Keyboard and Mouse Operation

Dynamic Model Operation

Drawing Area Selection

Lists and Multiple Selections

Drop Down Menus

File

Open

Print dialog
  Quick Guides
  Quick Guide: Printing drawing area to a file
  Quick Guide: Printing drawing area to paper

Movie Dialog
  Quick Guides
  Quick Guide: How to create a movie

Misc.

Toggle

Background

Help

Command File (Cfile) Dialog
  Introduction
  Dialog Description
  Quick Guides
  Quick Guide: How to record a command file
  Quick Guide: How to replay a command file

Ruler
  Introduction
  Dialog Description
  Quick Guides
  Quick Guide: How to use the ruler

Assign Menu Buttons
  Introduction
  Dialog Description
  Quick Guides
  Quick Guide: How to Assign menu buttons
  Quick Guide: How to clear menu button assignments

Command Interface

Animation Control

Rendering Hot Buttons

Main Menu

Page 1:Post Processing
  Introduction
Follow Options
Introduction
Interface Description
Quick Guides
  Quick Guide: How to follow a node
  Quick Guide: How to follow a plane

Formatted Output
Introduction
Interface Description
Quick Guides
  Quick Guide: How to write an output data file

Annotation
Introduction
Interface Description
  Quick Guide: How to add a label with an arrow

Section Plane
Introduction
Interface Description
Quick Guides
  Quick Guide: How to define a plane using 1p+NL
  Quick Guides
    Quick Guide: How to define a plane using N1-N2
    Quick Guide: How to define a plane using 3Nds
    Quick Guide: How to define a plane using 2Nds+D
Force: Section force plot interface
Quick Guides
  Quick Guide: How to display a force plot
MVP: Moving Section Plane Interface
Quick Guides
  Quick Guide: How to setup a moving plane
  Quick Guide: How to store and view multiple section cuts
Options: Section Plane Options
File Writing
Quick Guides
  Quick Guide: How to output section cut data
Crush: Intrusion Plotting
Quick Guides
  Quick Guide: How to display an intrusion plot
Line: Cut Line Plot interface
Quick Guides
  Quick Guide: How to plot Cut Line data

Set Fringe Range
Introduction
Interface Description
Fringe Color Palette
Quick Guides
  Quick Guide: How to set a fringe range
  Quick Guide: How to set the fringe color palette
  Quick Guide: How to use Show
  Quick Guide: How to show active minima and maxima
Find Ele/Node/Mat
Introduction

Quick Guides:
- How to show active minima and maxima
- How to use Show
- How to set the fringe color palette
- How to set a fringe range
- How to display an intrusion plot
- How to output section cut data
- How to define a plane using 1p+NL
- How to define a plane using N1-N2
- How to define a plane using 3Nds
- How to define a plane using 2Nds+D
- How to display a force plot
- How to follow a node
- How to follow a plane
- How to write an output data file
- How to add a label with an arrow
- How to follow a node
- How to follow a plane

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Identify Node/Element/Part</td>
<td>52</td>
</tr>
<tr>
<td>Introduction</td>
<td>52</td>
</tr>
<tr>
<td>Interface Description</td>
<td>52</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>52</td>
</tr>
<tr>
<td>Quick Guide: How to identify nodes/elements/parts</td>
<td>52</td>
</tr>
<tr>
<td>Time History Results</td>
<td>53</td>
</tr>
<tr>
<td>Introduction</td>
<td>53</td>
</tr>
<tr>
<td>Interface Description</td>
<td>53</td>
</tr>
<tr>
<td>Vol-Fail: Volume failure</td>
<td>54</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>54</td>
</tr>
<tr>
<td>Quick Guide: How to display a general time history plot</td>
<td>54</td>
</tr>
<tr>
<td>Quick Guide: How to display an element time history plot</td>
<td>54</td>
</tr>
<tr>
<td>Quick Guide: Using Volume Failure plotting with 1 level and n materials</td>
<td>54</td>
</tr>
<tr>
<td>Quick Guide: Using Volume Failure plotting with n levels and 1 material</td>
<td>54</td>
</tr>
<tr>
<td>Set Part Color</td>
<td>55</td>
</tr>
<tr>
<td>Introduction</td>
<td>55</td>
</tr>
<tr>
<td>Interface Description</td>
<td>55</td>
</tr>
<tr>
<td>Color Options</td>
<td>56</td>
</tr>
<tr>
<td>Transparency Options</td>
<td>56</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>57</td>
</tr>
<tr>
<td>Quick Guide: How to assign a color to a part/option</td>
<td>57</td>
</tr>
<tr>
<td>Quick Guide: How to assign a color to the color map</td>
<td>57</td>
</tr>
<tr>
<td>Quick Guide: How to assign a transparency to a part</td>
<td>57</td>
</tr>
<tr>
<td>Element Blanking</td>
<td>58</td>
</tr>
<tr>
<td>Introduction</td>
<td>58</td>
</tr>
<tr>
<td>Interface Description</td>
<td>58</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>58</td>
</tr>
<tr>
<td>Quick Guide: How to blank elements by using an area/polygon</td>
<td>58</td>
</tr>
<tr>
<td>Quick Guide: How to blank elements using a sphere</td>
<td>58</td>
</tr>
<tr>
<td>Explode Parts</td>
<td>59</td>
</tr>
<tr>
<td>Introduction</td>
<td>59</td>
</tr>
<tr>
<td>Interface Description</td>
<td>59</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>59</td>
</tr>
<tr>
<td>Quick Guide: How to explode a set of parts</td>
<td>59</td>
</tr>
<tr>
<td>Quick Guide: Using Move</td>
<td>59</td>
</tr>
<tr>
<td>Cross Plotting</td>
<td>60</td>
</tr>
<tr>
<td>Introduction</td>
<td>60</td>
</tr>
<tr>
<td>Interface Description</td>
<td>60</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>60</td>
</tr>
<tr>
<td>Quick Guide: Cross Plotting – File</td>
<td>60</td>
</tr>
<tr>
<td>Quick Guide: Cross Plotting – Window</td>
<td>60</td>
</tr>
<tr>
<td>XY-Plot Graphics Windows</td>
<td>61</td>
</tr>
<tr>
<td>Left Menu</td>
<td>61</td>
</tr>
<tr>
<td>Right Menu</td>
<td>62</td>
</tr>
<tr>
<td>Title</td>
<td>62</td>
</tr>
<tr>
<td>Scale</td>
<td>62</td>
</tr>
<tr>
<td>Attributes</td>
<td>63</td>
</tr>
<tr>
<td>Quick Guides</td>
<td>63</td>
</tr>
<tr>
<td>Quick Guide: Setting XY-Graph attributes using Auto Symbol</td>
<td>63</td>
</tr>
<tr>
<td>Quick Guide: Setting XY-Graph attributes manually</td>
<td>63</td>
</tr>
<tr>
<td>Filter</td>
<td>64</td>
</tr>
</tbody>
</table>
Quick Guide: How to delete a view

**Model Selection**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: Using multiple model selection
  - Quick Guide: Solving overlapping models

**Part Selection**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: How to use the Part Selection Interface

**Page 2: Pre/Post Processing**
**Introduction**

**CFD Time History Plot**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: How to load a CFD time history component
  - Quick Guide: How to view a CFD time history plot

**BINOUT**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: How to Load and display data stored in a BINOUT file

**CGAT – Circular Grid Analysis Technique**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: How to form a grid of circles
  - Quick Guide: How to view time history data for a circle

**Skid**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: How to display a skid
  - Quick Guide: How to view time history data for a skid trace

**General Selection Menu**
- Quick Guides
  - Quick Guide: Selecting nodes/elements/parts

**Move or Copy Elements**
- Introduction
- Interface Description
- Quick Guides
  - Quick Guide: How to change an element’s association with a part
  - Quick Guide: How to copy an element and apply an association with a part

**Transform Model**
- Introduction
<table>
<thead>
<tr>
<th>Interface Description</th>
<th>Quick Guides</th>
<th>Quick Guide: How to transform a model</th>
<th>Quick Guide: How to copy and transform a model</th>
</tr>
</thead>
<tbody>
<tr>
<td>88</td>
<td>88</td>
<td>88</td>
<td>88</td>
</tr>
</tbody>
</table>

**Translate Model**

- Introduction: 89
- Interface Description: 89
- Quick Guides: 89
- Quick Guide: How to Translate a model: 89
- Quick Guide: How to Translate a model: 89

**Reflect Model**

- Introduction: 90
- Interface Description: 90
- Quick Guides: 90
- Quick Guide: How to reflect a model: 90
- Quick Guide: How to copy and reflect a model: 90

**Rotate Model**

- Introduction: 91
- Interface Description: 91
- Quick Guides: 91
- Quick Guide: How to rotate a model: 91
- Quick Guide: How to copy and rotate a model: 91

**Project Model**

- Introduction: 92
- Interface Description: 92
- Quick Guides: 92
- Quick Guide: How to project a model: 92
- Quick Guide: How to copy and project a model: 92

**Scale Model**

- Introduction: 93
- Interface Description: 93
- Quick Guides: 93
- Quick Guide: How to scale a model: 93
- Quick Guide: How to copy and scale a model: 93

**Shell/Segment/Tshell Normals**

- Introduction: 94
- Interface Description: 94
- Quick Guides: 94
- Quick Guide: How to display Shell/Segment/Tshell normals: 94

**Page 3: Keyword File Editing**

- Introduction: 95

**Keyword Selection Menu**

- General Keyword Options: 96
- *CONTROL Extra Keyword Options: 96
- *EOS / *HOURGLASS / *MAT Extra Keyword Options: 96
- *MAT Extra Keyword Options: 96

**Ref By Interface**

- 97

**Keyword Input Dialog**

- Quick Guides: 98
Quick Guide: How to create a keyword using Page 3 _______________________________________________________98

SET Data ________________________________________________________________99
  Quick Guides.................................................................................................99
  Quick Guide: Entering Set Data.....................................................................99

Link Keyword Dialog ______________________________________________________100

Pick interface ..................................................................................................100
  Quick Guides.................................................................................................100
  Quick Guide: How to pick data from the model..............................................100

Dynamic Keyword Input Forms________________________________________________101

Define Table ..................................................................................................102
  Quick Guides.................................................................................................102
  Quick Guide: How to use define_table..........................................................102

Entity Draw Option ________________________________________________________103
  Introduction....................................................................................................103
  Interface Description......................................................................................103
  Quick Guides.................................................................................................103
  Quick Guide: How to display an entity from a keyword input form.................103

Page 4: Pre Processing Tools __________________________________________________104

Introduction ..................................................................................................104

Airbag Folding ___________________________________________________________105
  Introduction....................................................................................................105
  Interface Description......................................................................................105
  Quick Guides.................................................................................................106
    Quick Guide: Creating an airbag fold..........................................................106
    Quick Guide: How to simulate all folds.......................................................107
    Quick Guide: How to simulate a single fold.................................................107
    Quick Guide: Animating folds....................................................................107

Dummy Positioning Interface ________________________________________________108
  Introduction....................................................................................................108
  Interface Description......................................................................................108
  Rotation Interface..........................................................................................109
  Translation Interface......................................................................................109
  Mirror Interface............................................................................................110
  Limb Operations interface.............................................................................110
  Limb Orientation interface............................................................................111
  Joint setting interface....................................................................................111
  File Write interface.......................................................................................112
  Quick Guides.................................................................................................112
    Quick Guide: How to load an occupant for dummy positioning..................112
    Quick Guide: How to rotate the occupant about the H-point.......................112
    Quick Guide: How to translate the occupant................................................113
    Quick Guide: How to mirror the occupant....................................................113
    Quick Guide: How to rotate limbs...............................................................113

