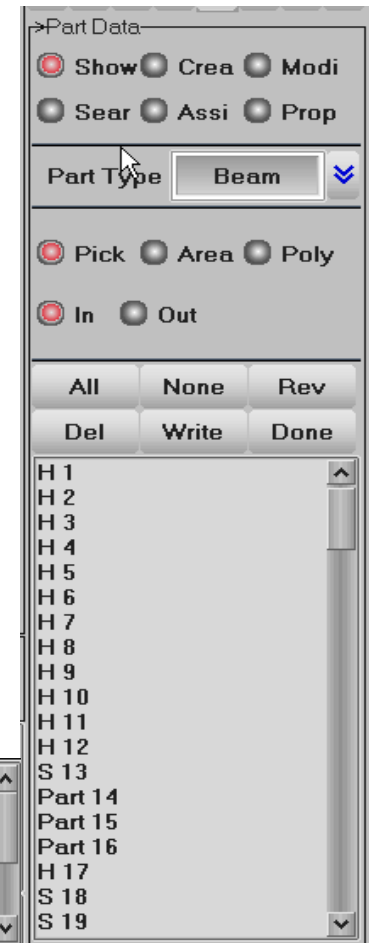
Part Data

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

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

```

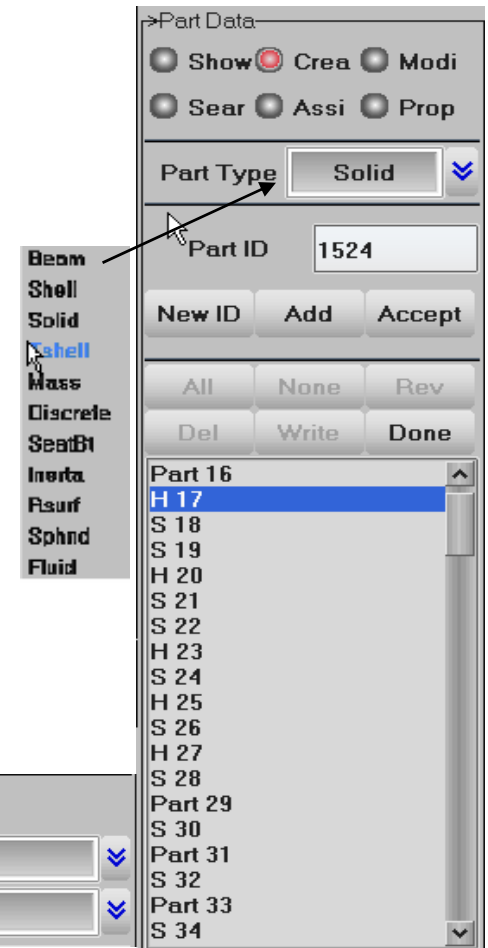
Solid PID=2 SID=2 MID=2 EOSID=0 HOURGID=0 #Elms=16 #Nodes=48 MinEID=2 MaxEID=17 MinNID=8 MaxNID=55
Solid PID=3 SID=3 MID=3 EOSID=0 HOURGID=0 #Elms=16 #Nodes=48 MinEID=18 MaxEID=33 MinNID=57 MaxNID=104
Solid PID=5 SID=5 MID=5 EOSID=0 HOURGID=0 #Elms=16 #Nodes=48 MinEID=242 MaxEID=257 MinNID=207 MaxNID=416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





Part Data

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



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





Part Data

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

SECID	MID	EOSID	HGID	TMID	GRAV	ADPOPT
13					NA	NA

Load RefBy Buff1 Search Part Apply

Part Data

Show Crea Modi
Sear Assi Prop

Part Type Solid

Search/Assign New data

All None Rev
Del Write Done

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





Part Data

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

The screenshot shows the LS-PrePost interface. On the right is the 'Part Data' dialog box with the following options: Show, Crea, Modi, Sear, Assi, Prop (selected), Part Type: Shell, Pick, Area, Poly, In, Out, and buttons for All, None, Rev, Del, Write, Done. A list of parts is shown below, with 'S 41' selected. On the left is the 'Material Data for Part Id : 65' form showing the keyword '*MAT_RIGID' and numerical values: 65, 6.0000E-7, 0.005000, 0.300000, 0.0, 0.0, 0.0.



Renumber

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

Renumber all Ids

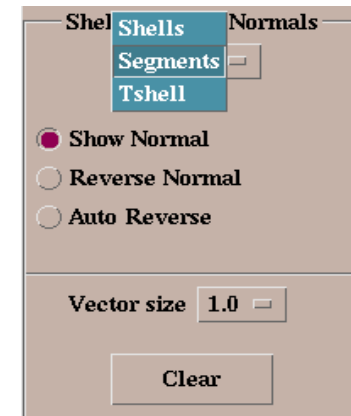
Renumber By range



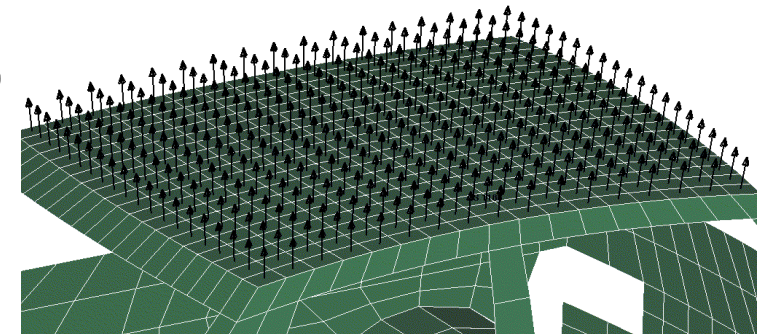
Normals

Element normals can be visualized and modified using this interface

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



Keyin	<input type="checkbox"/> Label	Selecting Shells (0)		
LabOn	Save	Whole	ByNode	BySet
LabOff	Load	All Vis	ByElem	Propagate
Clear	Desel	Rev	ByPart	3Dsurf

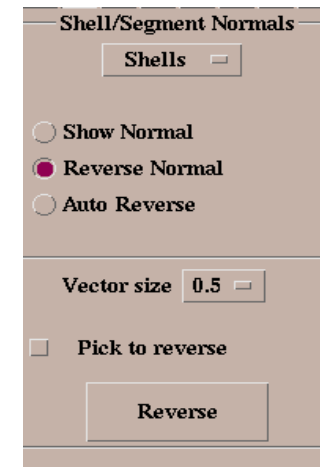
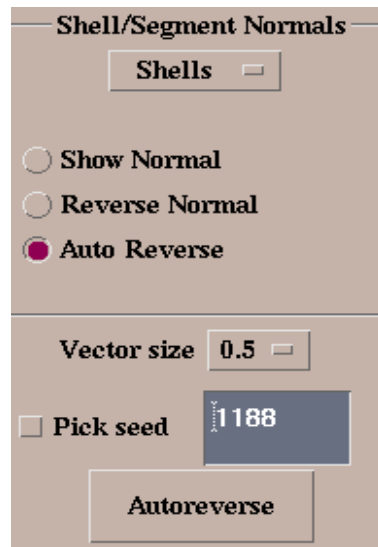




Normals

To Reverse the normals of the selected elements

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



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



Duplicate grids

Dupgrid interface allows the user to show and merge duplicate grids

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



Dupgrid Show/Merge

Tolerance:

Keep smaller NID

Keep Larger NID

Show Free Edges

Line Width

Line Color



Detach

Detach: This interface allows the user to detach elements.

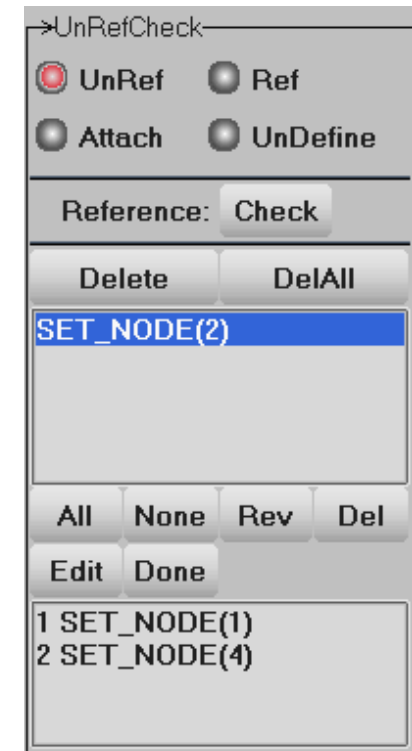
- Shell, Beam, Solid, Tshell – Separate chosen element type
- Node – Separate all elements at a node
- Element – Select group of elements to be detached
- Edge – Select edges of elements to be detached
- Starting NID – Enter starting node ID
- Show Free Edges – Displays free edges in white. Displays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

The screenshot shows a dialog box titled "Detach Element". It contains two sections of radio buttons. The first section has "Shell" selected, with other options being "Beam", "Solid", "Tshell", and "Node". The second section has "Element" selected, with "Edge" as the other option. Below these is a text field for "Starting NID" containing the value "3947". There is a checkbox for "Show Free Edges" which is currently unchecked. At the bottom are four buttons: "Detach", "Accept", "Reject", and "Done".



Reference Check - Unreferenced and Undefined

- ❑ **Use this to check and delete all unreferenced entities**
 - ❖ Check for unreferenced entities
 - ❖ Delete selected unreferenced data
 - ❖ Delete all unreferenced data
 - ❖ List of unreferenced entity types found
 - ❖ Select, deselect, reverse unreferenced data of selected entity type
 - ❖ Delete selected unreferenced data of selected entity type
 - ❖ Edit selected unreferenced data of selected entity type
- ❑ **Undefined entities can be checked/edited/deleted with this interface**

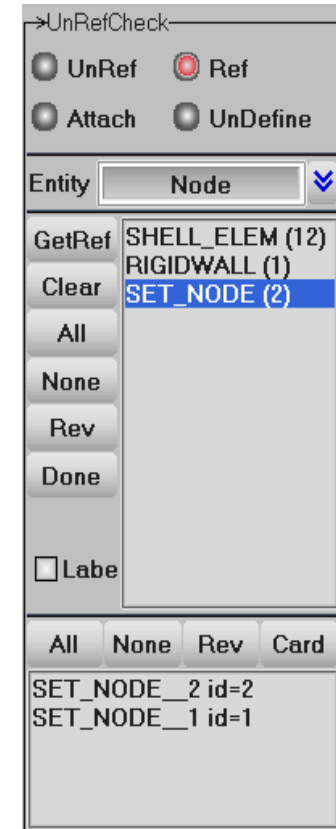




Reference Check – Reference

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

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

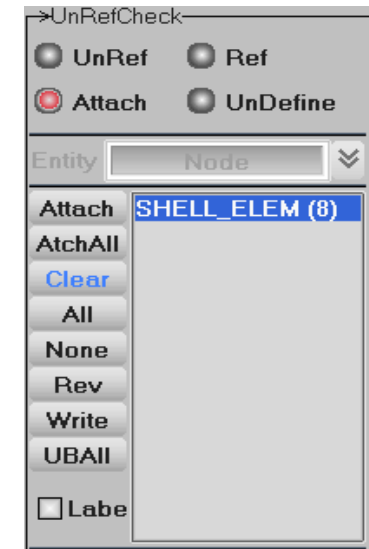
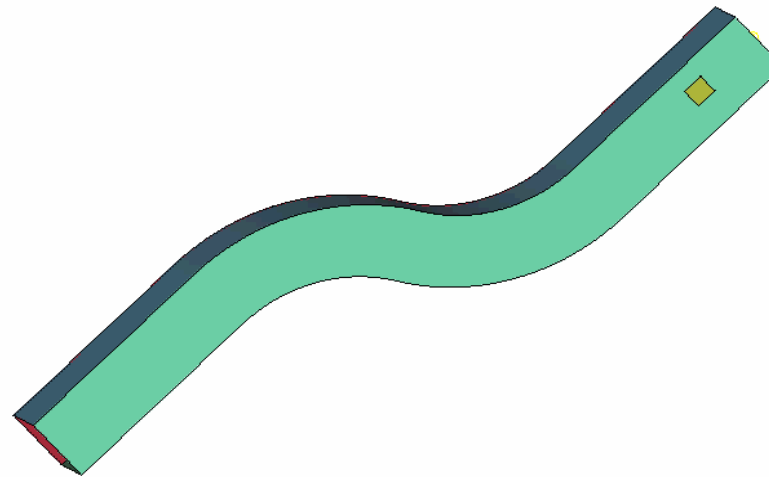




Reference Check - Attach

□ Use this to find all the attached entities to the entity selection by using bottom area general selection to get nodes

- ❖ Get all entities that are attached to the entities picked step by step or get all.
- ❖ Write the displayed entities into a file
- ❖ Unblank all the parts in display for restore model after attach
- ❖ Attach for pick node by general selection.

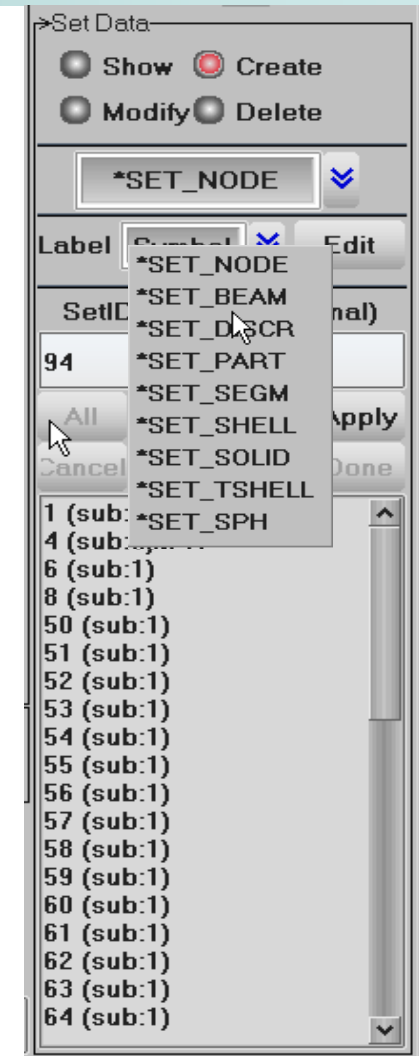




LSDYNA data - SetD

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

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

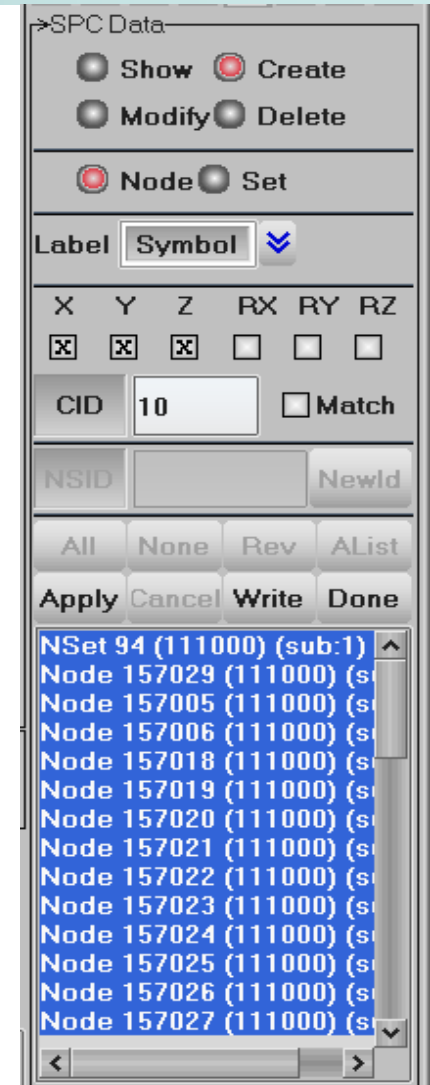




LSDYNA data - SPC

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

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

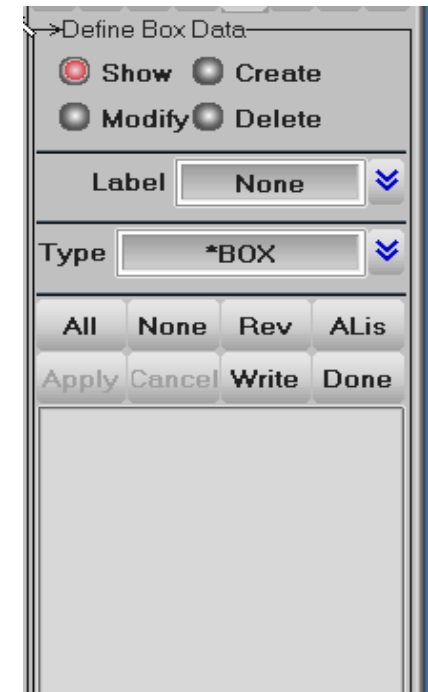




LSDYNA data - Box

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

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



TITLE	<input type="text"/>			<input type="radio"/> MinMax	<input type="radio"/> Translate	<input type="button" value="Allvis Node"/>
BOXID	<input type="text" value="1"/>	X	Min: 1954.12, Max: 3299.71	<input checked="" type="radio"/> Area In	<input type="radio"/> Scale	
		Y	-816.278, 16.0055	<input type="radio"/> Poly In		
		Z	142.818, 1140.2, 19551			





LSDYNA data – Mass Element

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

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

Mass Data

Show Create
 Modify Delete

Label ▾

Mass

All **None** Rev AList

Apply Cancel Done

1	NID(157011)	(sub:1)
2	NID(157010)	(sub:1)
3	NID(157045)	(sub:1)
4	NID(157002)	(sub:1)
5	NID(157000)	(sub:1)
6	NID(157001)	(sub:1)
7	NID(157003)	(sub:1)
8	NID(157004)	(sub:1)
9	NID(157007)	(sub:1)
10	NID(157008)	(sub:1)
11	NID(157009)	(sub:1)
12	NID(157012)	(sub:1)
13	NID(157013)	(sub:1)
14	NID(157014)	(sub:1)
15	NID(157015)	(sub:1)
16	NID(157016)	(sub:1)
17	NID(157017)	(sub:1)
18	NID(157041)	(sub:1)
19	NID(157042)	(sub:1)
20	NID(157043)	(sub:1)
21	NID(157044)	(sub:1)
22	NID(157046)	(sub:1)
23	NID(157047)	(sub:1)



LSDYNA data - CNRB

CNRB – to show/create/modify/delete

*constrained_nodal_rigid_body

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

→CNRB Data

Show Create
 Modify Delete

Label ▾

PID	CID	NSID
2		96

Pick PNODE

PNODE

IPRT ▾

DRFLAG ▾

RRFLAG ▾

All None Rev AList

Apply Cancel Done

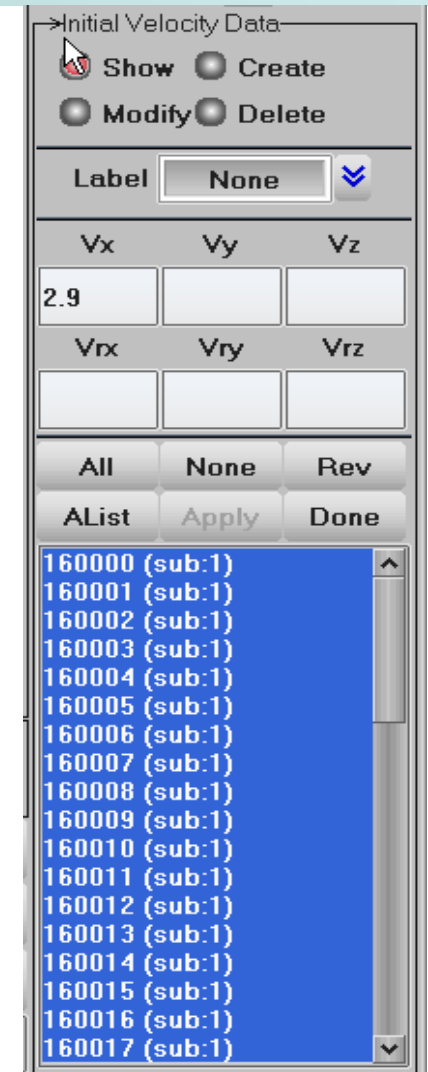
part(1).NSET95 (sub:1)



LSDYNA data – Initial Velocity

Inivel – to show/create/modify/delete *Initial_velocity data

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





LSDYNA data – Spotweld/Rivet

Spweld/Rivet – to show/create/modify/delete *Constrained_Spotweld and *Constrained_Rivet data

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

Rivet Data

Show Create
 Modify Delete

Label Autocreate

RID N1 N2

TF

All None Rev AList

Apply Cancel Done

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



LSDYNA Data – Coordinate System

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

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

>Coordinate Data

Show Create

Modify Delete

Label ▾

Coord Type ▾

CID NewID

Title

All None Rev AList

Apply Cancel Write Done

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



LSDYNA Data – Constrained_node_set

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

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

→CONSTN Data

Show Create
 Modify Delete

Label ▾

CNSID	NSID	
<input type="text"/>	98	New ID
DOF	<input type="text" value="1"/> ▾	TF <input type="text" value="1.0E+20"/>

All None Rev AList

Apply Cancel Done

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



LSDYNA Data – Generalize_weld

Gweld – to show/create/modify/delete

*Constrained_generalize_weld_spot data

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

WID	NSID
	98



LSDYNA Data - Seatbelt Accelerometer

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