Entity Operation Interface __________________________________________________114
  Introduction....................................................................................................114
  Interface Description......................................................................................114
  Using Show.....................................................................................................114
  Quick Guides.................................................................................................114
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quick Guide</td>
<td>How to Show entities on the drawing area</td>
<td>114</td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to Blank entities on the drawing area</td>
<td>115</td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to Identify entities on the drawing area</td>
<td>115</td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to use KCard</td>
<td>115</td>
</tr>
<tr>
<td>Using Modify/Delete</td>
<td></td>
<td>116</td>
</tr>
<tr>
<td>Quick Guides</td>
<td></td>
<td>116</td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to Select entities</td>
<td>116</td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to use KCard</td>
<td>116</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>117</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>117</td>
</tr>
<tr>
<td>Coordinate Type Options</td>
<td></td>
<td>117</td>
</tr>
<tr>
<td>Quick Guides</td>
<td></td>
<td>117</td>
</tr>
<tr>
<td>Quick Guide: Different ways of displaying set data</td>
<td>118</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to create a Set</td>
<td>118</td>
</tr>
<tr>
<td>Quick Guide: How to modify existing Set data</td>
<td>118</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to delete existing Set data</td>
<td>118</td>
</tr>
<tr>
<td>Quick Guide: How to write out Set data</td>
<td>119</td>
<td></td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>120</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>120</td>
</tr>
<tr>
<td>Coordinate Type Options</td>
<td></td>
<td>120</td>
</tr>
<tr>
<td>Quick Guides</td>
<td></td>
<td>120</td>
</tr>
<tr>
<td>Quick Guide: Different ways of displaying Box data</td>
<td>121</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to create a Box</td>
<td>121</td>
</tr>
<tr>
<td>Quick Guide: How to modify existing Box data</td>
<td>121</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to delete existing Box data</td>
<td>122</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>123</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>123</td>
</tr>
<tr>
<td>Coordinate Type Options</td>
<td></td>
<td>123</td>
</tr>
<tr>
<td>Quick Guides</td>
<td></td>
<td>123</td>
</tr>
<tr>
<td>Quick Guide: Different ways of displaying Coordinate data</td>
<td>124</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to create Coordinate data</td>
<td>124</td>
</tr>
<tr>
<td>Quick Guide: How to modify existing Coordinate data</td>
<td>124</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to delete existing Coordinate data</td>
<td>125</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>126</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>126</td>
</tr>
<tr>
<td>Coordinate Type Options</td>
<td></td>
<td>126</td>
</tr>
<tr>
<td>Quick Guides</td>
<td></td>
<td>126</td>
</tr>
<tr>
<td>Quick Guide: Different ways of displaying Constrained Generalized Weld Spot data</td>
<td>127</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to create a Constrained Generalized Weld Spot</td>
<td>127</td>
</tr>
<tr>
<td>Quick Guide: How to modify existing Constrained Generalized Weld Spot data</td>
<td>127</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to delete existing Constrained Generalized Weld Spot data</td>
<td>127</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>128</td>
</tr>
<tr>
<td>Introduction</td>
<td></td>
<td>128</td>
</tr>
<tr>
<td>Coordinate Type Options</td>
<td></td>
<td>128</td>
</tr>
<tr>
<td>Quick Guides</td>
<td></td>
<td>128</td>
</tr>
<tr>
<td>Quick Guide: Different ways of displaying Constrained Spot Weld data</td>
<td>128</td>
<td></td>
</tr>
<tr>
<td>Quick Guide</td>
<td>How to create a Constrained Spot Weld</td>
<td>129</td>
</tr>
<tr>
<td>Quick Guide: How to modify existing Constrained Spot Weld data</td>
<td>129</td>
<td></td>
</tr>
<tr>
<td>*Constrained_Spot_Weld Data (SpWeld)</td>
<td></td>
<td>130</td>
</tr>
</tbody>
</table>
Quick Guide: How to create a Constrained Node Set  
Quick Guide: How to modify existing Constrained Node Set data  
Quick Guide: How to delete existing Constrained Node Set data  

Database History Data  
Introduction  
Interface Description  
Quick Guides  
Quick Guide: Different ways of displaying Database History data  
Quick Guide: How to create Database History data  
Quick Guide: How to modify existing Database History data  
Quick Guide: How to delete existing Database History data  

Mass Data  
Introduction  
Interface Description  
Quick Guides  
Quick Guide: How to create Mass nodes  
Quick Guide: How to modify Mass nodes  

Part Data  
Introduction  
Interface Description  
Quick Guides  
Quick Guide: How to create Parts  
Quick Guide: How to modify Parts  
Quick Guide: How to use Search  
Quick Guide: How to assign new keyword values to multiple parts  

Page D: Display Interface  
Introduction  
Interface Description  
Boundary  
Constrained  
Contact  
Define  
Initial  
Load  
Set  
Seat Belt  
Rigid Wall  
Quick Guides  
Quick Guide: Displaying entities using Page D  

Appendix I  
Configuration File Options  
Introduction  
List of Configuration File Options  

8/27/2002 BETA
KEYBOARD AND MOUSE OPERATION

DYNAMIC MODEL OPERATION

Rotation – Left mouse button + shift key
Translation – Middle mouse button + shift key
Scaling – Right mouse button + shift key.

Fast rendering mode – Use Ctrl key instead of shift key, this will render the model in edge mode.

DRAWING AREA SELECTION

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

LISTS AND MULTIPLE SELECTIONS

If multiple selections are allowed then the following will apply.

1. Press and hold Ctrl key
2. Make multiple selections
DROP DOWN MENUS

FILE

New – Close all currently loaded models and prepare to load a new one
Open – File chooser to select and open files
Update – while LSDYNA is outputting d3plot files u can update
Save Keyword – Write pre processor data to a keyword file (*.k)
Save Config – Write configuration file (.lspostrc)
Print – Activate print dialog (see below for further details)
Movie – Activate movie creation dialog (see below for further details)
Exit – Exit LS Pre/Post v1.0
Save_Exit – Save current data and Exit LS Pre/Post v1.0

OPEN

Binary Plot – Open a binary plot file for post analysis
Time History – Open a time history data file
Command File – Open a (.Cfile)
Database File – Open a (.db) file
Message File – Open a (.msg) file
Crack – Open a crack file
Config File – Open a configuration file
FLD Curve – Open a FLD curve data file
Labels – Open a labels file
Xydata – Open a XY-Plot data file
Background – Load a background image
Keyword – Load a (.k) file
Interface Force – Open LS-DYNA interface force file
Nastran File – Open a Nastran file
PRINT DIALOG

Name – Select printer
Properties – Select printer settings
Print to file – Select if data is to be outputted to a file
File Name – Input file name
File Type – Select file format
Compression – Select file compression format
Print from – Select data to print

Quick Guides

Quick Guide: Printing drawing area to a file
1. Set model in position in drawing area
2. Open Printing Dialog
3. Select Print to file
4. Set background color
5. Enter file name
6. Select image format
7. Select compression format
8. Click [OK]

Quick Guide: Printing drawing area to paper
1. Set model in position in Drawing area
2. Open Printing Dialog
3. Select printer (note a printer must already be setup with the pc)
4. Modify printing properties
5. Deselect Print to File
6. Click [OK]
**MOVIE DIALOG**

- Format – Select movie file format
- Repeat – Set number times to repeat animation in movie
- Use Playback File – Use command_record data file during movie generation
- Size – Select movie resolution
- Window size – Enter movie resolution
- File Name – Enter movie filename
- Gamma – Enter gamma correction

**Quick Guides**

**Quick Guide: How to create a movie**

1. Open a model
2. Select movie file format
3. Select number of times to record animation
4. Check Use Playback to generate movie of a pre recorded set of commands. (See Command File for details, note the command history must be recorded using the options under the Misc. menu)
5. Select movie format and size or enter a custom size
6. Enter file name
7. Enter gamma correction
8. Click [Start]

**MISC.**

- Reflect – Reflect model about a global plane
- Model Info – Give model size information
- Swap byte on Title – Show title in a different byte order
- Mesh line width – Specify mesh line width in pixels
- Edge line width – Specify edge line width in pixels
- Feature angle – Select feature line angle (if the angle between 2 elements is greater than the feature line angle a line will drawn.
- Start record – Begin recording command file data
- Stop record – Stop recording command file data
- Playback – Playback recently recorded command file data (command_record in working directory)
- Ruler – Opens ruler dialog (see below for further details)
- Command file – Open command file dialog
- Title – Modify model title
- Assign Menu Button – Open assign menu button dialog (see below for further details)
- View Message Dialog – Open keyword file read error message dialog

---

**Movie Dialog**

- **Format:** AVI/RLE
- **Repeat:** 1
- **Use Playback File**
- **Size:** NTSC (640x480)
- **Window size:** (WxH): 640x480
- **File Name:** image_1
- **Gamma:** (0.5-2.0): 1.0

**Misc.**

- Reflect
- Model Info
- Swap byte on Title
- Mesh linewidth
- Edge linewidth
- Feature angle
- Start record
- Stop record
- Playback
- Ruler
- Command file
- Title
- Assign Menu Button
- View Message Dialog
DROP DOWN MENUS

TOGGLE

In the following the first option is the default option.
The setting displayed in the menu indicates the next setting not the current setting.
Local axes – off/on
Texture – switch texture mode off/on (only applicable when in Fringe mode)
Lighting – Two-sided/One-sided
Fringe Light – off/on When this option is off the model will be in fringe color mode.
   (Lighting effects are disabled)
Outlining – Edge/feature/off, select outline type.
PlotUnode – Toggle plot un-referenced nodes on/off
Highlight – Toggle highlights off/on
Result On Screen – Toggle Show results on screen on/off (only applicable if show results is selected under Identify interface)
Shade – Flat/smooth
Outline – Select outline color black/white
Deleted Nodes – on/off
Record Message – off/on When on all messages displayed in the command window will be saved to file, Default file = lspost.msg
Beam Style – Line/Prism
Animate mode – Loop/Once
Deleted elements – off/on

BACKGROUND

Full – Show drawing area in full screen mode
Plain – Set single color background (Select color from color interface)
Fade – Set two color faded background (Select color from color interface)
DiaFade – Set two color double faded background (Select color from color interface)
TriFade – Set three color faded background (Select color from color interface)
Picture – Set jpeg as background (Use File/Open to import jpeg image first)

HELP

Help – Not implemented
Label – Toggle floating help bubbles off/on.
COMMAND FILE (CFILE) DIALOG

Introduction

Accessible via the Misc. or File/Open drop down menu. A Cfile is a log of every command executed during a program session. This log can be viewed using a normal text editor, or replayed within LS Pre Post. CFiles are saved in the same directory as the program executable.

Dialog Description

Pause – Time delay between displaying commands
Loop continuously – Loop Cfile playback
Open – Use to open a Cfile if the original one is closed
Start – Start playback
Stop – Stop playback
Step – Step through a single command at a time
Rewind – Return to the beginning of the command file
Close – Close current Cfile
Done – Exit Cfile Dialog

Quick Guides

Quick Guide: How to record a command file
1. Click Misc./Start recording
2. Proceed to carry out commands to record
3. Click Misc./Stop recording
4. The recorded commands are saved as ‘command_record’ in the same directory as the current model.
Or Copy/Rename lspost.cfile found in the same directory as the program executable. Note this file contains command data for the complete session.

Quick Guide: How to replay a command file
1. Click Misc./Playback to replay a recently recorded command file
Or Click File/Open/Command File, Select the Cfile and click [Open]
2. Select pause time between each command
3. Click [Start] to begin replay
RULER

Introduction
Display ruler(s) in drawing area.

Dialog Description
- Ruler – Select ruler type
- Line Width – Select ruler line width
- Color – Select ruler line color
- Horizontal Ruler – Adjust horizontal ruler position
- Vertical Ruler – Adjust vertical ruler position
- [Reset] – Reset rulers to default settings
- [Close] – Close ruler dialog
- [Quit] – Close ruler dialog and remove ruler(s)

Quick Guides

Quick Guide: How to use the ruler
1. Select Misc./Ruler to open dialog
2. Select rulers to display
3. Select ruler line width
4. Select ruler line color
5. Position ruler using slider
6. Click close to close dialog and keep the rulers
   Or
7. Click Quit to close the dialog and remove the rulers
   (Note: A model must be open to use rulers)
ASSIGN MENU BUTTONS

Introduction
This dialog allows the manipulation of interface buttons on the right hand menu.

Dialog Description
List of available interfaces
[Save] – Save button assignments to .lspostrc file
[Reset] – Reset main menu to default button assignments
[Clear] – Clear all assignments on selected page
[Done] – Close dialog (Note any changes made without saving will remain for the current session only)

Quick Guides

Quick Guide: How to Assign menu buttons
1. Open assign menu button dialog
2. Select button for assignment (does not have to be empty)
3. Select interface from the list to assign to the selected button
4. Click [Save] to save changes in .lspostrc file

Quick Guide: How to clear menu button assignments
1. Open assign menu button dialog
2. Select button for clearing
3. Click [Clear]
COMMAND INTERFACE

1. Finished reading model

open d3plot "D:\models\draw\d3plot"
Reading binary plot files
Finished reading model

Last Command Text Field– The last command issued is displayed here (1)
Previous command list – All issued commands are shown in this list. Also, if a command is issued with missing criteria an error message may be displayed here. (2)
(Left click in the Last Command Text Field to open, left click again to close.)

> 

Command line input – Commands may be entered here when a help comment is not being displayed.