Accelerometers Data

Show Create

Modify Delete

Label ▾

SBACID NewID

All None Rev AList

Apply Cancel Done

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

<input checked="" type="checkbox"/>	NID1	<input type="text" value="0"/>	IGRAV	<input type="text" value="0"/>	▾
<input type="checkbox"/>	NID2	<input type="text" value="0"/>	INTOPT	<input type="text" value="0"/>	▾
<input type="checkbox"/>	NID3	<input type="text" value="0"/>			



LSDYNA Data - RigidWall

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

>RigidWall Data

Show Create

Modify Delete

geometry planar

Cylind Flat

Prism Sphere

Label ▾

All None Rev AList

Apply Cancel Done

1 GEOMETRIC_CYLINDER
 2 GEOMETRIC_CYLINDER
 3 GEOMETRIC_FLAT (sub:1)
 4 GEOMETRIC_PRISM (sub:1)
 5 GEOMETRIC_SPHERE (sub:1)
 6 PLANAR (sub:1)
 7 PLANAR_FINITE (sub:1)
 8 PLANAR_ORTHO_FINITE

Change bottom working area

Modify preview Rigid wall

Cylinder		Tail	Head	<input checked="" type="checkbox"/> Node	17110	<input checked="" type="radio"/> NormX	<input checked="" type="radio"/> 1n+NL	<input type="radio"/> TRAN	<input checked="" type="radio"/> GeoVector
Radius	10	X	13.525	14.5251		<input type="radio"/> NormY	<input type="radio"/> N1-N2	<input type="radio"/> ROTA	<input type="radio"/> Title
Length	20	Y	76.4262	76.426		<input type="radio"/> NormZ	<input type="radio"/> 3Nds	<input type="radio"/> REFL	<input type="radio"/> SlaveNode
		Z	70.0000	70.000					<input type="checkbox"/> Motion

C
L
E
A
R
Cre_Motion





Database Cross Section

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

→XSection Data

Show Create
 Modify Delete

Plane Set

Label ▾

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

XV	YV	ZV	Tail	Head	<input checked="" type="checkbox"/> Node	62872	<input checked="" type="radio"/> NormX	<input checked="" type="radio"/> 1n+NL	<input type="radio"/> TRAN	C L E A R	CSID	ITYPE	J
LENL	LENM	X	0	0			<input type="radio"/> NormY	<input type="radio"/> N1-N2	<input type="radio"/> ROTA		Title		
0.0	0.0	Y	0	0			<input type="radio"/> NormZ	<input type="radio"/> 3Nds	<input type="radio"/> REFL		PSID		Cre_PSet
		Z	0	0									

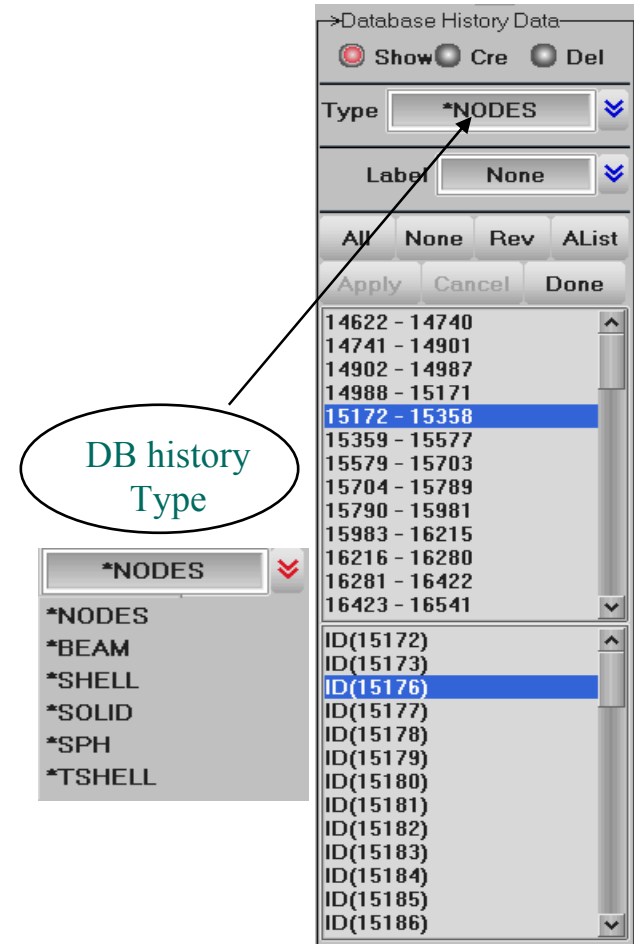




Database History

□ Show, create and delete Database History

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





Tied Nodes Failure

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

→TiedNF Data

Show Create
 Modify Delete

Label ▾

NID NewId

NSID NewId

EPPF

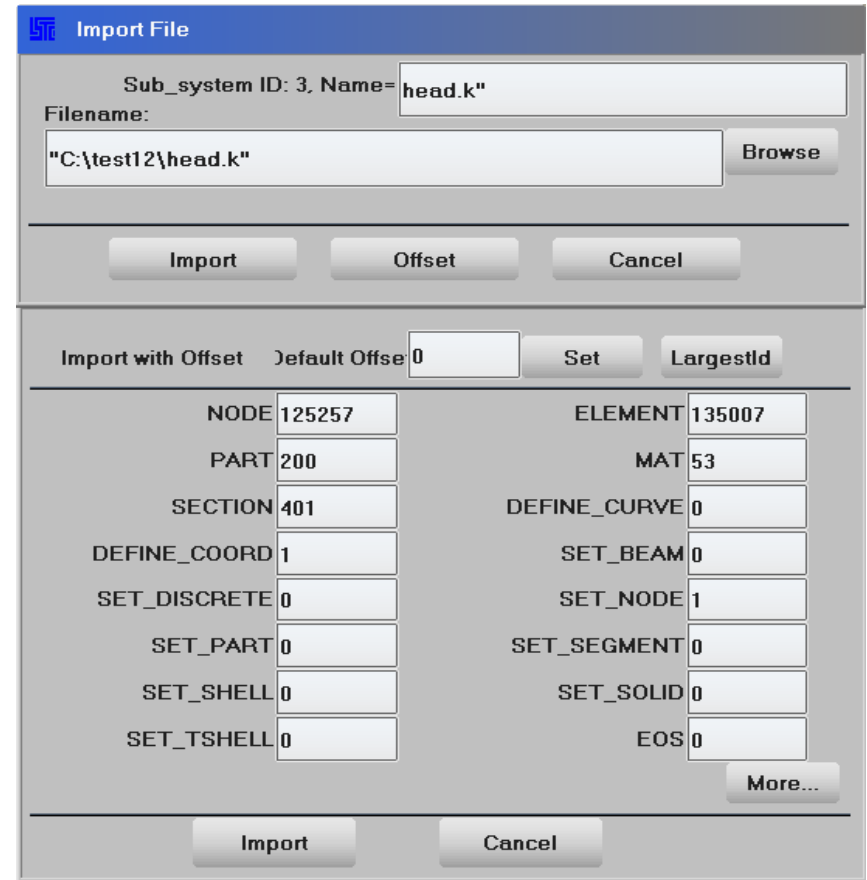
All None Rev
AList Apply Done

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



Input keyword

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





LSDYNA Keyword data

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

The screenshot shows the LS-PREPOST keyword editor interface. At the top, there is a grid of keyword buttons, some of which are highlighted in blue. Below this grid, there are buttons for 'Cre/Edt' and 'Delete', and a row of buttons for 'All', 'Model', and 'Done'. At the bottom, there is a list of keywords with their respective counts in parentheses. Three callout boxes with arrows point to specific elements: 'Pag3 and page4 for all keywords' points to the top grid, 'Keyword edit operation buttons' points to the 'Cre/Edt' and 'Delete' buttons, and 'Edit keyword by list' points to the list of keywords.

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Element	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwall
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rgc	*Intrfac	*Termnt
Damping	*Load	*User

→Keyword *CONTROL

Cre/Edt Delete

All Model Done

[*]ACCURACY (1)
[*]ENERGY (1)
[*]TERMINATION (1)
[*]Timestep (1)

Callouts:
- Pag3 and page4 for all keywords
- Keyword edit operation buttons
- Edit keyword by list



Keyword Edit - Feature

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

The screenshot displays two windows from the LS-PrePost software. The top window is titled 'KEYWORD INPUT' and shows a table for '*ELEMENT_SHELL (91352)'. The table has columns for EID, PID, N1, N2, N3, and N4. The first row contains the values 1, 1, 10, 1, 2, and 19. Below the table is a 'COMMENT:' field. The bottom of the window shows summary statistics: 'Total Card: 91352', 'Smallest ID: 1', 'Largest ID: 2091352', and 'Total deleted card: 0'. The right side of the window has a list of node IDs from 1 to 10.

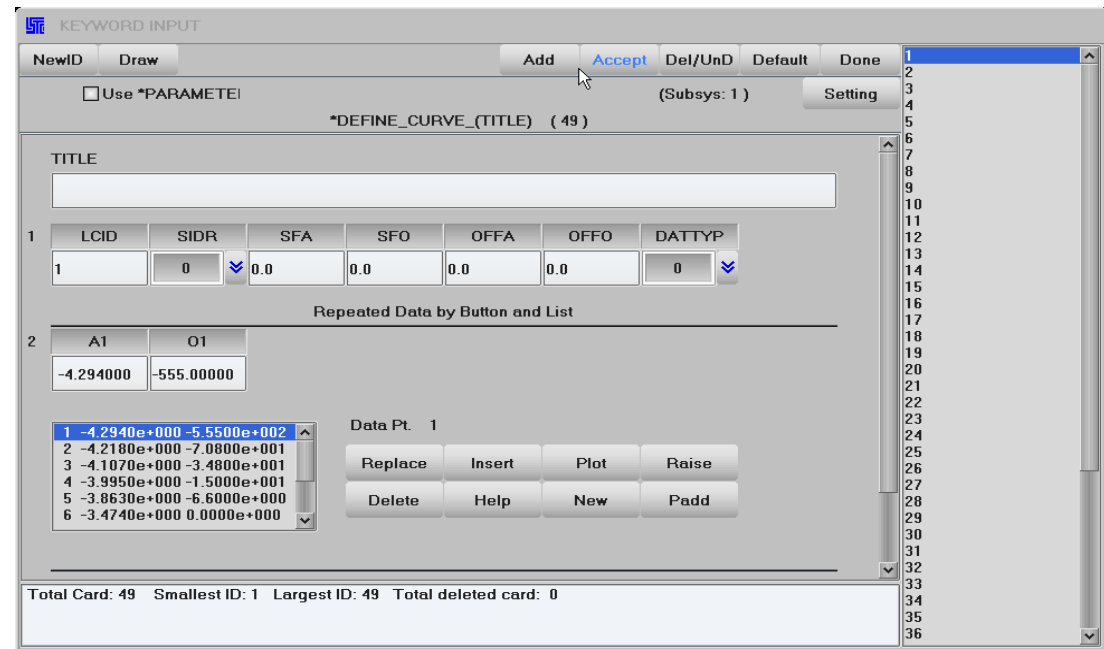
The bottom window is titled 'KEYWORD INPUT' and shows a table for '*NODE (91475)'. The table has columns for NID, X, Y, Z, TC, and RC. The first row contains the values 1, 0.0, 1.00000, -0.10002, 0, and 0. Below the table is a 'COMMENT:' field. The right side of the window has a list of node IDs from 1 to 13.

Two callouts are present: 'Read data' with an arrow pointing to the top window, and 'Link Dialog' with an arrow pointing to the bottom window.



Keyword Edit - Form

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



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



Keyword Edit - Sort and Reference

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

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





Keyword Edit – Delete

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



The screenshot shows the 'KEYWORD INPUT' window with a table of keywords. A dialog box titled 'Delete Mat' is open, displaying the message '5 card(s) have been marked for deletion' and 'Delete' and 'Cancel' buttons. The status bar at the bottom indicates 'Total Card: 9 Smallest ID: 1 Largest ID: 401 Total deleted card: 5'.

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SET
	20	2	1.00000	5	1	0.0	1	1
2	T1	T2						
	2.00000	2.00000						



Save Keyword File

Save keyword data into file

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

Save Keyword File

Filename: Browse

Output Version: V960 Advanced...

Save Renumber Offset Cancel BySubSystem

Subsystem: main.k

Main File Path: C:\test12\ Browse Apply

Mark Name: Apply

File Name: C:\test12\main.k Browse

1 main.k C:\test12\main.k
2 single.fold.k C:\test12\single.fold.k
3 single.fold.k C:\test12\single.fold.k
4 single.fold.k C:\test12\single.fold.k
5 ih.k C:\test12\ih.k



Save Active keyword

Save Active keyword data into file

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



Import of other formatted files

- Import of Nastran bulk data
 - ❖ Elements
 - ❖ Grids
 - ❖ Material
 - ❖ SPC
- Import of STL files
 - ❖ Ascii format
 - ❖ Binary (automatically endian swapped)
- Import of Abaqus input files



LIVERMORE SOFTWARE TECHNOLOGY CORPORATION

LS-PREPOST

LS-PREPOST

Applications





Metal Stamping – Part Travel

Purpose: To measure the distance between two parts

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

The screenshot shows the 'Part Travel' dialog box with the following settings:

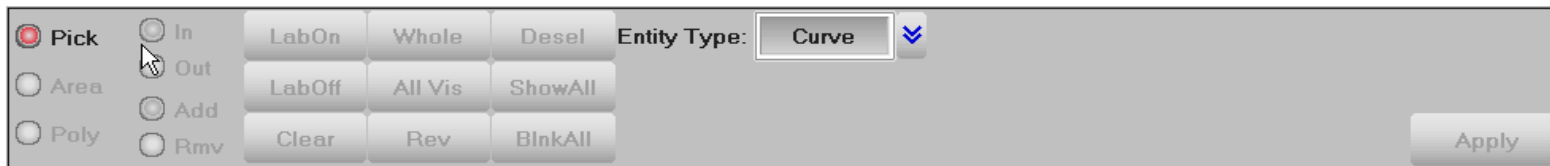
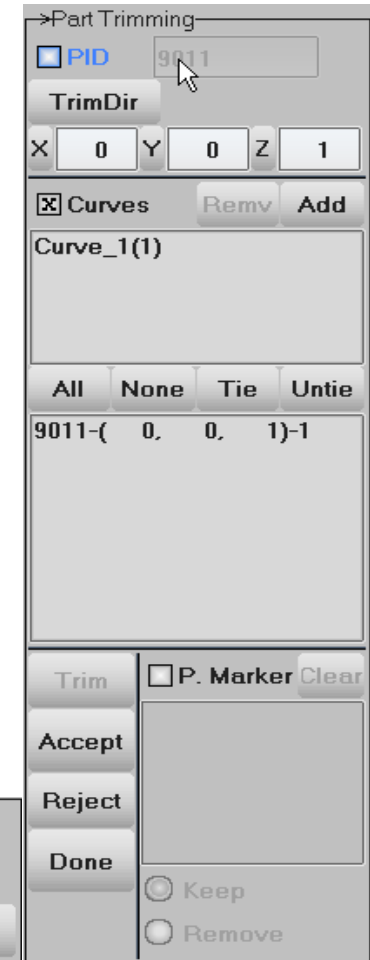
- Moving Pt: 4
- Fixed Pt: 2
- Coordinate Systems: Global (selected)
- Distance: 0.0
- Travel Axis: AX (selected), AY, AZ
- T1, T2
- Closest Axial Point: @ node: ,Elem:
- Buttons: Compute, Move, AutoPos, Animate, Reset, Accept, Done



Metal Stamping - PTRIM

• Part Trimming

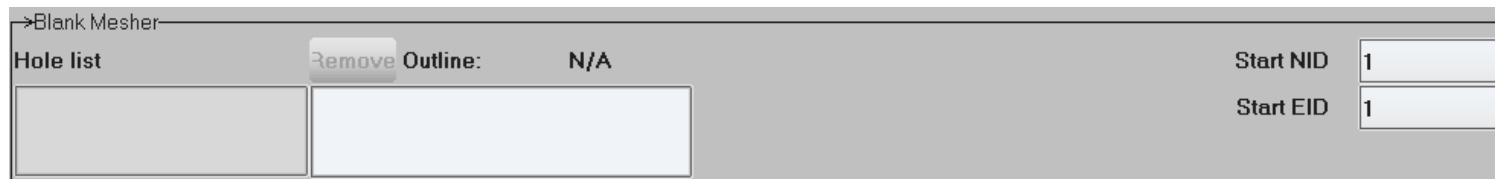
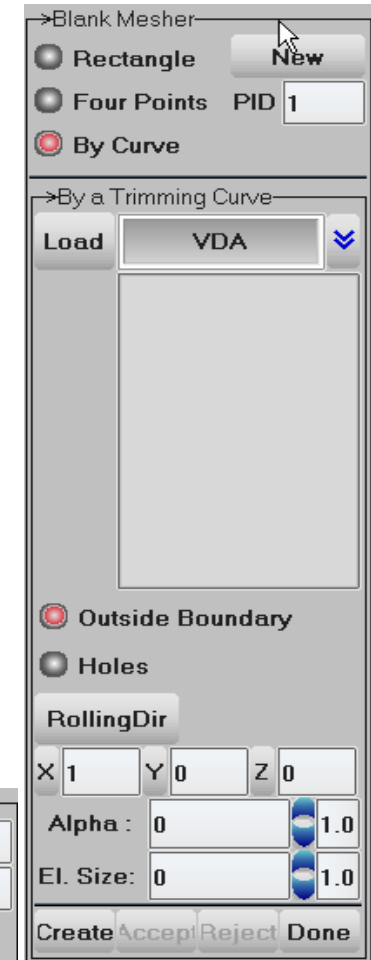
- Trimming a given part with open curves.
 - ❖ Trimming algorithms taking care of open curves.
 - ❖ Several open curves are allowed to trim at the same time.
 - ❖ Seed nodes can be provided to keep/remove the needed region.
- Trimming algorithm will take care of the adapted elements as well as element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





Metal Stamping - BMesh

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





Wall Creation for Binder

Purpose: To create binder wall

Selects the edges of the binder to create wall.

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

Binder Wall Data

Pick Partid

5 newpid

40413 Neweid

angle 15 5

El_normal Direction

Direction

dirx 0

diry 0

dirz 1

Reverse Direction

length 12 1

Segment 1 1

Build

Reject

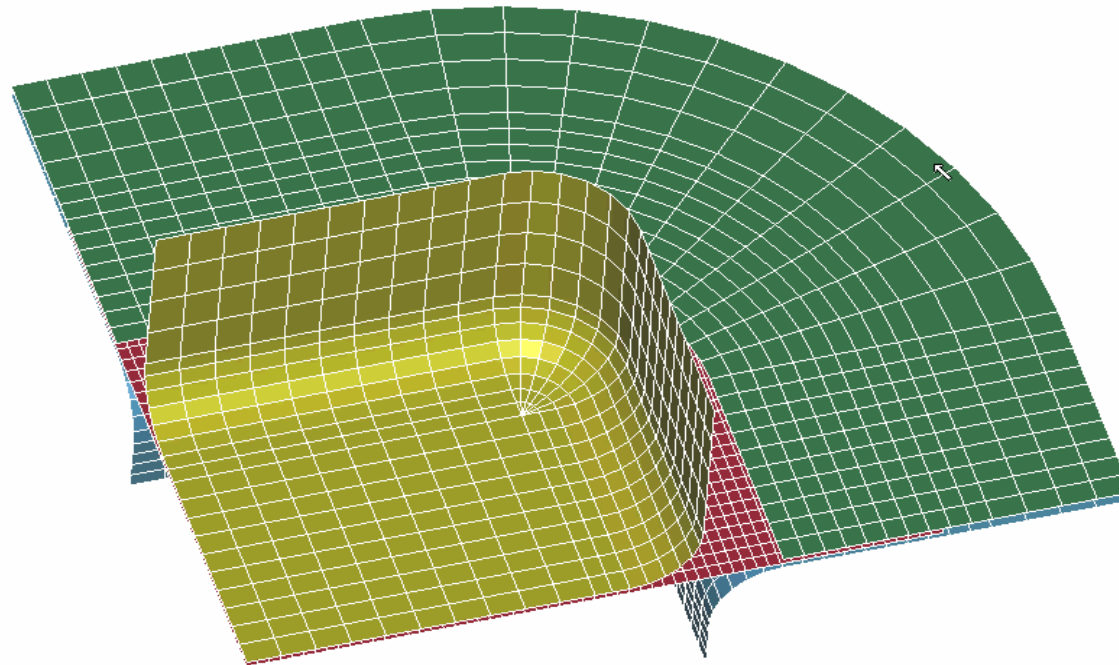
Accept

Done



Wall Creation for Binder

Rectangular Cup Drawing





Guide Building

Purpose: To create guides that will keep the blank in position

Selects a direction and two positions to create Guide.

Giving angle, length and below percent.