Auto center the model to the window

Help Comments – These are displayed in the command line input area when the mouse cursor is placed over an interface. Further help may be shown in the previous commands list.
**Animation Control**

First – Enter initial state to display
Last – Enter final state to display
Inc – Enter number of states to increment per displayed frame

- **Animate Forwards and Backwards**
- **Stop Animation**

Increment Animation Forwards and Backwards

- **Reverse animation direction**
- **Slider**

Speed Up
Animation Speed Control
Slow Down

- **Enter Displacement Scale Factor**
Number of intermediate steps between states
(For EIGEN value analysis this is the number of steps in each mode)


**Rendering Hot Buttons**

<table>
<thead>
<tr>
<th>Title</th>
<th>Legd</th>
<th>Tims</th>
<th>Triad</th>
<th>Bcolr</th>
<th>Mcolr</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hide</td>
<td>Shad</td>
<td>View</td>
<td>Wire</td>
<td>Feat</td>
<td>Edge</td>
</tr>
</tbody>
</table>

- **Title** – Toggle showing title on/off
- **Legd** – Toggle showing legend on/off
- **Tims** – Toggle showing time stamp on/off
- **Triad** – Toggle showing triad on/off
- **Bcolr** – Toggle background color black/white
- **Mcolr** – Toggle mesh line color black/white

- **Frin** – Color fringe contour
- **Isos** – Iso-surface plot, (applicable to 3D solid elements only)
- **Lcon** – Color line contour

- **Hide** – Display model with hidden line removed
- **Shad** – Display model in color shading mode
- **View** – Display model in plain color mode
- **Wire** – Display model as a wire frame
- **Feat** – Display model in feature line mode (default angle = 30 deg)
- **Acen** – Automatically center model to fit within window
- **Pcen** – Pick a node to be the new center point for model rotation
- **Zin** – Zoom in, draw a box using the left mouse button and dragging
- **Zout** – Zoom out to previous zoom position

- **+10** – Rotate 10 degrees about the global X, Y, Z axes respectively. Right click on Rx, Ry, Rz to switch rotation axis.

- **Parallel view enabled,** click to switch to perspective view
- **Perspective view enabled,** click to switch to parallel view
### RENDERING HOT BUTTONS

<table>
<thead>
<tr>
<th>DeOff</th>
<th>DeOn</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeOff – Turn off all displayed entities</td>
<td>DeOn – Turn on all previously displayed entities</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Clp</th>
<th>All</th>
<th>Rpat</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clp – Clear all picked or highlighted information</td>
<td>All – Restore all entities to be active</td>
<td>Rpat – Restore the last removed part</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Top</th>
<th>Front</th>
<th>Right</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bottom</td>
<td>Back</td>
<td>Left</td>
</tr>
<tr>
<td>Switch to labeled view</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Redw</th>
<th>Home</th>
<th>Anim</th>
<th>Rset</th>
</tr>
</thead>
<tbody>
<tr>
<td>Redw – Re-draw current model</td>
<td>Home – set model in home position</td>
<td>Anim – Start or Stop Animation</td>
<td>Rset – Restore model to original position and state</td>
</tr>
</tbody>
</table>
**MAIN MENU**

Select page to display different interface selections

Interface Selection Area
Click on a tab to display interface in the dynamic interface area.

Dynamic Interface Area
**INTRODUCTION**

The interfaces found on this page are for general model manipulation and post processing.

<table>
<thead>
<tr>
<th>Follow</th>
<th>Split</th>
<th>Explod</th>
</tr>
</thead>
<tbody>
<tr>
<td>Output</td>
<td>Trace</td>
<td>Xyplot</td>
</tr>
<tr>
<td>Anno</td>
<td>Light</td>
<td>FLD</td>
</tr>
<tr>
<td>SPlane</td>
<td>Setting</td>
<td>State</td>
</tr>
<tr>
<td>Range</td>
<td>Vector</td>
<td>Measur</td>
</tr>
<tr>
<td>Find</td>
<td>Ident</td>
<td>ASCII</td>
</tr>
<tr>
<td>Fcomp</td>
<td>History</td>
<td>Views</td>
</tr>
<tr>
<td>Appear</td>
<td>Color</td>
<td>Model</td>
</tr>
<tr>
<td>Group</td>
<td>Blank</td>
<td>SelPar</td>
</tr>
</tbody>
</table>
FOLLOW OPTIONS

Introduction

This interface is used in conjunction with model animation. By defining a single point or a plane, this will be displayed in the center of the screen at all times during the model animation.

Interface Description

Follow Point – Pick node to follow
Follow Plane – Pick n1, n2 and n3 to follow a plane

Node 1, 2 and 3 – Enter node number for nodes 1, 2 and 3 respectively
All – Follow plane with regards to whole model
Part – Follow plane with regards to selected parts. Pick/Enter part ID

[Apply] – Apply follow to model view
[Clear] – Clear pick list
[Reset] – Deactivate follow mode and restore model to previous position
[Done] – Exit Follow Options interface

Quick Guides

Quick Guide: How to follow a node
1. Click [Follow] to open interface
2. Select Follow Point
3. Pick/Enter 1 node ID
4. Click [Apply]

Quick Guide: How to follow a plane
1. Click [Follow] to open interface
2. Select Follow Plane
3. Pick/Enter 3 node IDs
4. Click [Apply]
***FORMATTED OUTPUT***

**Introduction**

Use this interface to output model data.

**Interface Description**

Format – Select output style  
Active parts only – Write out active parts and elements only  
Entire Model – Write data for the entire model  

Internal Number – Write elements and nodes using internal numbers  
User Number – Write elements and nodes using user numbers  

Element – Write element connectivity to file  
Node coordinate – Write nodal coordinates to file  
Node displacement – Write nodal displacement to file  
Element results – Write element results to file  
Nodal results – Write nodal results to file  
Shell P-strain – Write shell principal strain to file  

State No: – Enter state sequence to be written (e.g. 1:5:2 would write out data for states 1, 5 and 2 in that sequence)  
Current – Set State No: to current state  
Append – Select to append data to an existing file  
Write – Start writing file  
Done – Exit Formatted output interface  

**Quick Guides**

**Quick Guide: How to write an output data file**

1. Select output format  
2. Select elements and parts for output  
3. Select the ID format  
4. Select additional data output options  
5. Enter state sequence for output  
6. Select Append to append an existing file  
7. Click [Write] to begin data output
ANNOTATION

Introduction
Use this interface to add annotations to the model.

Interface Description
Select label from list (1)
[Add] – Add new label to list
[Clr] – Clear label input field
[Del] – Remove selected label from list

Text – Enter new label to add to list (2)

Position – Position label in drawing area
Move – Move label in drawing area
Select – Select label in drawing area
Arrow – Position label arrow in drawing area

[Del] – Delete selected labels in drawing area
[Dall] – Delete all labels in drawing area
[Set] – Function no longer required
[Font] – Function no longer required

Col – Select label and arrow color
Ang – Select label angle
Pts – Select label font point size
Font – Select basic font for program

[Reset] – Delete and clear all labels
[Done] – Exit Annotation interface
Quick Guide: How to add a label with an arrow

1. Enter new label
2. Click [Add]
3. Select label from list
4. Select label and arrow color
5. Select label angle
6. Select label font size
7. Select label font
8. Select Position from interface
9. Left click in drawing area to position label
10. Select Select from interface
11. Left click on positioned label
12. Select Arrow from interface
13. Left click on an existing label, hold and drag to create the arrow
SECTION PLANE

Introduction

Use this interface to create section planes that can be used to perform section cuts of the model for further analysis.

Interface Description

FixSp – Section Plane is fixed in space
FixMd – Section Plane is fixed to the model

1p+NL – Define plane with 1 point and a normal

BasePt – Define base point by entering x, y, z coordinates
BaseNd – Define base point by picking/entering a node ID

X, Y, Z – Enter respective base point coordinates
Node – Enter node number for base point

Norm X, Y, Z – Enter respective components of normal vectors

[Centroid] – Use centroid of model as base point
[Reset] – Clear base point and normal settings

Quick Guides

Quick Guide: How to define a plane using 1p+NL

1. Enter coordinates
   Or select BaseNd and pick a node from the model
   Or click [Centroid] to use center of model as base point
2. Enter normal vector or use X/Y/Z axes by clicking on the buttons [NormX/Y/Z]
N1-N2 – Define point normal by picking 2 nodes
3Nds – Define point normal by picking 3 nodes
Node 1, 2 and 3 – Enter or pick a respective node number
2Nds+D – Define plane by picking 2 nodes and a direction cosine
Dcos X, Y and Z – Enter respective direction cosine components

[Reset] – Clear all picked nodes

Quick Guides

Quick Guide: How to define a plane using N1-N2
1. Pick 2 nodes from the model
Note: Node 1 is a point on the plane and N1-N2 is a direction vector

Quick Guide: How to define a plane using 3Nds
1. Pick 3 nodes from the model
Note: All 3 nodes will be on the plane

Quick Guide: How to define a plane using 2Nds+D
1. Pick 2 nodes from the model
2. Enter a direction cosine
[Cut] – Cut the section
[Model] – Display normal model
[Force] – Open section force plot interface
[Mvp] – Move section plane along normal direction
[Options] – Other options or section plane
[Crush] – Plot intrusions relative to a plane
[Line] – Open section cut line plot interface
[Done] – Exit Section Plane interface

**Force: Section force plot interface**

Select items for plotting (1)

[Plot] – Plot section variable vs time in current XY-Plot window
[New] – Plot section variable vs time in a new XY-Plot window
[Padd] – Add selected data to current XY-Plot window
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows
[Clear] – Clear selected items in the list

**Quick Guides**

*Quick Guide: How to display a force plot*

1. Define a plane
2. Click [Force]
3. Select a section force
4. Click [Plot]/[New]
MVP: Moving Section Plane Interface

# of Cuts – Enter number of cuts to make in model
Distance – Enter distance to move plane along the normal

Step plane backwards or forwards in direction of the normal
Use slider to move plane in direction of the normal
Keep all Sections – Store all section cuts in memory
Keep all Cuts – Display all section cuts in drawing area
Mp anim – Animate moving plane along the normal
Update Basept – Update base point with current position

Quick Guides

**Quick Guide: How to setup a moving plane**
1. Click [MVP]
2. Select [# of Cuts] and enter number of cuts to make in model
3. Select [Distance] and enter distance to move plane from origin
4. Use slider or arrows to move section plane or click [mp anim] to animate

**Quick Guide: How to store and view multiple section cuts**
1. Setup a moving plane as described above
2. Check Keep all sections (Each section cut made from now on will be stored)
3. Once all cuts have been stored check Display all sections (All currently stored sections will be displayed)
Options: Section Plane Options

ShowPl – Show section plane with model
ShMesh – Show section plane as mesh
Enter grid size for mesh (1)
Outline – Show 3D model outline with section
LnWidth – Set section cut line width
LnColor – Set section cut line color

File Writing

State No. – Enter state number for the section data to be written
Curr State – Write section data for current state
Write – Start writing section data to file

Quick Guides

Quick Guide: How to output section cut data
1. Define a section plane
2. Enter state no. / click [Curr State]
3. Click [Write]
4. Enter filename
5. Click [Save]
Crush: Intrusion Plotting

LnWidth – Select intrusion plotting line width
LnColor – Select intrusion plotting line color

Plot Type – Select history plot type

Node ID – Enter or pick a node ID to compute intrusion
(Hit Enter key to accept an entered node ID)

[Plot] – Plot intrusion line data in current XY-Plot window
[New] – Plot intrusion line data in a new XY-Plot window
[Padd] – Add intrusion line data to current XY-Plot window
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows
[Clear] – Clear picked/entered node IDs

Quick Guides

Quick Guide: How to display an intrusion plot
1. Click [Crush]
2. Select line width/color
3. Select plot type
4. Pick/Enter node IDs
5. Click [Plot]/[New]
Line: Cut Line Plot interface

[Plot] – Plot section distance vs fringe component in current XY-Plot window
[New] – Plot section distance vs fringe component in a new XY-Plot window
[Padd] – Add section distance vs fringe component data to current XY-Plot window
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows
[Flip] – Flip the XY curve in the x-axis

Quick Guides

Quick Guide: How to plot Cut Line data
1. Select a Fringe Component using Fcomp (see Fringe Component for details)
2. Define a section plane and perform a cut
3. Click [Line] to open Cut Line interface
4. Pick a part
5. Click [Plot]/[New]

Selected PartID: 45
Coord at X = 0:
X= 6301.936035
Y= -219.145782
Z= 848.104980

Coord at X = End:
X= 7139.279785
Y= -1382.768921
Z= 848.104980

Plot New Padd
Flip Raise Pop
SET FRINGE RANGE

Introduction

Use this interface to set fringe and Iso-surface ranges.