→Guide Building Data

<input type="checkbox"/> Pick	Partid	
6	newpid	
40417	Neweid	
Pos1	Pos2	
X 89	114	
Y 20.4842	20.4842	
Z 2	2	
Direction		
dix	0	
diry	0	
dirz	1	
Angle	15	5
Length	40	
below	10	5
Build	Reject	
Accept	Done	

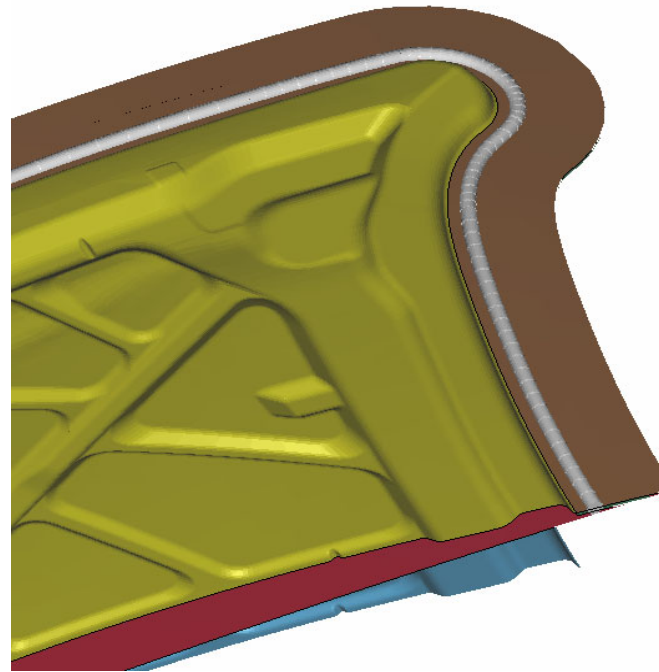




Metal Stamping – Draw Bead

To show/create/modify/delete drawbead data

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



→Drawbead Data

Show Create
 Modify Delete

SSTYPE BeamPart

Pick PBeam
 Pick MSID
 Pick Dield
 Pick Boxid 9

Creat Apply

Depth LCIDRF LCIDNF
LCEPS2 OFFSET LCEPS

Show All

All None Rev AList
Apply Card Done

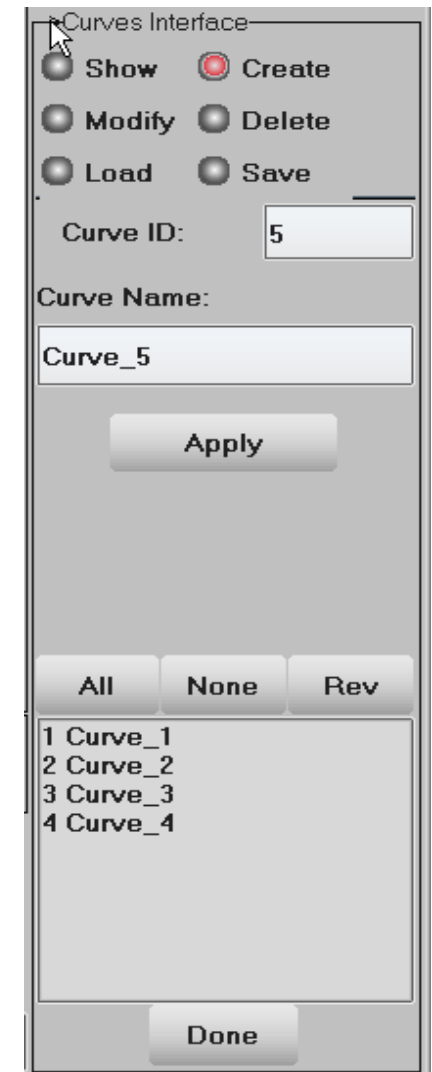
1 - 5 (sub:1)
2 - 6 (sub:1)
3 - 7 (sub:1)
4 - 8 (sub:1)
5 - 9 (sub:1)
6 - 10 (sub:1)
7 - 11 (sub:1)



Metal Stamping – Curves

To show/create/modify/delete curves data

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

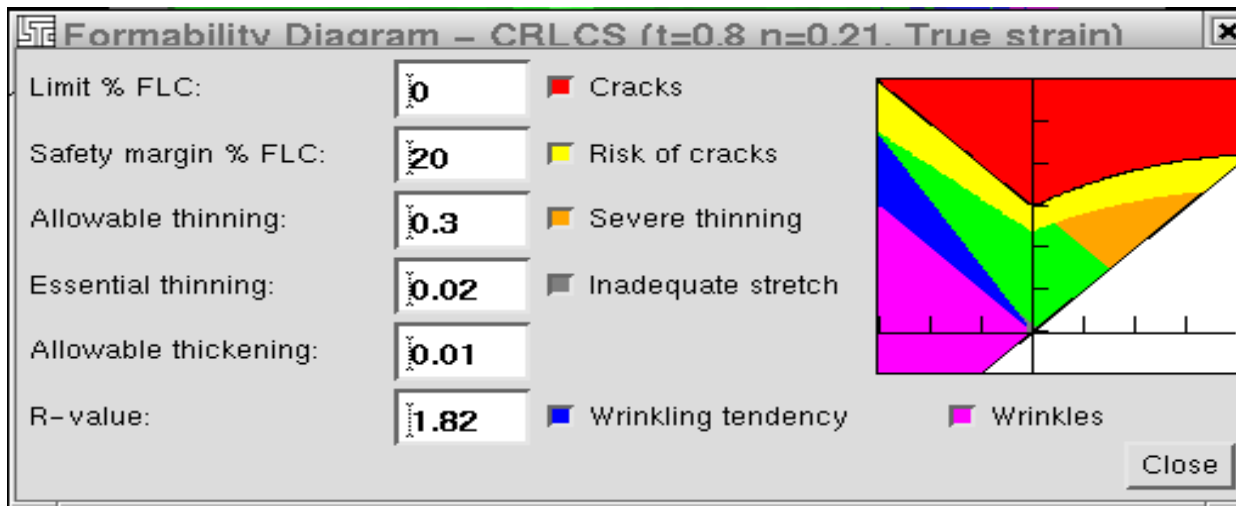




Metal Stamping – FLD Forming Limit Diagram



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



Forming Limit Results

Calculated FLD Curve:

t= 0.8 n= 0.21 Set

t_n File Browse

Point Tracer

FLD E'Strain

Material Clr Info

Element Formability

Area

Polygon In Out

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

Position: Middle

Plot New Padd

Clear Raise Pop Done





LIVERMORE SOFTWARE TECHNOLOGY CORPORATION

Metal Stamping – FLD Forming Limit Diagram

LS-PREPOST

Input
61679, #elem=28898
Surface

PlotWindow-1

CRLCS (t=0.8 n=0.21, True strain)

Major True Strain

Minor True Strain

Formability key

- Cracks
- Risk of cracks
- Severe thinning
- Good
- Inadequate stretch
- Wrinkling tendency
- Wrinkles

Output	Trace	Xyplot
Anno	Light	FLD
SPlane	Setting	State
Range	Vector	Measur
Find	Ident	ASCII
Fcomp	History	Views
Appear	Color	Model
Group	Blank	SelPar

Forming Limit Results

Calculated FLD Curve:

t= n= Set

◆ t_n File Browse

◆ Point Tracer

FLD E'Strain

Material Clr Info

Element Formability

◆ Area In Out

Polygon In Out

E# = 39019,
Node = 32783, 32784, 165
31, 32785
T = 0.934, %R = 6.69
E1 = 0.134, E2 = -0.0514

Position: Middle

Plot	New	Padd
Clear	Raise	Pop
		Done

Formability Diagram - CRLCS (t=0.8 n=0.21, True strain)

Limit % FLC: Cracks

Safety margin % FLC: Risk of cracks

Allowable thinning: Severe thinning

Essential thinning: Inadequate stretch

Allowable thickening: Wrinkling tendency

R-value: Wrinkles

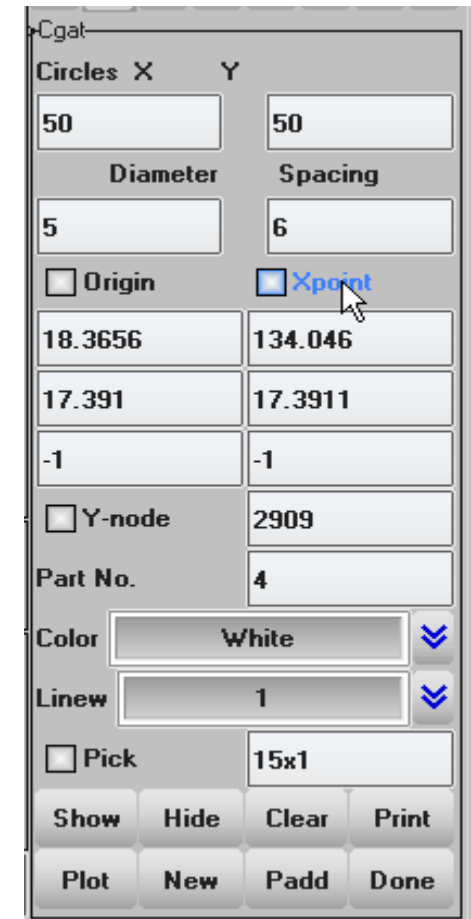
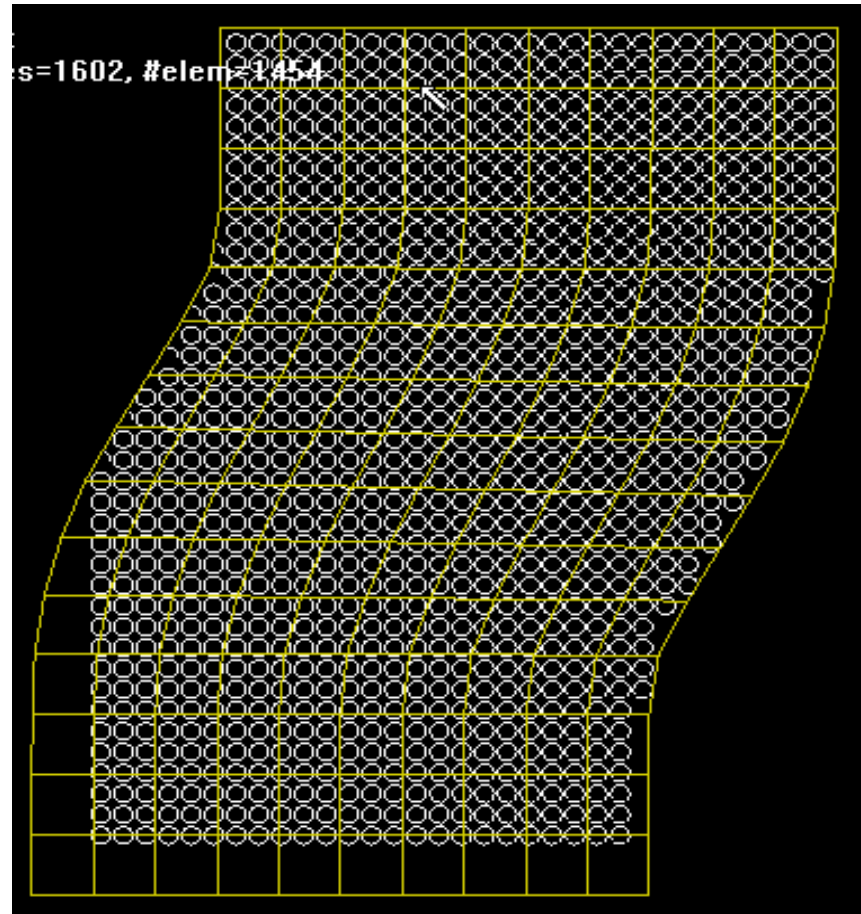
Close





Circular Grid Generation Technique

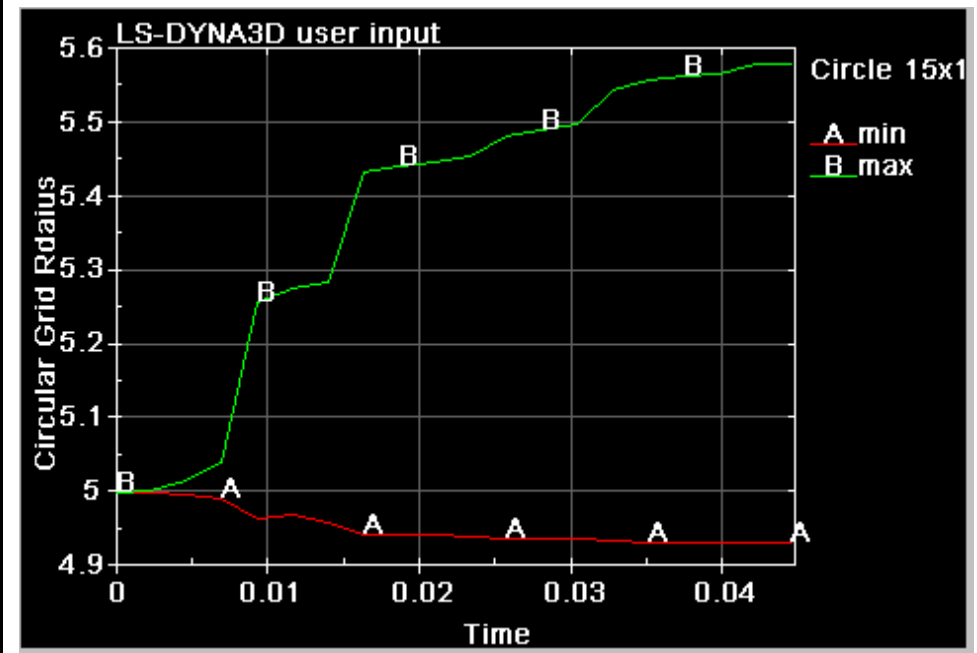
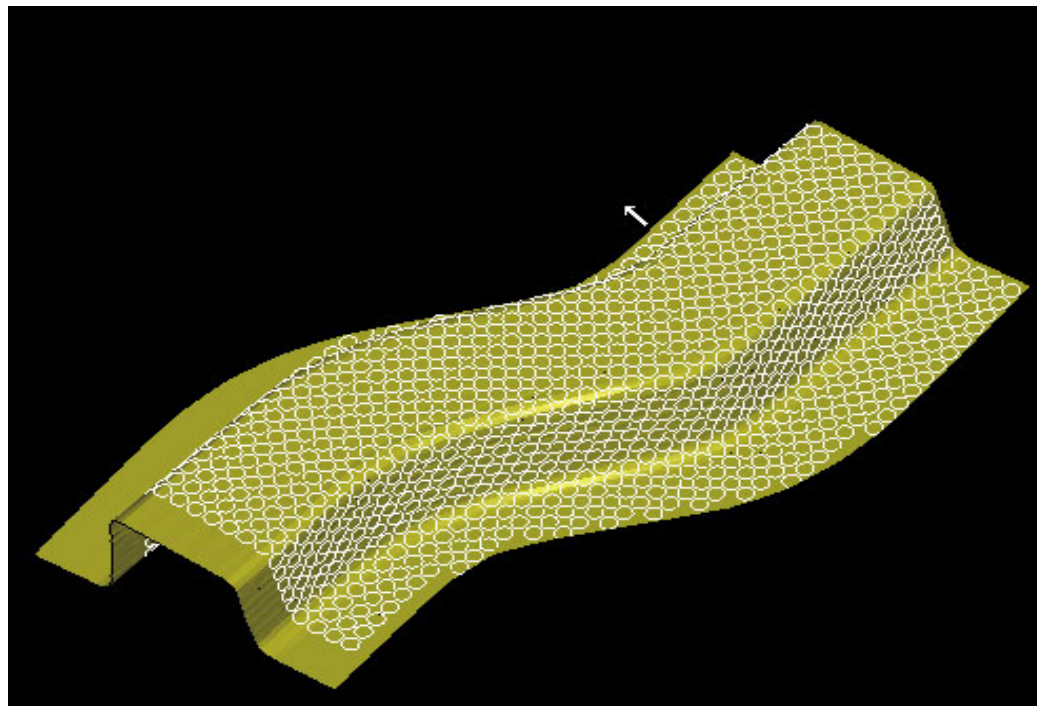
- A method for measuring the biaxial strain on a formed part.
- The menu allows this procedure to be simulated by tracing parametric points on the mesh through the forming simulation





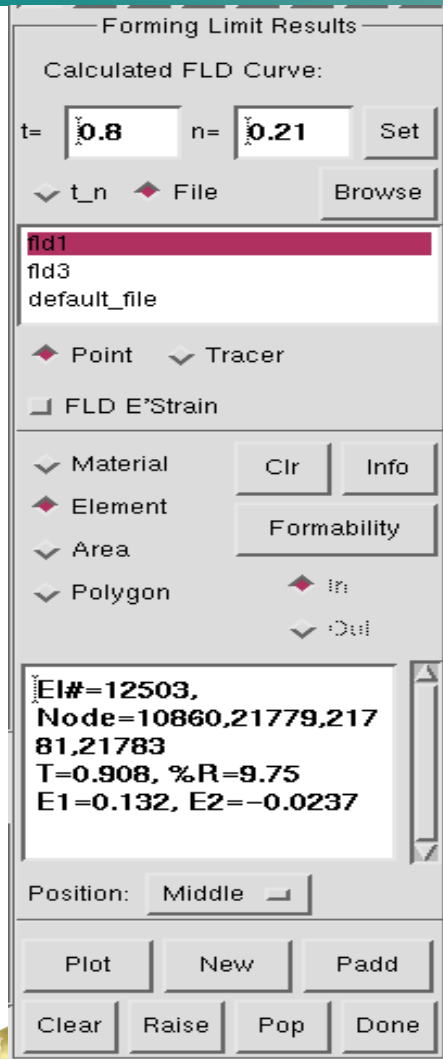
Circular Grid Generation Technique

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





Metal Stamping – FLD Forming Limit Diagram



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



Metal Forming Skid Mark Traces

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



View of Skid Traces

The screenshot displays the LS-DYNA3D user input interface. The main window is split into two views: a wireframe model on the left and a shaded model on the right. The wireframe model shows a curved surface with several vertical lines representing skid traces. The shaded model shows the same surface with a green color gradient. The interface includes a menu bar (File, Misc., Toggle, Background, Help), a toolbar with various functions (CFD, Cgat, Binaski, Skid, ElEdit, Dupgrid, Movecopy, Elcheck, Normals, Trnsfrm, Reflect, Project, Translt, Rotate, Scale), and a panel for Skid marks. The Skid marks panel includes a table with columns 1 through 7 and a D column, and a section for Skid marks with options for Pick blank, Pick curve, and Pick point. The Pick curve option is checked, and the X, Y, and Z coordinates are displayed. The Closest node is 419. The Curve Coords section lists five points with their x, y, and z coordinates. The Set line color/width section has radio buttons for Skid and Tool, and a color selection box set to White. The PlotWindow-1 window shows a graph of Skid Data vs Time. The graph has three data series: A (red), B (green), and C (blue). The Y-axis ranges from 0 to 3.5, and the X-axis ranges from 0 to 0.04. The legend indicates: A Skid Mark Trace Gap, B Skid Mark Trace Gap-2, and C Skid Mark Trace Gap-6. The Time is 0.02115. The interface also includes a navigation panel with buttons for Ry, Rz, Top, Front, Right, Redw, Home, All, Rpat, Bottom, Back, Left, Anim, Rset, and a section for Inc, SF, Time, No.of Div, State#, and Perf.





Stamping Process and Stage Manage

Multistage Process Setup

- Define a single stage
- Create keyword deck for a single stage
- Define a multiple stages
- Save/import stage templates
- Save/import process templates
- Establish Communication with Process Simulation Manager



Stamping Process and Stage Manager

The screenshot displays the LS-PREPOST 2.0 (Beta) - 14APR2005 software interface. The main window shows a 3D model of a stamped part with a blue top surface, a red middle section, and a yellow-green bottom section. A coordinate system (X, Y, Z) is visible in the bottom-left corner of the model area.

The 'Forming Process Interface' panel on the right contains the following elements:

- Buttons: New, Load, Save, Done
- Process: Fender
- Buttons: CAD, Pre, Stage, Post
- Buttons: Add, Rmv, Cpy, Ltmp
- Text: Drawing(none)
- Buttons: MkTmp, Write, Start, Info
- Buttons: Visual, Anim, Check, Optn

Annotations with red arrows point to:

- Process**: Points to the 'Fender' text in the Process field.
- Stage**: Points to the 'Drawing(none)' text in the drawing list.
- create keyword file**: Points to the 'MkTmp' button.

At the bottom of the interface, there is a toolbar with various options like Title, Legd, Tims, Triad, Bcolr, Mcolr, Frin, Isos, Lcon, Acen, Zin, +10, Rx, Deoff, Spant, Top, Front, Right, Redw, Home, Hide, Shad, View, Wire, Feat, Edge, Grid, Mesh, Shm, Pcen, Zout, //, Clp, All, Rpart, Bottm, Back, Left, Anim, Reset. Below the toolbar are radio buttons for Off, Shift, and Control, and a performance indicator 'Perf: 0.02 s/f'. A command line at the bottom shows '>' and 'ac'.