Interface Description

Dynamic – A set of min/max ranges is computed for each time state
Static – A constant min/max range is computed using all time states
User – Range set by user, enter min/max values below
Show – Show elements within the range entered below

Entire Model – Range computed for entire model
Active Part only – Range computed for active parts only
Active Ele only – Range computed for active elements only

Blank out of range – Blank all elements with values outside the user defined range
No average – Fringe value is not averaged at node
Show active min/max – Show min and max values for current view
Set Isos Range – Set Iso-surface Range values independently
Reverse Palette – Reverse the color palette
Ident Min value – Identify first N minimum values
Ident Max value – Identify first N maximum values

No. of min/max entities – Enter number of min/max values to be identified
Levels – Select or enter number of color in the fringe palette

[Palette] – Open fringe color palette
[Update] – Update range settings
[Done] – Exit Set Fringe Range interface
**Quick Guides**

**Quick Guide: How to set a fringe range**
1. Select range type
   (Min/Max values need to be entered for User and Show)
2. Select entities to be used for fringe viewing
3. Enable/Disable extra options
4. Enter number min/max entities to show in top left
5. Select/Enter number of fringe levels to be used.

**Quick Guide: How to set the fringe color palette**
1. Click [Palette] to open color palette
2. Select Max and use slider to set value
3. Select Min and use slider to set value
4. Select between hue / gray for palette color
5. Click [Apply] to store changes in database (db) file

**Quick Guide: How to use Show**
1. Select show
2. Enter min/max values (Hit enter after each entry to accept it)

**Quick Guide: How to show active minima and maxima**
1. Apply a fringe component to the model
2. Open Fringe Range interface
3. Check Show active min/max
4. Click [Zin]
5. Define area in drawing area
   (Note: to be able to zoom in Show active min/max must be unchecked)
Find Ele/Node/Mat

Introduction

Use this interface to find a node/element/part by entering its unique ID.

Interface Description

Node – Find a node by entering a node ID
Elem – Find an element by entering an element ID
Part – Find a part by entering a part ID

Any – Find elements of any element type
Beam – Find beam elements
Shell – Find shell elements
Solid – Find solid elements
Tshell – Find Tshell elements
SPH node – Find SPH elements

Show only – Show the entity only
Highlight – Highlight the entity

Enter ID (1)
[Find] – Find and display the entity data
[Neighbors] – Find neighboring nodes/elements
[Done] – Exit Find interface
[Un blank All] – Un blank all blanked elements
[Un blank Part] – Un blank part of found elements

Quick Guides

Quick Guide: How to find an entity
1. Select entity type
2. Select entity sub class
3. Select how to display entity
4. Enter entity ID
5. Click [Find]
**FRINGE COMPONENT**

**Introduction**

Use this interface to display fringe component data on the model.

**Interface Description**

- [Stress] – Global Stress/Strain components
- [Ndv] – Nodal Displacement/Velocitv Contour
- [Result] – Stress resultant components
- [Strain] – Logarithmic strain components
- [Misc] – Pressure, Temperature, Thickness, etc.
- [Infin] – Infinitesimal Strains
- [Green] – Green-St. Venant Strains
- [Almans] – Almansi Strain
- [S.Rate] – Strain rates
- [Residue] – Residual elastic strains
- [FLD] – FLD strain components
- [Beam] – Beam fringe components
- [CFD] – Navier-Stokes components
- [D3Mean] – Time-average window fringe components
- [User] – User defined fringe components
- [Apply] – Collect fringe data

- **Fringe result type (MP)**
  - Fringe
  - Iso-surface
  - Line Contour
  - Fringe Iso-surface
  - XFringe – finding the max value across all surface

- **Set shell stress surface position**
  - Low
  - Mid
  - Upper
  - Maximum
  - Integration Point
  - Beam Integration Point

- **Set axes for stress components**
  - Global Axes
  - Local Axes
Quick Guides

Quick Guide: How to display fringe components

1. Select fringe component
2. Select model result type (note ISO Surface can only be used on solid elements)
3. Select shell stress surface position
4. Select axes Global/Local
5. Click [Apply]
**SET PART APPEARANCE**

**Introduction**

Use this interface to change the appearance of selected parts. This information can be saved using the Save/Retrieve Views interface.

**Interface Description**

Pick Part – Assign appearance by part  
Area – Assign appearance to parts inside/outside a user defined area  
Polygon – Assign appearance to parts inside/outside a user defined region

- In – Set parts inside the area/polygon
- Out – Set parts outside the area/polygon
- Off – No change to the part display mode
- Hide – Display part(s) with hidden line removed
- Shad – Display part(s) in shaded mode
- View – Display part(s) in non-shaded color mode
- Wire – Display part(s) in wire frame mode
- Feat – Display part(s) in feature line mode
- Edge – Display part(s) in edge line mode
- Grid – Display part(s) with grid points
- Frin – Display part(s) with fringe
- Mesh – Toggle mesh outlining on/off with shad and fringe
- Shrn – Toggle shrink plot on/off
- Isos – Display part(s) with iso-surfaces
- Lcon – Display part(s) with line-contours

[AllVis] – Assign appearance to all visible parts
[Done] – Exit Set Appearance interface

**Quick Guides**

**Quick Guide: How to set part appearance**

1. Select appearance options
2. Pick parts using pick/area/polygon
   Or click AllVis to apply to all visible parts
GROUP OPERATIONS

Introduction

Use this interface to create and manipulate groups of parts.

Interface Description

Group list – Select created group from list (1)
Group Name – Enter new group name (2)
[Create] – Create a new group from the active set
[Delete] – Delete selected group from the list
[Rename] – Re-name a selected group
[Select] – Make the selected group the active set
[Add] – Add the selected group to the active set
[Sub] – Subtract the selected group from the active set
[Save] – Save group data to a file
[Load] – Load group data from a file
[Done] – Exit Group Operations interface

Quick Guides

Quick Guide: How to create a group
1. Use Part Selection interface [SelPar] to select parts for group
2. Enter a name for the group
3. Click [Create]

Quick Guide: Available group operations
Or + [Add] – All parts from both groups are added together
Or + [Sub] – Parts in the highlighted non-active set are removed from the active set.
And + [Add] – Parts common to both groups are kept
And + [Sub] – Parts common to both groups are removed
Xor + [Add] – Parts not common to both groups are kept
Xor + [Sub] – Parts common to both groups are removed; only uncommon parts from the active group remain.
**SPLIT WINDOW**

**Introduction**

Use this interface to open up to four drawing areas. Each window can be operated independently and can display different models. The windows can also be synchronized by state and all animated together.

**Interface Description**

1x1 – Single drawing area  
1x2 – Split window horizontally (two drawing areas)  
2x1 – Split window vertically (two drawing areas)  
2x2 – Split window horizontally and vertically (four drawing areas)

Select model to display in active drawing area (1)

Tleft – Draw to top left drawing area  
Bleft – Draw to bottom left drawing area  
Tright – Draw to top right drawing area  
Bright – Draw to bottom right drawing area

Draw all areas – Dynamic model operations will affect all drawing areas

[Sync] – Set all drawing areas (models) to state 1  
[Done] – Exit Split Window interface

**Quick Guides**

*Quick Guide: How to use multiple windows*

1. Select number of drawing areas  
2. Select active drawing area using TLeft…BRight  
3. Select model to view in active drawing area  
   (Use File/Open to open more models)

Each active drawing area can be operated upon independently unless Draw to all windows is used.

Use right click on drawing area to make it active
NODE TRACE

Introduction

Use this interface to trace the paths of nodes over time.

Interface Description

Pick Node – Pick nodes to trace
Area – Select nodes inside/outside a user defined area
Polygon – Select nodes inside/outside a user defined region

In – Select nodes inside the area/polygon
Out – Select nodes outside the area/polygon

Key in Node IDs – Enter node IDs to trace
Node label – Turn node label on/off
Line Width – Select trace line width
Line Color – Select trace line color
Starting State – Enter starting state number

[Write Curve] – Write trace curves for selected nodes
[Write Trace] – Write coordinate history of selected nodes
[Clear Trace] – Clear traced entities
[Done] – Exit Node Trace interface

Quick Guides

Quick Guide: How to generate a node trace
1. Select trace line width/color
2. Pick nodes to trace using single pick/area/polygon
3. Animate model

Quick Guide: How to write a node trace curve
1. Select trace line width/color
2. Enter starting state
3. Pick nodes to trace using single pick/area/polygon
4. Click [Write Curve]

Quick Guide: How to write a node trace in text format
1. Select trace line width/color
2. Enter starting state
3. Pick nodes to trace using single pick/area/polygon
4. Click [Write Trace]
**LIGHTING SETUP**

**Introduction**

Use this interface to apply different lighting effects using up to ten independently setup lights.

**Interface Description**

Select light number to define light
On – Turn highlighted light on
Off – Turn highlighted light off

Ambient – Adjust the ambient component of light
Diffuse – Adjust the diffuse component of light
Specular – Adjust the specular component of light

[Reset] – Reset light to default values

RGB – Set RGB values

X, Y, Z-pos – Enter X, Y, Z coordinates for light source MP

**Quick Guides**

**Quick Guide: How to setup model lighting**

1. Select a light
2. Turn it on
3. Select Ambient and adjust sliders
4. Select Diffuse and adjust sliders
5. Select Specular and adjust sliders
6. Enter coordinates for light source (Hit enter to accept each value)
**SET DISPLAY OPTIONS**

**Introduction**

Use this interface to organize personal display preferences.

**Interface Description**

Axes/Surface – Open element axes and shell surface integration points interface
Displ. factor – Open displacement scale factor interface
Reflections – Open global reflections interface
Hic.Csi const. – Open Hic/Csi constants interface
SPH nodes – Open SPH display options interface
Thickness Scale Factor – Open shell thickness scale factor interface

FLD E’Strain – Toggle FLD Strain (Engineering/True)
Local B’Strain – Toggle Local Brick Strain (General Electric)

El Axes – Select element axes for stress results
Surface – Select position in shells for stress results

Select shell integration point
(Only applicable when Surface = IntPt / BeamIPt)

[Aply] – Apply selected options to the model
[Clr] – Clear and picked parts from the list
[Rset] – Reset model to default options
[Done] – Exit Set Display Options interface
Displacement Scale Factor

Whole – Displace whole model
Part – Displace part of model
Displacement Scale Factor – Enter/Select displacement scale factor
X, Y, Z – Toggle directions to apply displacement scale factor

Reflections

Whole – Reflect whole model
Part – Reflect part of model
Reflect about – Toggle planes to reflect about

Head Injury Criteria (Hic) / Chest Severity Index (Csi)

Time Units – Enter/Select Hic time units
Gravity constant – Enter/Select Hic gravity constant
SPH nodes

SPH Radius Scale – Enter/Select SPH radius scale factor
SPH Sphere divs – Enter/Select SPH sphere divisions
Style – Select SPH sphere style
Fixed radius – Keep SPH radius constant

Shell Thickness Scale Factor

Thickness Scale Factor – Enter/Select shell thickness scale factor
Change in Thickness Scale Factor – Enter/Select change in shell thickness scale factor

Fringe Scale Factor

Fringe Scale Factor – Select/Enter fringe scale factor
Vector Plot

Introduction
Display normal vectors for any element in the model.

Interface Description
Select Vector Type
X, Y, Z – Toggle X,Y,Z components of vector

Vector range – Enter minimum and maximum values for vector range
(Press enter to accept values)

Dynamic – A set of min/max ranges is computed for each time state
Static – A constant min/max range is computed using all time states
User – Range set by user, enter min/max values above
Show – Shows elements within the range entered above

SF – Enter scale factor for vector plot
Hidden line vector off – Switch off hidden line for vectors
(If a vector is behind part of the model it will not be shown, check this option
to display the vector)

Whole – Apply vector plot to whole model
Part – Pick parts for vector plot
Area – Define an area for vector plot
El/Node – Pick an element or node for vector plot

[Apply] – Apply vector plot
[Clear] – Clear vector plot
[Done] – Exit Vector Plot interface

Quick Guides

*Quick Guide: How to display a vector plot*
1. Select a vector type
2. Select a direction (if applicable)
3. Select vector range type
(Enter Min/Max values for User and Show)
4. Select/Enter vector arrow scale factor
5. Select entities
6. Click [Apply] to display vector plot
**IDENTIFY NODE/ELEMENT/PART**

**Introduction**

Use this interface to display the ID for any node/element/part in the model.

**Interface Description**

Pick Entity – Pick and identify an entity  
Area – Identify entities inside/outside a user defined area  
Polygon – Identify entities inside/outside a user defined region

In – Identify entities inside the area/polygon  
Out – Identify entities outside the area/polygon

Key in ID – Enter entity IDs to be highlighted  
Key in xyz coord – Enter XYZ coordinates to be highlighted  
(Separate each value by a space/coma)

Node – Identify nodes  
Element – Identify elements  
Part – Identify parts

Show Results – Show Fringe Component results in Command Window

[Unpick last] – Unpick last picked entity  
[Clear Node] – Clear all identified nodes  
[Clear Ele] – Clear all identified elements  
[Clear Part] – Clear all identified parts  
[AllVis] – Identify all visible entities  
[Clear All] – Clear all identified entities  
[Done] – Exit Identify Node/Element/Part interface

**Quick Guides**

*Quick Guide: How to identify nodes/elements/parts*

1. Select entity type to identify  
2. Select picking method and pick entities from model  
   Or enter entity ID / x, y, z coordinates  
   Information will be displayed in Command Window  
   (Note to use Show Results first apply a Fringe Component to the model using the Fcomp interface)
TIME HISTORY RESULTS

Introduction

Use this interface to display and plot data for various data over time.

Interface Description

Global – Select global history plot
Nodal – Select nodal history plot
Element – Select element history plot
Int.pt. – Select element integration point history plot
Material – Select material history plot
R-Nodal – Select relative nodal history plot
Scalar – Select fringed scalar history plot
Vol-fail – Select volume of material failure option

Sum mats – sum materials for material history plot

Select a time history component (1)

Value – Select element values or max/min element value for material
  Elm – Plot data for selected elements
  Max – Plot data for element with max value for selected material
  Min – Plot data for element with min value for selected material

E-type – Select element type for time history plotting
E-axes – Select shell element axes for time history plotting
Surface – Select shell stress surface position

[Plot] – Plot time history data in current XY-Plot window
[New] – Plot time history data in a new XY-Plot window
[Padd] – Add time history data to current XY-Plot window
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows
[Clear] – Clear selected items in list
[Done] – Exit Time History Plot interface
Vol-Fail: Volume failure

Failure levels – Select failure levels from list
Eroded – Add item to failure list
[Add] – Add entered value to the failure levels list
[Clr] – Clear text field and selected items
[Del] – Delete selected item from the failure levels list
Value – Enter failure level value to be added to the list

Quick Guides

Quick Guide: How to display a general time history plot
1. Select a time history type
2. Select time history component
3. Pick nodes for Nodal and R-Nodal/elements for Element and Int. Pt./materials for Materials/select a fringe component for Scalar (Use Fcomp)
4. Click [Plot]/[New]

Quick Guide: How to display an element time history plot
1. Select Element
2. Select element type
3. Select time history component
4. Select element value type (Elm/Max/Min)
5. Select element axes Global/Local
6. Select shell surface stress position
7. Pick elements from model
8. Click [Plot]/[New]

Quick Guide: Using Volume Failure plotting with 1 level and n materials
1. Enter Failure level
2. Click [Add]
3. Pick n materials from drawing area
4. Click [Plot] / [New]

Quick Guide: Using Volume Failure plotting with n levels and 1 material
1. Enter Failure level
2. Click [Add] (repeat to enter multiple levels)
3. Pick 1 material from drawing area
4. Click [Plot] / [New]
**SET PART COLOR**

**Introduction**

Use this interface to apply different colors and transparency levels to selected parts. This information can be saved using the Save/Retrieve Views interface.

**Interface Description**

Pick Part – Assign color/transparency by part
Area – Assign color/transparency to parts inside/outside a user defined area
Polygon – Assign color/transparency to parts inside/outside a user defined region

In – Assign color/transparency to parts inside the area/polygon
Out – Assign color/transparency to parts outside the area/polygon

Color – Set color for parts
Transp – Set transparency for parts

Part Colormap – Color map used to assign colors to parts automatically, can be used as a pallet to assign colors to parts manually
Editmap – Clicking on the color map will assign a chosen color to the color map (toggle on/off)
[Reset] – Reset the color map to default settings

[AllVis] – Assign color/transparency to all visible parts
[Amap] – Apply the current color map to parts
[Rstp] – Reset transparency of all parts to opaque
[Smap] – Save part colors to an ASCII file
[Lmap] – Load part colors from an ASCII file
[Done] – Exit Set Part Color interface
Color Options

RGB – Use sliders/text fields to set values for each color component
Color Chooser – Displays a user defined color or a color picked from a part/color map. (1)
Set – Assign user defined color to part/color map/option
Show – Show picked part/source color in the color chooser

Options
[Backg] – Apply selected color as plain background color
[Mesh] – Apply selected color as mesh color
[Hilite] – Apply selected color as highlight color
[Text] – Apply selected color as text color
[Label] – Apply selected color as label color
[Outlin] – Apply selected color as outline color
[Sky] – Apply selected color as sky color, (for Fade, Dia-fade and Tri-fade backgrounds)
[Ground] – Apply selected color as ground color, (for Fade, Dia-fade and Tri-fade backgrounds)
[Middle] – Apply selected color as middle color, (for Tri-fade background)

Transparency Options

Transparency Factor – Transparency Factor component slider
Set Transparency – Assign transparency to a part
Get Transparency – Show part transparency value on slider
Quick Guides

Quick Guide: How to assign a color to a part/option
1. Select Color
2. Select show
3. Select a color from the color map/generate a color using the sliders
4. Select set
5. Select parts using pick/area/polygon
Or select an option from the lower interface

Quick Guide: How to assign a color to the color map
1. Select Color
2. Select a color from the color map/generate a color using the sliders
3. Check Editmap
4. Select box in color map to assign new color

Quick Guide: How to assign a transparency to a part
1. Select Transp
2. Select Set Transparency
3. Move slider to desired transparency
(Note: The slider will not set a transparency of 0, use Rstp)
4. Select parts using pick/area/polygon
**ELEMENT BLANKING**

**Introduction**

Use this interface to blank selected elements.

**Interface Description**

Pick Element – Blank a single element
Area – Blank elements inside/outside a user defined area
Polygon - Blank elements inside/outside a user defined region
Sphere – Blank elements inside/outside a spherical region
Unblank Part – Pick an element to un-blank the whole part
In – Blank elements inside the area/polygon
Out – Blank elements outside the area/polygon

Center – Enter a node ID or x, y, z coordinates to define the center of the sphere
Radius – Enter radius for spherical blanking
Set Radius – Use slider to set radius for spherical blanking
[Blank Sphere] – Press to perform spherical blank

[Blank All] – Blank all visible elements
[UnBlank All] – Un-blank all elements
[Reverse] – Reverse blanked/un-blanked elements
[Update Surf] – Update the 3D outer surface
[UpdEdge] – Update edge line
[UnBlank Last] – UnBlank last blanked elements
[Done] – Exit Element Blanking interface

Auto Update – Auto update the 3D outer surface
Surface Only – Blank surface of solid elements only

**Quick Guides**

**Quick Guide: How to blank elements by using an area/polygon**
1. Select area/polygon
2. Select In/Out
3. Define area/region
(Note: Elements will be blanked automatically)

**Quick Guide: How to blank elements using a sphere**
1. Select Sphere
2. Pick a node from the model to be the origin
3. Enter a radius/move slider
4. Click blank sphere
EXPLODE PARTS

Introduction

Use this interface to explode/separate parts so that they can be viewed more easily.

Interface Description

Factor – Enter/Select an explode scale factor
Directions – Select directions to explode model
Move – Move exploded parts using mouse pointer
All – Move /explode all parts
Part – Move/explode picked parts only
PtGroup – Move/explode part groups only

Part Groups List – Select part group from list

[Apply] – Explode selected parts
[Reset] – Reset model
[Clear] – Clear picked parts and list selections
[Done] – Exit Explode Parts interface

Quick Guides

Quick Guide: How to explode a set of parts
1. Set scale factor
2. Set direction
3. Select parts to explode
4. Click Apply

Quick Guide: Using Move
1. Toggle Move to activate
2. Left mouse button operates rotation
3. Middle mouse button translates along axis
4. Right mouse button translates 90deg to axis indicated in 3.
**CROSS PLOTTING**

**Introduction**
Control all open XY-Plot windows and files using this interface.

**Interface Description**
File – Show list of XY-Plot data files
Window – Show list of current XY-Plot windows

Select file/window to be shown (1)
Select XY-Plot to show or to cross plot (2)

[Add] – Open and add a XY-Plot data file to the filename list
[Rem] – Remove a XY-Plot data file from the filename list
[Info] – Show full XY-Plot data file path in command window
Show – Show selected plot
Cross – Cross selected plots

[Plot] – Plot XY-Plot data in current XY-Plot window
[New] – Plot XY-Plot data in a new XY-Plot window
[Padd] – Add XY-Plot data to current XY-Plot window
[Clear] – Clear selected items in list
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit Cross Plotting interface

**Quick Guides**

**Quick Guide: Cross Plotting – File**
1. Select File
2. Click Add to open a new XY-Plot data file
3. Select the new file
4. Select multiple stored XY-graphs
5. Select show and click [Plot]/[New] to open
6. Select cross and click [Plot]/[New] to cross graphs

**Quick Guide: Cross Plotting – Window**
1. Select Window
2. Select window from the main list
3. Select a curve that is in that window
4. Click [New] to open curve in a new window
XY-Plot Graphics Windows

[Edit] – Edit XY-Plot Data (Not implemented yet)
[Title] – Set and change titles/axis labels
[Scale] – Set and change offset/scale of curves
[Attr] – Change attributes of XY-Plot
[Filter] – Set curve filtering
[Print] – Open print dialog
[Save] – Write XY-Plot data to a file

[User] – Load XY-Plot data from a file
[Oper] – Process XY-Plot data
[Hide] – Hide buttons (Esc to restore them)
[Close] – Close the plot window
[Quit] – Close and delete plot window from the program memory

Left Menu

[All] – Select all curves in the list
[Rev] – Reverse curve selection
[Clr] – Clear curve selection
[Del] – Remove selected curve from list
Curve List (1)
Page 1: Post Processing

Right Menu

Grid – Add/remove grid lines in XY-Plot window
Tick – Add/remove tick marks in XY-Plot window
Frame – Add/remove frame in XY-Plot window
Legend – Add/remove legend in XY-Plot window
Autofit – Toggle Automatically Fit data in XY-Plot window (on/off)
Timeline – Toggle time line in XY-Plot window
Invert – Invert background color in XY-Plot window
Maxgraph – Use all of XY-Plot window to display data
MinMax – Add/remove min/max label in XY-Plot window

Title

Plot Title:

X-Label:

Y-Label:

Legend Label:

Plot Title – Input plot title
X/Y-Axis labels – Input X/Y axis labels
Modify Curve Legend – Modify curve name in legend (Press enter to update)
Legend Title – Input legend title

[Show All] – Show all curves in XY-Plot window
[Show Select] – Show selected curves in XY-Plot window
[Apply] – Apply settings to XY-Plot window
[Reset] – Reset settings to default values
[Done] – Close the XY-Plot options menu

Scale

Xmin 0 Xmax 0.128998
Ymin 0 Ymax 460.636
X-offset 0 X-scale 1
Y-offset 0 Y-scale 1

Xmin/max – Input min/max values for X-axis.
Ymin/max – Input min/max values for Y-axis.
X-offset – Input offset value for X-axis.
Y-offset – Input offset value for Y-axis.
X-scale – X axis scale factor
Y-scale – Y axis scale factor
Axes – Select logarithmic/linear scale for each axis.