Define Blank (Deformable) Part

Stage Information

Part Type: **Deformables** | Template Part: **N/A** | Name: **Drawing**

Part Name: **Blank**

Material: **1.0338 DC04 0.75-1.2mm Hill** | Thickness: **0.84**

Rolling Direction: Visual | Hardening Rule: **Load Curves**

X: **-0.800958** | Yield locus | FLC | Kf(phi)

Y: **0.597553**

Z: **0.03736** | Write Dynain

Adaptivity | SPC | Initial velocity | Coarsen | **Advance**

Loads | Auto-Pos | Annealing | Shell check

Adaptivity

Number: **100** | Min E-Size: **1** | Maximum Level: **6**

Loads

Body force | Visualize | **Pressure** | Visualize

Coordinate system: **global** | Pressure Curve: **global**

Z | Amount: **9.81** | Mask

Cr/Bm | Done

Tool Coord. Sys.: **global**

Symmetry Planes

Add | **Rmv**

Tolerance: **0.01**





Define Die (Rigid body) Part

Stage Information

Part Type: Rigid tools | Template Part: N/A

Part Name: Die

Contact partners:

- Blank, CONTACT 0.07, one, SN-Off, Sticky

Activate Contact

Friction: 0.07

One Way Two Way

SN-Logic Sticky

DOF Motion Loads Rigid BS Auto-Pos

Degrees of freedom: Visualize

Working Coordinate System: global

Translation	Rotation
<input checked="" type="checkbox"/> X	<input checked="" type="checkbox"/> X
<input checked="" type="checkbox"/> Y	<input checked="" type="checkbox"/> Y
<input type="checkbox"/> Z	<input checked="" type="checkbox"/> Z

Prescribed motion: Visualize

Translation	Rotation	Curve type over time
<input type="checkbox"/>	<input type="checkbox"/>	<input type="radio"/> Displacement
<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="radio"/> Velocity

Name Drawing

- 1:DF:Blank
- 2:RD:Die
- 3:RD:Punch
- 4:RD:Binder stage control

Cr/Bm Done

Tool Coord. Sys. global

Symmetry Planes

Tolerance 0.01





Define Stage Control Data

The screenshot shows the 'Stage Information' dialog box in LS-PrePost. The 'Part Type' is 'Rigid tools' and the 'Template Part' is 'N/A'. The 'Name' is 'Drawing'. The 'Input' field is empty, and the 'Home' directory is 'C:\test\forming\Drawing'. The 'Termination' section has 'Time' set to 'Stop at' with a value of 'unch sinusoid'. The 'Displacement' section has 'Stop at' checked with a value of 'Die'. The 'Output' section has a list of values: 1, 2.34439, 2.79823, 3.2, 4.4, 5.6, 6.8. The 'Intervals' section has 'History' set to 0.01 and 'Restart' set to 0.0. The 'Control Cards' section has 'Explicit Control' selected. The 'Available' list includes CONTROL_DYNAMIC_RELAXATIO, DAMPING_FREQUENCY_RANGE, DAMPING_PART_MASS, DAMPING_PART_STIFFNESS, and DAMPING_RELATIVE. The 'Used' list includes CONTROL_ACCURACY, CONTROL_ENERGY, CONTROL_CONTACT, CONTROL_HOURLASS, CONTROL_OUTPUT, CONTROL_SHELL, and DAMPING_GLOBAL. The 'Symmetry Planes' section has 'Tolerance' set to 0.01.





Curve Dialog

- Load Curve Creation/Edition/Selection Dialog

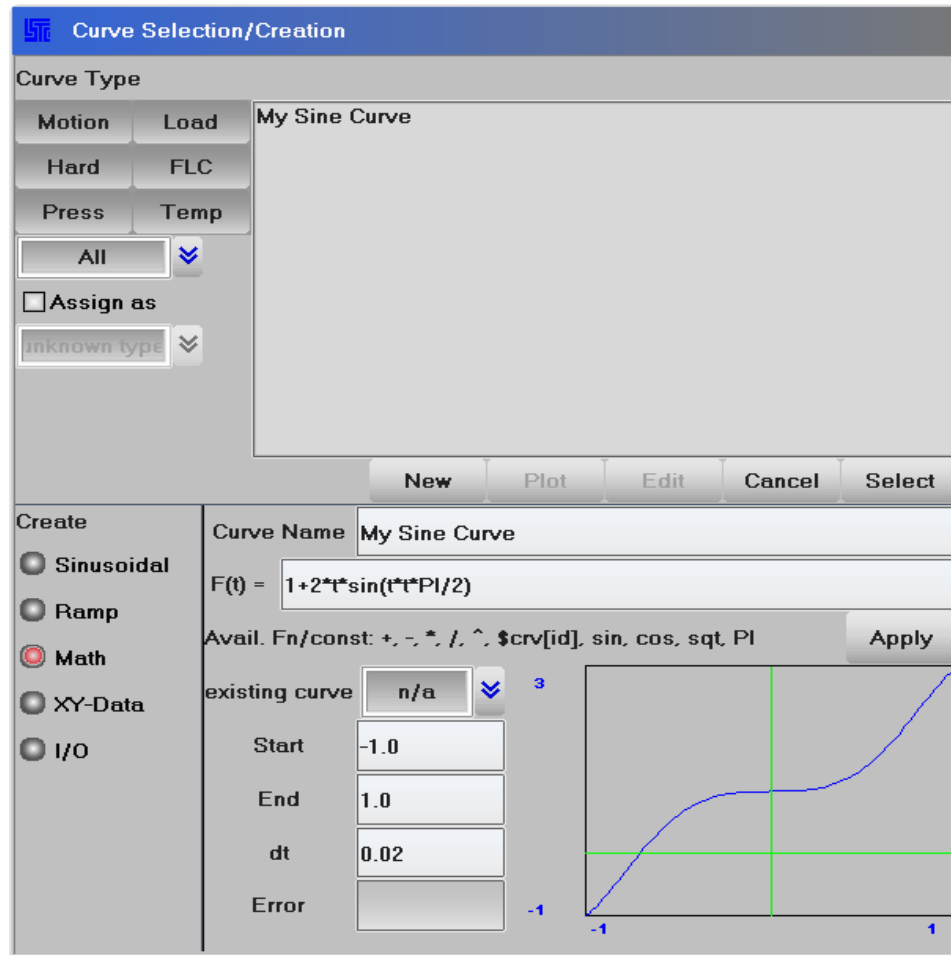
Load curves used in stamping process can be created, edit and/or selected here, then applied to certain fields of the user interface.

- Load Curve creation method includes –

- Sinusoidal: A curve that maps the keyword `*DEFINE_CURVE_SMOOTH` will be created
- Ramp: a ramp-shaped curve, that will increase linearly to a maximum ordinate value and then keep the curve with a constant value until termination time reached.
- Math: regular mathematical expressions are accepted, user can even select existing curves and associate them together with mathematical operators. (i.e. $\$scr1 + \$scr2 * t * \sqrt{t}$) Defined curves will be discredited as requested by users.
- XY-Data: User input (X,Y) data pairs, preview panel shows the current curve defined.
- I/O: Import/Export curves from files with `*DEFINE_CURVE` keyword to/from the current model.



Load Curve Dialog



Load Curve dialog accepts mathematical expressions like the one below. It provides a preview for the curve as well.

$$f(t) = 1 + 2t \sin\left(\frac{\pi}{2} t^2\right)$$

$$-1 \leq t \leq 1$$

$$dt = 0.02$$



Airbag Folding

To create fold definition and fold an airbag

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

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

Save	All
Load	Clear
Cut	Delete

Airbag Folding

Create Modify

>Fold Definition

ID: Title: Show

Fold Type: ▾

N1-N2 Pt+Dir

N1 N2

Active:

Up Down

Thick:

Scale:

Ignore

Ignore move

Anim Loop

Apply Fold 1Fold Line

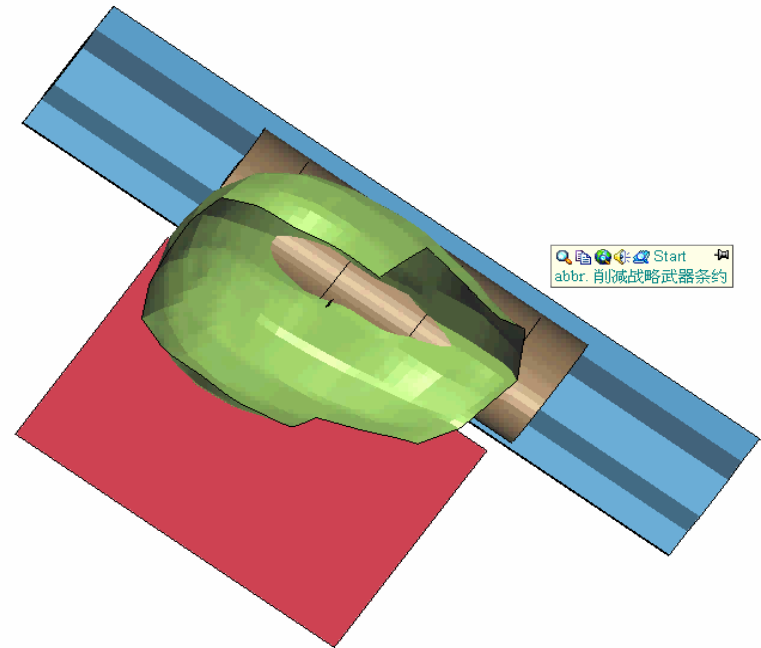
Step Reset Unfold Done





Head Impact Positioning

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



Bcolr	Mcolr	Frin	Isos	Lcon	Acen	Zin	+10	Rx	Deoff	Spant	Top	Front	Right	Redw	Home
Feat	Edge	Grid	Mesh	Shrn	Pcen	Zout	//	Clp	All	Spant	Bottom	Back	Left	Anim	Reset
Horizontal Angle Increment:		10	10	30	180		30	Initial							
Vertical Angle Increment:		10	2	-10	10		-10	Apply							
Velocity: VX		2.49711e-042	VY	2.34003e-041	VZ	2.50132e-042	Generate								
										Original					





Occupant – Dummy Positioning



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



Occupant – DmyPos Basic Operations

LSTC LS-PREPOST Ver 1.0 - 22APR2004(0707)

File Misc. Toggle Background Applications Target Help

Hybrid III 50th% Rigidize Dummy Mod

ABFold DmyPos BeltFit
Guide
TiedNF BinderW
XVect Vector Drawbeac
IniVel Accels DBHist
SpWeld Spc Wall
Box Rivet GWeld
Coord Constn CNRB
SetD PartD MassD

1 2 3 4 5 6 7 D

»Dummy Positioning
 Sel Del Cre
Read Write Reset Load
Done CDmy Occ#1 1

»Positioner interface
 H-Point operations
 Limb operations
Save Load Set Reset

»Limbs operations
 Orient Limb
 Joint settings

- [*]torso
- [*]upper_leg_left
- [*]lower_leg_left
- [*]foot_left
- [*]upper_leg_right
- [*]lower_leg_right
- [*]foot_right
- [*]upper_arm_left
- [*]lower_arm_left
- [*]hand_left
- [*]upper_arm_right
- [*]lower_arm_right

Rot. Ang 15.18 about 0. 1.