8/27/2002 BETA
Attributes

Pts/Sym – Input number of points per symbol
Join symbols with line – Add/remove lines between symbols when using auto symbol
[Auto Symbol] – Automatically replace curve with symbols
Symbol – Select symbol style (Use [Apply]/[Reset] to make individual changes)
Color – Select curve and symbol color (Use [Apply]/[Reset] to make individual changes)
Width – Select line width (Use [Apply]/[Reset] to make individual changes)
Style – Select line style (Use [Apply]/[Reset] to make individual changes)

Quick Guides

**Quick Guide: Setting XY-Graph attributes using Auto Symbol**
1. Select curves
2. Enter the number of points per symbol
3. Enable/Disable Join symbols with line
4. Click [Auto Symbol] this will add symbols to more than 1 line

**Quick Guide: Setting XY-Graph attributes manually**
1. Select a single curve
2. Select Symbol type, click [Apply] to apply symbol type only
3. Select line color, click [Apply] to apply line color only
4. Select line width, click [Apply] to apply line width only
5. Select style, click [Apply] to apply style only
6. Click [Apply] at bottom to apply all changes
Filter – Select filter to apply
Time – Select time units
C/s(Hz) – Select/Enter frequency

Quick Guides

**Quick Guide: How to apply a filter**

1. Select a curve
2. Select filter type
3. Select time units
4. Select frequency
5. Select point average
6. Click [Apply] to apply all changes

Print

(See Print Dialog information under drop down menus)
Save

Output Type – Select data output type
Output Interval – Select/Enter number of time steps between outputs

Filename - Enter filename including full path with no spaces
[Save] – Write XY-Plot data to given filename

Quick Guides

Quick Guide: Saving XY-Plot data to a file
1. Select curves to output
2. Select output format
3. Select output interval
4. Enter filename (If no path is given file will be saved in the same directory as the executable)
5. Click [Save]

Load

Filename list – Select a file to show stored data.
[Add File] – Add a file to the filename list.
[Remove File] – Remove a file from the filename list.
[Select] – Open selected data files.
Quick Guides

Quick Guide: How to load a XY-Plot data file

1. Click [Add File]
2. Select file and click [Open]
3. Select file in filename list
4. Click [Select]
5. Stored curves should now be visible in the left menu, select curves from here to display them
Operations List – Select an operation to apply to a curve
Curve 1, 2, and 3 – Pick/Enter curves from curve list for operation
Time Units – Select/Enter HIC time units
Gravity Constants – Select/Enter HIC gravity constants
[Save Result:] – Save resultant curve as a XY-Plot data file

Quick Guides

**Quick Guide: Operating on a curve**
1. Select operation from the list
2. Select a curve(s) from the left list
3. Click [Apply]

**Quick Guide: How to subtract/multiply/divide curves**
1. Select appropriate operation
2. Select/Enter curve 1 from left list
3. Select/Enter curve 2 from left list
4. Click [Apply]

**Quick Guide: How to apply resultant2 and 3**
1. Select appropriate operation
2. Select/Enter curve 1 from left list
3. Select/Enter curve 2 from left list
4. Select/Enter curve 3 from left list (resultant 3 only)
5. Click [Apply]

**Quick Guide: How to apply Hic15/Hic36/CSI**
1. Select appropriate operation
2. Select/Enter time
3. Select/Enter gravitation constant
4. Click [Apply]

**Quick Guide: Saving results**
1. Carry out operations as described above
2. Enter a new filename in the text field provided
3. Click [Save Result]
FORMING LIMIT RESULTS

Introduction

Use this interface for metal forming analysis.

Interface Description

t – Enter sheet thickness in mm
n – Enter FLD crit. formula index
[Set] – Apply the new t and n values
t_n – Set thickness and index of FLD curve
File – Read thickness and index of FLD curve from file
FLD Data File list – Select FLD data file,
(use File/Open/FLD Curve to open a new file) (1)

Point – Select an item for point on FLD plot
Tracer – Select an item for tracer on FLD plot

Material – Select a material for FLD plot
Element – Select an element for FLD plot
Area – Define an area for FLD plot.
Polygon – Define a region for FLD plot.
In – Select entities within the area/polygon.
Out – Select entities outside the area/polygon.

[Clr] – Clear information in popup windows.
[Info] – Open/close FLD information dialog.
[FormaDia] – Open/close formability diagram dialog.

Strain results for last element picked (2)

Position – Select shell surface for FLD results

[Plot] – Plot FLD diagram in current XY-Plot window
[New] – Plot FLD diagram data in a new XY-Plot window
[Padd] – Add data to FLD diagram in current XY-Plot window
[Clear] – Clear picked entities
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows

[Done] – Exit Forming Limit Result interface
**Formability Diagram**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Limit % FLC</td>
<td>0</td>
<td>Cracks</td>
</tr>
<tr>
<td>Safety margin % FLC</td>
<td>20</td>
<td>Risk of cracks</td>
</tr>
<tr>
<td>Allowable thinning</td>
<td>0.3</td>
<td>Severe thinning</td>
</tr>
<tr>
<td>Essential thinning</td>
<td>0.02</td>
<td>Inadequate stretch</td>
</tr>
<tr>
<td>Allowable thickening</td>
<td>0.01</td>
<td></td>
</tr>
<tr>
<td>R-value</td>
<td>1.82</td>
<td>Wrinkling tendency</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Wrinkles</td>
</tr>
</tbody>
</table>

**Quick Guides**

**Quick Guide: Displaying Forming Limit Diagram**

1. Enter sheet thickness and FLD criterion (t and n)
   Or select File and open a pre loaded FLD curve file
   (Use File/Open/FLD curve to load a FLD curve file)
2. Click [FormaDia] to set FLD parameters
3. Select a material/element using single or area pick
4. Click [Plot]/[New] to display results

Cracks – Show crack region
Risk of cracks – Show risk of cracks region
Severe thinning – Show excessive thinning region
Inadequate stretch – Show lack of stretch region
Wrinkling tendency – Show tendency to wrinkle region
Wrinkles – Show wrinkled region
[Close] – Close Formability Diagram dialog
STATE TIME

Introduction

Use this interface to activate/deactivate time states and apply overlays to the model.

Interface Description

State List (1)
Choose – Selecting a state from the state list will display it
Delete – Selecting a state from the state list will make it inactive
Turn overlay mode on/off (2)
Overlay State – Enter state number for overlay
Overlay Color – Select overlay color
Overlay Type – Select overlay type
[Reset] – Reset all states to active

[Done] – Exit State Time interface

Quick Guides

Quick Guide: How to use overlays
1. Click on (2) to activate overlay mode
2. Select overlay color
3. Select overlay type
4. Select a state from the list
(Now when using measure differences between the overlay and current state will also be displayed in the command window)
MEASURE

Introduction

Use this interface to make measurements of various aspects of the model and to create local coordinate systems.

Interface Description

- Active Element Only – Measure part using active elements only
- Item – Select an item to measure/Create local coordinate system
- [Cancel pick] – Cancel last picked node (For Angle 3/4Node / Create Axes)
- Element – Display measurement for selected elements
- Part – Display measurement for selected parts
- All – Display measurement for all visible parts
- [Apply] -
- [Delete] – Delete selected local axes
- Reference Axes – Select current reference axes for measurement (1)
- History – Select history type to plot (2)
- [Plot] – Plot History data in current XY-Plot window
- [New] – Plot History data in a new XY-Plot window
- [Padd] – Add History data to current XY-Plot window
- [Clear] – Clear selected items in list
- [Raise] – Raise all open XY-Plot windows
- [Pop] – Open and Raise all closed XY-Plot windows
- [Done] – Exit Measure interface

Quick Guides

**Quick Guide: How to measure a coordinate**

1. Select Reference Axes
2. Select Coordinate
3. Pick 1 Node
4. Coordinate for current state is displayed in command window (lower left)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot

**Quick Guide: How to measure a distance**

1. Select Reference Axes
2. Select Item: Distance / Distance N2S
3. Pick 2 nodes / 1 node and 1 element
4. Distance for current state is displayed in command window (lower left)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot
Quick Guide: How to measure an angle
1. Select Reference Axes
2. Select Angle 3Nodes / Angle 4Nodes
3. Pick 3 nodes / Pick 4 nodes, 2 per line
4. Angle for current state is displayed in command window (lower left)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to measure an area
1. Select Reference Axes
2. Select Item: Area
3. Select Element/Part/All (Click [Apply] if using All)
4. Check Active Elements Only to ignore blanked elements
   (Part/Element/All, area for current state is displayed in command window)
5. Select history type
6. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to measure a volume/mass
1. Select Reference Axes
2. Select Item: Volume/Mass
3. Select Element/Part/All (Click [Apply] if using All)
   (Part/Element/All, volume/mass for current state is displayed in command window)
4. Select history type
5. Click [Plot]/[New] to view time history plot

Quick Guide: How to measure inertia/angular velocity
1. Select Reference Axes
2. Select Item: Inertia/Ang Velocity
3. Select Part/All (Click [Apply] if using All)
   (Part/All, Inertia/Angular velocity for current state displayed in command window)
4. Select history type
5. Click [Plot]/[New]/[PAdd] to view time history plot

Quick Guide: How to define a local coordinate system
1. Select Item: Create Axes
2. Pick 3 nodes to generate local coordinate system
   To delete,
   1. Select local coordinate system to remove
   2. Click [Delete]
ASCII

Introduction

Use this interface to browse and display data contained in various ASCII data files output by DYNA.

Interface Description

File – Load an ASCII file from an alternative directory
Load – Load existing file for selected ASCII file type
Unload – Un-Load selected ASCII file (to free memory)
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows
[Done] – Exit ASCII interface

[Plot] – Plot items from ASCII file in current XY-Plot window
[New] – Plot items from ASCII file in a new XY-Plot window
[Padd] – Add items from ASCII file to current XY-Plot window
[All] – Select all ASCII items
[Clear] – Clear all selections
[Rev] – Reverse selection
[Info] – Show information on the loaded ASCII file

ASCII items list – select items loaded from ASCII file(1)
Main Branch(1)
Sub Branch(2)
{matsum} [Total] – Toggle the combining of multiple ASCII items
Affects {secforc}, {spcforc}, {nodfor}, {bndout}, {sleout} and {gceout}

{abstat} [Airbag/Material] – Select airbag/material IDs
{elout} [Clrcp] – Clear Elout component list
{elout} [Stress/Strain/Beams] – Select Elout component types
{deforc} [Trans/Rotation] – Select translational force /rotational moment of spring element
{spcforc} [Force] – Select forces for plotting
{spcforc} [Momt] – Select moment for plotting
{jntforc} [Jforc] – Select joint forces for plotting
{jntforc} [Jstif] – Select joint stiffness for plotting
{rbdout} [Local] – Plot curve in local coordinate system
{sphout} [Stress/Strain] – Select sphout component types
{nodout} [HicCsi] – Open Hic/Csi options interface

1-Kinetic Energy
2-Internal Energy
3-Total Energy
4-Energy Ratio
5-Spring_Damper Energy
6-Hourglass Energy
7-Damping Energy
8-Sliding Energy
9-External Work
10-Eroded Kinetic Energy
11-Eroded Internal Energy
12-Erg Ratio w/o Erade
13-Stonewall Energy
14-Step Size
Head Injury Criteria (Hic) / Chest Severity Index (Csi) Options

T – Enter/Select time units for Hic/Csi/Filter
Filter – Select filter type for filtering data
G – Enter/Select gravitation constant in model units
Pt Ave – Select number of points for point averaging filter
C/sHz – Enter/Select filtering frequency in model time units
[Done] – Exit Hic/Csi Options interface

Quick Guides

Quick Guide: How to load an ASCII item
1. Select ASCII item to load from list
2. Click [Load]
3. Find ASCII file for that item
4. Click [Open]

Quick Guide: How to plot stored data
1. Load ASCII item
2. Select Main branch
3. Select IDs (If applicable)
4. Select Sub-branch
5. Click [Plot]/[New]/[PAdd]

Quick Guide: How to set Hic/Csi options
1. Load {nodout} ASCII item
2. Click [Hic/Csi]
3. Enter Time for Hic
4. Enter Gravitation Constant for Csi
5. Select filter type
6. Select number of points per average
7. Enter/Select frequency
8. Click done to close interface
SAVE/RETRIEVE VIEWS

Introduction
Use this interface to save and retrieve information regarding model appearance, color and orientation.

Interface Description
List of saved views(1)
View Name_number – Enter a new view name
Color – Toggle Save/Retrieve model color on/off
Appearance – Toggle Save/Retrieve model appearance on/off
Orientation – Toggle Save/Retrieve model orientation on/off
[Save] – Save the current view to memory
[Retrieve] – Retrieve selected view from memory
[Delete] – Delete selected view from memory
[Done] – Exit Save/Retrieve Views interface

Quick Guides

Quick Guide: How to save a view
1. Enter view name
2. Select what to save (Color/Appearance/Orientation)
3. Click save

Quick Guide: How to retrieve a view
1. Select view from list
2. Select what to retrieve (Color/Appearance/Orientation)
3. Click Retrieve

Quick Guide: How to delete a view
1. Select view from list
2. Click delete
MODEL SELECTION

Introduction

Use this interface to open and select multiple models.

Interface Description

Model List – Select model to display (1)
[Delete] – Delete the highlighted model from program memory
[Reset] – Reset model to original position
[Done] – Exit Model Selection interface
[X] – Select direction for model translation
Translation Distance – Enter distance to translate model

Multi Model Selection – Activate multiple models (Note once activated model selection for translation etc can only be done from the Model ID list and not the Model List. The Model List is used to select models for multiple selections)
Sync States – Synchronize animation for multiple models
Translation – Enter amount for translation
Model ID List – Select model for translation when using multi model selection

Quick Guides

Quick Guide: Using multiple model selection
1. Open models using File/Open
2. Activate multiple model selection
3. Select models to be displayed together from the Model list

Quick Guide: Solving overlapping models
1. Select one of the overlapping models(1)
2. Select direction of translation
3. Click left or right arrow to translate model
PART SELECTION

Introduction

Use this interface to select which parts to display.

Interface Description

Beam – Select/deselect Beam elements
Shell – Select/deselect Shell elements
Solid – Select/deselect Solid elements
Tshell – Select/deselect Tshell elements
Mass – Select/deselect Mass elements
Discrete – Select/deselect Discrete elements
Seatbelt – Select/deselect Seatbelt elements
Inertia – Select/deselect Inertia elements
Rsurf – Select/deselect Rigid-surface elements
SPHnd – Select/deselect SPH nodes elements
Fluid – Select/deselect Fluid elements

Single – Remove/Keep parts using single pick
Area – Remove/Keep parts inside a user defined area
Polygon – Remove/Keep parts inside a user defined region

[Save] – Save current active parts to selected buffer
[Load] – Load parts from selected buffer
[Buff(n)] – Select part buffer to save/load

Rm – Remove parts inside the area/polygon
Kp – Keep parts inside the area/polygon
[Info] – Show information on selected parts
[All] – Select all parts.
[None] – Deselect all parts
[Rev] – Reverse part selection
[Apply] – Update drawing with selected parts. (when Auto=off).
[Done] – Exit Part Selection interface

Quick Guides

Quick Guide: How to use the Part Selection Interface

1. Select Remove or Keep
2. Select parts from the list
   Or pick from the model
3. Part types can be selected by checking the appropriate box on the left side
4. If Auto is not active (red) click [Apply]
The first four interfaces are for post processing, while the others are for pre processing. The pre processing capabilities include model operations like transform, translate, reflect, rotate, project and scale. Soon to be implemented will be element creation and error checking.
**CFD Time History Plot**

**Introduction**

**Interface Description**

Node list for history plot (1)
History Comp – List of Time History Components (2)
[Load] – Load D3THINS file to plot
[Upload] – Un-Load D3THINS files to free memory
[Plot] – Plot XY-Plot data in current XY-Plot window
[New] – Plot XY-Plot data in a new XY-Plot window
[Add] – Add XY-Plot data to current XY-Plot window
[Clear] – Clear selected nodes
[Raise] – Raise all open XY-Plot windows
[Pop] – Open and Raise all closed XY-Plot windows
[Done] – Exit CFD Time History Plot interface

**Quick Guides**

**Quick Guide: How to load a CFD time history component**
1. Click [Load]
2. Select d3thins or d3mean file and click [Open]
   (Node and component data should load)

**Quick Guide: How to view a CFD time history plot**
1. Select node(s) from list (1)
2. Select History Component (2)
3. Click [Plot]/[New]
**Introduction**

Use this interface to browse and display data stored in binary ASCII files. The branch system makes it easy to navigate and there is even the option to open multiple files for comparison.

**Interface Description**

[Load] – Load a BINOUT file  
[UnLoad] – Un-load selected BINOUT file  
[Save] – Save selected BINOUT branch to a file  
[Done] – Exit BINOUT interface

Open Files list – Select from list of currently loaded BINOUT files (1)  
Main Branch list – Select main branches in selected BINOUT file (2)  
Sub-Branch list – Select sub-branch part of selected main branch (3)  
List of main branch entities (4)  
List of main branch components (5)

[Plot] – Plot selected component in current XY-Plot window  
[New] – Plot selected component in a new XY-Plot window  
[Padd] – Add selected component to current XY-Plot window  
[Pop] – Open and Raise all closed XY-Plot windows  
[Clear] – Clear list selections  
[Info] – Show the general information about the current selections (6)

**Quick Guides**

**Quick Guide: How to Load and display data stored in a BINOUT file**

1. Click Load  
2. Select BINOUT file from list (1)  
3. Select main branch (2)  
4. Select sub-branch (3)  
5. Select entity and component (4) and (5)  
6. Click [Plot] or [New]
CGAT – CIRCULAR GRID ANALYSIS TECHNIQUE

Introduction

Interface Description

Circles X – Number of circles in local X direction
Circles Y – Number of circles in local Y direction
Diameter – Diameter of circles
Spacing – Distance between circles
Origin – Pick/Enter origin of circular grid (X,Y,Z coordinates)
Xpoint – Pick/Enter direction of grid x-axes (X,Y,Z coordinates)
Y-node – Pick/Enter a node for y-axes
Part No. – Enter part number or part pick at origin
Color – Color of circle line
Linew – Line width of circles
Pick – Pick/Enter circle ID e.g.: 2x3 for history plot
[Show] – Generate and display Grid circles
[Hide] – Remove grid circles from display
[Clear] – Delete all generated grid circles
[Plot] – Plot history of pick circles
[New] – Plot history of pick circles in a new window
[Padd] – Add plot history of pick circles current plot
[Done] – Exit Circular Grid interface
Quick Guides

Quick Guide: How to form a grid of circles
1. Enter number of columns (circles in X-direction)
2. Enter number of rows (circles in the Y-direction)
3. Enter circle diameter (calculated by default per model)
4. Enter spacing, measured from origin to origin
5. Check Origin to pick origin of grid
   Or Enter XYZ coordinates in text fields
6. Check Xpoint to pick point to form X-axis of grid
   Or Enter XYZ coordinates in text fields
7. Check Y-node to pick a point to from a XY plane for the grid
   Or Enter node ID in text field
8. Enter Part ID of part to apply circles
9. Select circle line color
10. Select Circle line width
11. Click [Show]

Quick Guide: How to view time history data for a circle
1. Check Pick to pick a circle from the grid
2. Or Enter the circle’s grid location e.g.: 2x3
3. Click [Plot]/[New]/[Padd]
Introduction

Interface Description

Pick blank – Pick blank part for skid tracing
Pick curve – Pick points along tool for skid tracing
Pick point – Pick a point on tool for skid tracing
Closest node – Displays node nearest picked coordinates

[Clr] – Clear all points in curve list
[Del] – Delete highlighted point from curve list
[Rep] – Replace highlighted point in curve list
[Add] – Add new point after highlighted point in curve list

Curve List: – List of curve coordinates

Skid – Change skid trace color/width
Tool – Change tool line color/width
Start – Change start line color/width
End – Change end line color/width
[Color] – Select line color
[Linew] – Select line width
Pick trace – Pick/Enter skid trace for time history plot

[Proj] – Project picked points/curve onto tool
[Show] – Generate and display skid mark tracks
[Hide] – Remove skid marks from display
[Clear] – Delete all generated skid marks
[Plot] – Plot history of pick skid mark
[New] – Plot history of pick skid mark in a new window
[Padd] – Add plot history of pick skid mark current plot
[Done] – Exit Skid Mark interface
Quick Guides

**Quick Guide: How to display a skid**

1. Place model at state 1
2. Check Pick blank to pick the blank part
   Or Enter Part ID in text field
3. Check Pick curve to pick multiple points on the tool
   Or Check Pick Point to pick a single point on the tool (click [Add] to add point to curve list)
4. Select Skid/Tool/Begin/End
5. Select line color
6. Select line width
7. Click [Proj]
   Increment states to display the skid marks

**Quick Guide: How to view time history data for a skid trace**

1. Check Pick trace to pick a skid trace
   Or Enter the trace number
2. Click [Plot]/[New]/[Padd]
### General Selection Menu

<table>
<thead>
<tr>
<th>Pick</th>
<th>Area</th>
<th>Poly</th>
<th>In</th>
<th>Out</th>
<th>Add</th>
<th>Rm</th>
<th>Keyin</th>
<th>Label</th>
<th>Selecting Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
<td><img src="image" alt="Check Box" /></td>
</tr>
</tbody>
</table>

- **Pick** – Select/Deselect a single entity
- **Area** – Select/Deselect entities inside/outside a user defined area
- **Poly** – Select/Deselect entities inside/outside a user defined region
- **In** – Select/Deselect entities inside the area/polygon
- **Out** – Select/Deselect entities outside the area/polygon
- **Add** – Add entities to selection
- **Rm** – Remove entities from selection

**[Keyin]** – Press this button to enter entity IDs (Hit enter to accept value)

- **Label** – Turn label on/off for new selection
- **LabOn** – Display labels for all entities
- **LabOff** – Hide labels for all entities
- **Clear** – Clear all entity selections
- **Save** – Save entities to buffer
- **Load** – Load saved entities from buffer
- **Desel** – Deselect last selected entity
- **Whole** – Select all entities within model
- **AllVis** – Select all visible entities
- **Rev** – Reverse entity selection
- **By Node** – Selection by node
- **By Elem** – Selection by element
- **By Part** – Selection by part
- **By Set** – Select entities using SET data
- **3Ds surf** – Toggle select surface nodes/elements on/off (3D parts only)

## Quick Guides

### Quick Guide: Selecting nodes/elements/parts

1. Select picking method Pick/Area/Poly
2. Select In/Out for area/poly
3. Select Add/Remove
4. Select picking ‘by’ method (Note ‘by SET’ opens a link dialog)
5. Make selections form model
   Or Enter node/element/part ID by clicking [Keyin] (Hit enter to accept entry)
**Move or Copy Elements**

**Introduction**

Use this interface to change element Part IDs.

**Interface Description**

Move – Move elements to a target part
Copy – Copy elements to a target part
[Apply] – Apply move or copy operation
Pick Target Part – Pick target part from drawing area
Part ID – Enter target part ID

Element ID Offset – New element IDs will be created using the existing IDs + ID value
Starting Element ID + 1 – New element IDs will be created by incrementing the ID value.
Element ID Offset – Enter ID value, used as ID offset/starting ID
Node ID Offset – Enter node ID offset
[Done] – Exit Move/Copy interface

**Quick Guides**

**Quick Guide: How to change an element’s association with a part**

1. Select elements using the general selection interface
2. Check Pick Target Part to pick from a part ID from the model
   Or Enter a Part ID (Hit enter to accept the value)
   (This is the Part ID that will be applied to the selected elements)
3. Click [Apply]

**Quick Guide: How to copy an element and apply an association with a part**

1. Select elements using the general selection interface
2. Check Pick Target Part to pick from a part ID from the model
   Or Enter a Part ID (Hit enter to accept the value)
   (This is the Part ID that will be applied to the selected elements)
3. Select Elem ID Offset to use an offset to create the new element IDs,
   enter the offset value in the box labeled ID (Hit enter to accept the value)
   Or Select Starting Elem ID +1 to use a new set of IDs to create the new element IDs. Enter the starting ID in the box labeled ID (Hit enter to accept the value)
4. Enter a node ID offset to create the new node IDs
5. Click [Apply]
TRANSFORM MODEL

Introduction

Use this interface to transform parts of the model.

Interface Description

[Transform] – Apply transformation
From – Form the coordinate system the model will transform from
To – Form the coordinate system the model will transform to
Node1F, 2F and 3F – Pick or enter node IDs to form the coordinate system the model will transform from
Node1T, 2T and 3T – Pick or enter node IDs to form the coordinate system the model will transform to

Copy elements – Copy elements while transforming the model
Transfer – Transfer selection to newly copied elements
Offset – EID Offset is the offset for the new element ID
Start ID – EID Offset is the starting element ID
Pick Part – Pick target part from drawing area or enter part ID
[Plist] – Open part list dialog
EID Offset – Enter element ID offset or starting element ID
NID Offset – Enter node ID offset
[Undo] – Undo last operation
[Done] – Exit Transform Model interface

Quick Guides

Quick Guide: How to transform a model
1. Select nodes/elements/parts to transform
2. Check From
3. Select nodes to form From coordinate system
4. Check To
5. Select nodes to form To coordinate system
6. Click [Transform]

Quick Guide: How to copy and transform a model
1. Follow above guide up to step 5
2. Select and enter a new starting ID
   Or Select and enter an ID offset
3. To associate new elements to a part pick/select/enter a part ID
4. Click [Transform]
TRANSLATE MODEL

Introduction

Use this interface to translate entities.

Interface Description

[Tran -] – Translate in negative direction
[Tran +] – Translate in positive direction
Direction – Select direction of translation
Translate distance – Enter distance to translate

Node 1, 2 and 3 – Pick/Enter node IDs to define N1-N2, N1-N2-N3 and N1 to N2 directions

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation
[Done] – Exit Translate Model interface

Quick Guides

Quick Guide: How to Translate a model
1. Select nodes by node/element/part using pick/area/region
2. Select Direction
3. If using N1-N2/N1-N2-N3/N1 to N2 then define node IDs
   (note when using N1 to N2 the distance between the N1 and N2 will be the
distance used for translation)
4. Click [Tran - / +] to translate entities

Quick Guide: How to Translate a model
1. Follow above guide up to step 6
2. Check Copy Elements
3. Select and enter a new starting ID
   Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Tran - / +] to translate nodes/elements/parts
**Reflect Model**

**Introduction**

Use this interface to reflect the model.