Joint ID Lock children
CJSID:12 Unlock all joints
Undo limb Reset limb

occpos resettall





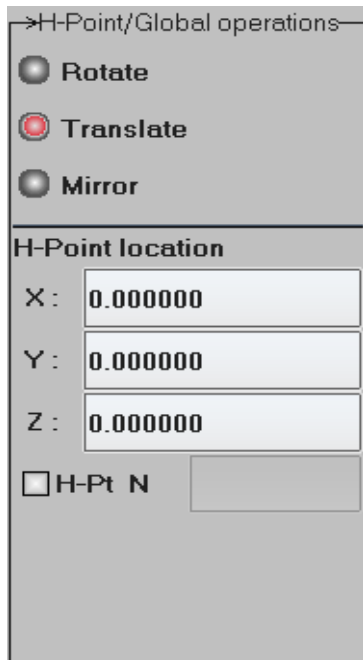
Occupant – DmyPos Basic Operations

•Importing Occupants

- By central lists
- By keyword and tree files

•H-Point/Limb Operations

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





Occupant – DmyPos Advance Topics

•Multiple Occupants

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

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

•Tree files

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

•Angle Files

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

•Stop Angles

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



IIHS

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

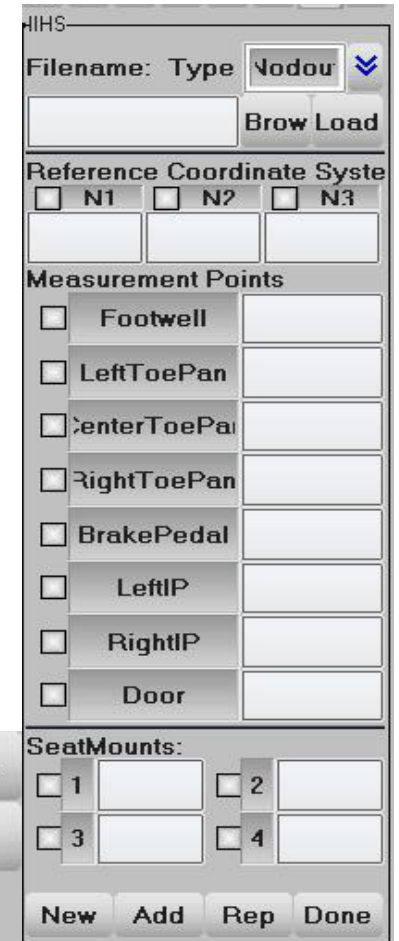
The screenshot shows the IIHS software interface. At the top, it says 'IIHS'. Below that, there is a 'Filename: Type' dropdown menu with 'Nodout' selected. To the right of this dropdown are buttons for 'Nodout', 'IN', and 'OUT'. Below the filename field, there are checkboxes for 'N1' and 'N2'. The 'Measurement Points' section contains a list of checkboxes and input fields for: Footwell, LeftToePan, CenterToePan, RightToePan, BrakePedal, LeftIP, RightIP, and Door. The 'SeatMounts' section contains checkboxes and input fields for 1, 2, 3, and 4. At the bottom, there are buttons for 'New', 'Add', 'Rep', and 'Done'.



IIHS

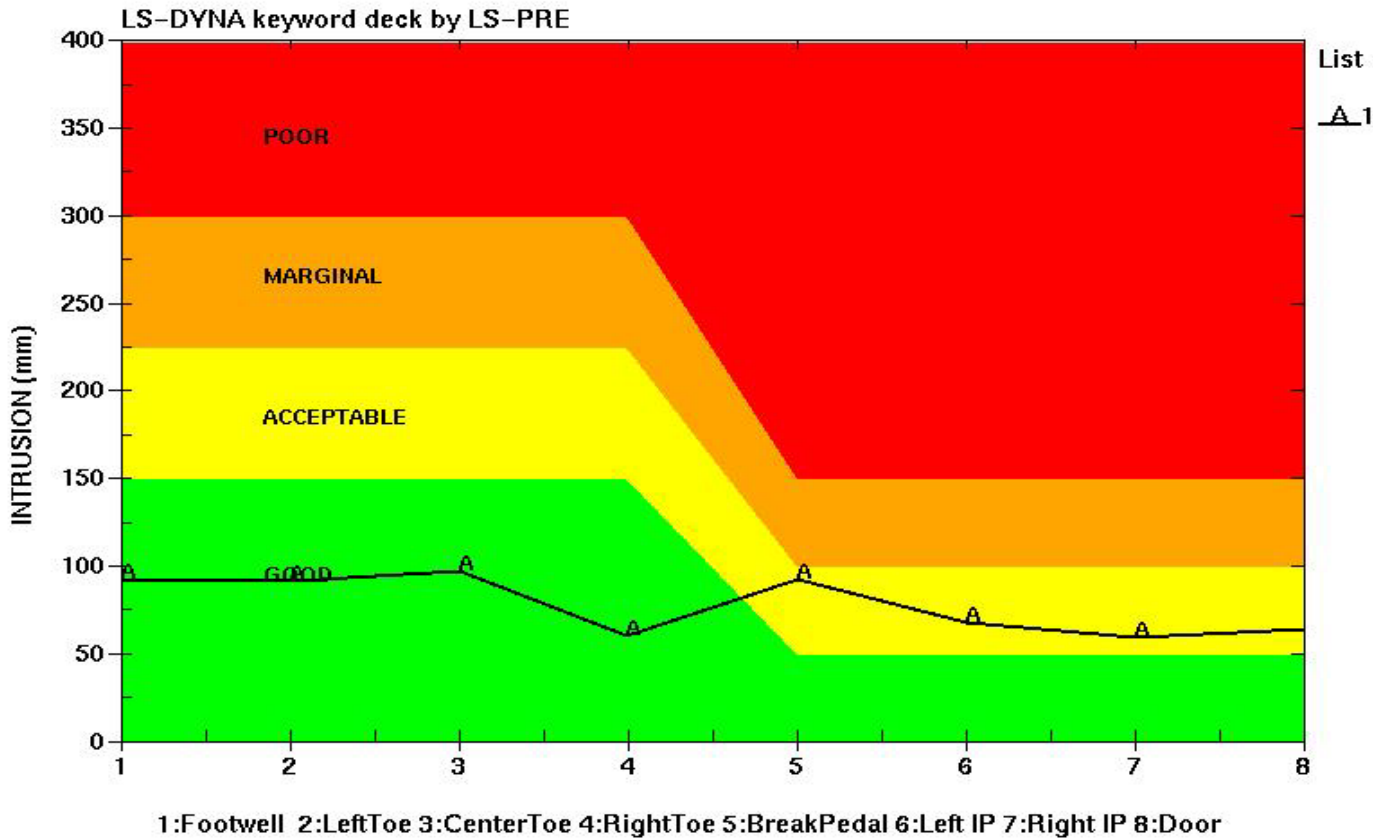
- Use this interface to get IIHS intrusion plot
- Filename: Type Nodout/IN/OUT
- Once the appropriate file is read in, you can pick the nodes from graphical interface or Key in the node numbers assigned to definition of Reference plane and points to be used to plot the intrusion chart.
- [NEW] New IIHS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

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





IIHS



List
A.1

IIHS

Filename: Type **Nodou**

nodout **Brow Load**

Reference Coordinate System

N1 N2 N3

123 345 567

Measurement Points

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

SeatMounts:

- 1 5656 2 6546
- 3 3456 4 5764

New Add Rep Done

Plot Delete

Aplot Write

Modify



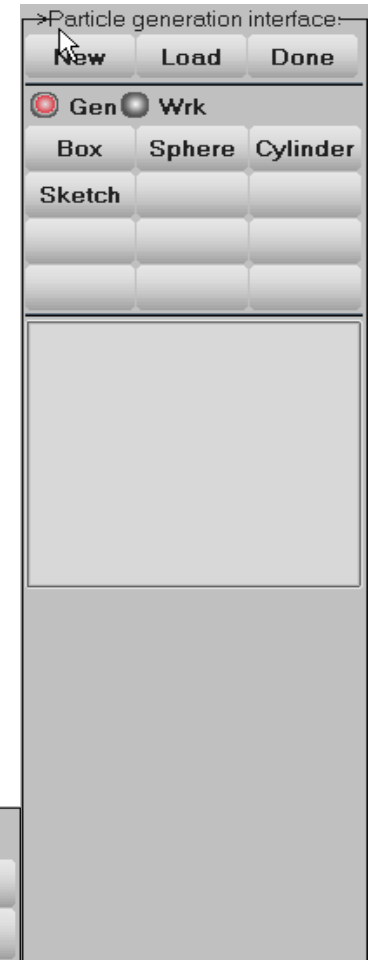


SPH - SphGen

Generating SPH particles in simple geometries.
Current available geometries are

- Box
- Sphere
- Cylinder

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



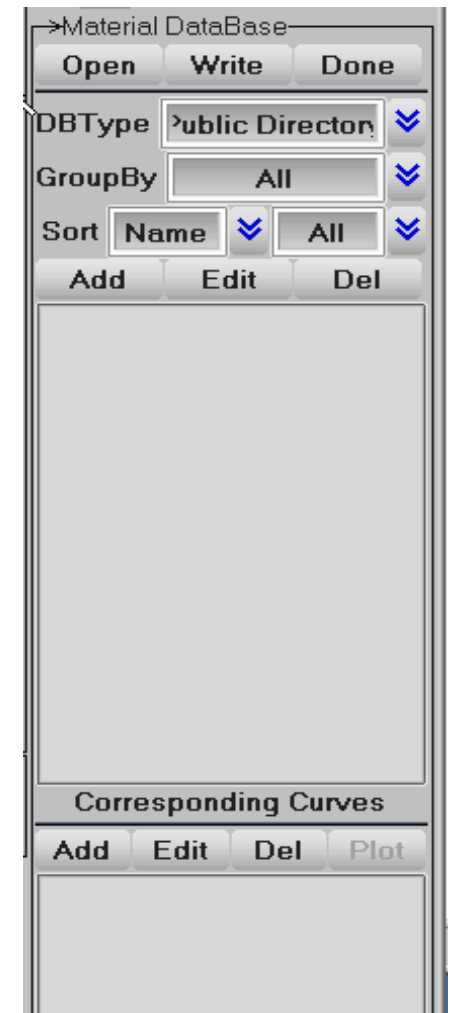
X1: 0.000000	X2: 1.000000	Nx: 10	Box name	BOX01	<input type="checkbox"/> Interferred remove
Y1: 0.000000	Y2: 1.000000	Ny: 10	Part ID	0	
Z1: 0.000000	Z2: 1.000000	Nz: 10	Mat ID	1	>>

Apply
Done



Material Database

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





Configuration File

File name - .lspostrc

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