**Interface Description**

[Reflect] – Perform reflection for the selected nodes
Pick node as pt. of reflc – Pick a node as the origin of the reflection
Node ID – Enter a node ID as the origin of the reflection (Hit enter to accept)
XYZ – Enter x,y,z coordinates as the origin of the reflection
Plane – Select plane of reflection
Node 1, 2 and 3 – Enter node IDs to define planes using N1-N2 and N1-N2-N3

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation
[Done] – Exit Reflect Model interface

**Quick Guides**

**Quick Guide: How to reflect a model**

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates as the origin of the reflection
3. Select a plane of reflection
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Click [Reflect]

**Quick Guide: How to copy and reflect a model**

1. Follow above guide up to step 4
2. Check Copy Elements
3. Select and enter a new starting ID
   Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Reflect]
**ROTATE MODEL**

**Introduction**

Use this interface to rotate the model.

**Interface Description**

Rotate -/+ – Rotate the selected nodes in negative / positive direction
Pick node as origin – Pick a node as the origin of rotation
Node ID – Enter node ID as the origin of rotation (Hit enter to accept)
XYZ – Enter x, y, z coordinates as the origin of rotation
Rot. Axis – Select axis/plane of rotation
Rot. Angle – Enter angle of rotation in degrees
Node1, 2 and 3 – Enter node IDs for N1-N2 and N1-N2-N3 planes

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation
[Done] – Exit Rotate Model interface

**Quick Guides**

**Quick Guide: How to rotate a model**

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates as the origin of rotation
3. Select an axis/plane of rotation
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Enter rotation angle
6. Click [Rotate]

**Quick Guide: How to copy and rotate a model**

1. Follow above guide up to step 5
2. Check Copy Elements
3. Select and enter a new starting ID
   Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Rotate]
PROJECT MODEL

Introduction

Use this interface to create a projection of the model.

Interface Description

[Project] – Perform projection for selected nodes
Pick node on plane of p – Pick a node on the plane of projection
Node ID – Enter a node ID on the plane of projection (Hit enter to accept)
XYZ – Enter x, y, z coordinates on the plane of projection
Plane – Select plane of projection
Node 1, 2 and 3 – Enter node IDs for N1-N2 and N1-N2-N3 planes

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation
[Done] – Exit Project Model interface

Quick Guides

Quick Guide: How to project a model
1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates for a point on the plane of projection
3. Select a plane of projection
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Click [Project]

Quick Guide: How to copy and project a model
1. Follow above guide up to step 4
2. Check Copy Elements
3. Select and enter a new starting ID
   Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Project]
**SCALE MODEL**

**Introduction**

Use this interface to alter the scale of the model.

**Interface Description**

[Scale Down/Up] – Scale the model down/up with selected nodes
Pick node as origin – Pick a node as the origin of scaling
Node ID – Enter a node ID as the origin of scaling (Hit enter to accept)
XYZ – Enter x, y, z coordinates as the origin of scaling
Scale Dir – Select direction of scaling
Scale Factor – Enter scale factor
Node 1, 2 and 3 – Enter node IDs for N1-N2 and N1-N2-N3 planes

(See Transform Interface for details on Copy Elements interface)

[Undo] – Undo last operation
[Done] – Exit Scale Model interface

**Quick Guides**

**Quick Guide: How to scale a model**

1. Select nodes by node/element/part using pick/area/region
2. Pick/Enter a node ID or x, y, z coordinates as the origin of scaling
3. Select a direction/plane of scaling
4. If using N1-N2/N1-N2-N3 then define node IDs
5. Enter scale factor
6. Click [Scale Down/Up]

**Quick Guide: How to copy and scale a model**

1. Follow above guide up to step 4
2. Check Copy Elements
3. Select and enter a new starting ID
Or Select and enter an ID offset
4. To associate new elements to a part pick/select/enter a part ID
5. Click [Scale Down/Up]
SHELL/SEGMENT/TSHELLNORMALS

Introduction

Use this interface to display shell/segment/tshell normals

Interface Description

Select entity type(1)
Show Normal – Click element to Show element normal
Reverse Normal – Interface to reverse element normal

Vector size – Set normal vector size (>1.0) larger
Pick element – Pick element to reverse normal
Pick seed – Pick elements as a seed for auto-reverse
[Autoreverse] – Auto-reverse normal according to seed element
[Done] – Exit Shell/Segment/Tshell Normals interface

Quick Guides

Quick Guide: How to display Shell/Segment/Tshell normals
1. Select normal type
2. Select Show/Reverse normal
## Introduction

Page 3 provides viewing and editing tools for LS-DYNA keyword files. Each main keyword has its own section accessible via the buttons shown below. Each section contains a list of sub-keywords which can be selected opening individual keyword input dialogs. Although page 3 allows the creation of individual keyword entries, to create a keyword file from scratch use the tools on Page 4.

<table>
<thead>
<tr>
<th>*Damp</th>
<th>*Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>*Airbag</td>
<td>*Dbase</td>
</tr>
<tr>
<td>*Ale</td>
<td>*Define</td>
</tr>
<tr>
<td>*Boundry</td>
<td>*Element</td>
</tr>
<tr>
<td>*Cnstrnd</td>
<td>*Eos</td>
</tr>
<tr>
<td>*Contact</td>
<td>*Initial</td>
</tr>
<tr>
<td>*Control</td>
<td>*Intgrtn</td>
</tr>
<tr>
<td>*Deform</td>
<td>*Intrfac</td>
</tr>
<tr>
<td>*Part</td>
<td>*Rgdwall</td>
</tr>
<tr>
<td>*Hrglass</td>
<td>*Section</td>
</tr>
<tr>
<td>*Set</td>
<td>*Termnt</td>
</tr>
<tr>
<td>*User</td>
<td></td>
</tr>
</tbody>
</table>
Keyword Selection Menu

Access the main keyword branch by selecting from the main menu shown on the previous page. This will open up main branch navigation interfaces below, in the dynamic interface area. To open a specific keyword select it from this list and click [Edit]. There are additional options available for specific keywords and these are described below.

General Keyword Options

[Edit] – Open keyword input window for selected keyword
[All] – Display all keywords
[Model] – Display all keywords in current model
[Done] – Close the current interface

*CONTROL Extra Keyword Options

As there can only be one of each *CONTROL keyword type they are deleted here instead of the keyword input dialog.

Create / Edit *CONTROL keyword data
Delete *CONTROL keyword data

*EOS / *HOURGLASS / *MAT Extra Keyword Options

Open Reference by interface

*MAT Extra Keyword Options

Use this menu to limit the materials shown in the list by type
Sort list be Name or Type number
Limit list alphabetically / numerically
REF BY INTERFACE

[All] – Select all
[None] – Deselect all
[Rev] – Reverse selection
[Done] – Close Referenced By interface
[Save] – Save selected part IDs in selected buffer
[Buff n] – Select buffer to save data
**KEYWORD INPUT DIALOG**

- [NewID] – Generate a new ID for keyword
- [Clear] – Clear all entries (only on some keyword input dialogs)
- [Pick] – Open pick interface to allow data to be picked directly from the model
- [Add] – Create a fresh keyword using default values
- [Accept] – Accept changes
- [Del/UnD] – Delete/Undelete selected keyword
- [Default] – Set entries to default values
- [Done] – Close Keyword Input dialog, (choose to accept changes)

**Quick Guides**

**Quick Guide: How to create a keyword using Page 3**

1. Open specific keyword form
2. Click Add to open a default form with a new ID (Note each keyword must have a unique internal ID which is generated using Add/NewID. On some forms it may be possible to input a user ID for this purpose.
3. Enter data in specified fields
4. Click Accept to save keyword to memory
5. To save changes permanently the keyword file must be saved via: File/Save keyword
### Quick Guides

#### Quick Guide: Entering Set Data

1. Enter values in associated card
2. Click [Insert] to add the set data. If set data already exists, the new data will be added after the currently highlighted set.
   
   Or  Click [Replace] to replace the currently highlighted set.
LINK KEYWORD DIALOG

This dialog displays a list of keyword IDs suitable for the text field used.

[Done] – Accept selection and close Link Keyword Dialog
[Cancel] – Cancel selection and close Link Keyword Dialog
[Read] – Open selected keyword (for viewing only)

PICK INTERFACE

[Accept] – Accept selections and resize Keyword Input dialog
[Cancel] – Cancel selections and resize Keyword Input dialog
[ResDlg] – Resize Keyword Input dialog

Quick Guides

Quick Guide: How to pick data from the model
1. Click [Pick] to open Pick Interface
2. Check the field to pick data
3. Pick data by clicking on the model
4. After data has been picked the next field will be checked ready for picking, or check another manually
5. Click [Accept]
**Dynamic Keyword Input Forms**

If a keyword input field has a red heading then that field is used as a dynamic form switch. In the following example entering a value greater than one will result in card 2 being replaced. There are various other keywords that have similar abilities and information on the triggering value should be displayed in the help comment for that field.

---

<table>
<thead>
<tr>
<th>TITLE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>1</th>
<th>MID</th>
<th>RO</th>
<th>PR</th>
<th><strong>N</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>2</th>
<th>CID</th>
<th>D01</th>
<th>CID</th>
<th>C20</th>
<th>C02</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>3</th>
<th>P0</th>
<th>PHI</th>
<th>IVS</th>
<th>G</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

N = 0 therefore the standard form is shown.

---

<table>
<thead>
<tr>
<th>TITLE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>1</th>
<th>MID</th>
<th>RO</th>
<th>PR</th>
<th><strong>N</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>2</th>
<th>SGL</th>
<th>SW</th>
<th>ST</th>
<th>LCID</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>3</th>
<th>P0</th>
<th>PHI</th>
<th>IVS</th>
<th>G</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

N > 0 therefore card 2 has changed to allow different keyword data entries.
Define table is a unique keyword form as it allows one to plot data stored in the list.

[Replace] – Replace selected data with data in text fields
[Insert] – Insert data after the currently selected one
[Delete] – Delete the selected data from the list
[Help] – Display input help
[Plot] – Plot load curve data in current XY-Plot window
[Raise] – Raise all open XY-Plot windows

**Quick Guides**

**Quick Guide: How to use define_table**
1. Enter a title
2. Enter a table ID
3. Enter a value and a load curve ID
4. Click [Insert]/[Replace] to add to table
5. Click [Plot] to display table data in a XY-Plot widow
ENTITY DRAW OPTION

Introduction

If a draw button is shown in the top left of a keyword input form then the keyword entity can be displayed in the drawing area. Once an entity type has been drawn, more of this entity type can be selected using the Entity Operation interface. See instructions under Page 4 section for further details.

Interface Description

[Draw] – Draw entity
[Restore Form]– Restore keyword input form

Quick Guides

Quick Guide: How to display an entity from a keyword input form
1. Select the entity ID from the list on the right side of the form
2. Click [Draw]
   (The form will shrink to one side and the selected entity will be displayed)
3. Use the entity operation interface to show/blank multiple entities
4. Click [Restore Form] to retrieve the keyword input form
   (The drawn entity will be removed from the drawing area upon restoring the keyword input form)
The interfaces on this page are all pre-processing tools, including airbag folding, dummy manipulation and positioning and entity generation. Although entities can be generated on page 3, the interfaces on this page allow the user to use the drawing area to visualize entities while working on them.
AIRBAG FOLDING

Introduction
Use this interface to define and view air bag folds.

Interface Description
Crea – Create a new fold definition
Mod – Modify an existing fold
Show – Show the selected folds

Fold Definition
ID – Enter fold ID
Title – Enter title for this fold
Fold Type – Select fold type
N1-N2 – Pick/Enter 2 node IDs to define the fold line
Pt+Dir – Define a point and a direction to define the fold line
N1 – Pick/Enter a node ID to define N1
N2 – Pick/Enter a node ID to define N2
N3 – Pick/Enter a node ID to define the active side of the fold
Up – Fold the active side of the fold, up
Down – Fold the active side of the fold, down
Thick – Enter thickness of shell
Scale – Enter scale value
Radius – Enter radius for a circle within which to form the spiral fold
Clearance – Enter clearance value
Ignore – Ignore thickness from previous folds
iMove – Move the fold line nodes back by this amount

Enter the number of steps to take for each fold during animation
Anim – Animate folds upon clicking folding
Loop – Loop animation of folds

[Apply] – Apply changes to fold definition
[Fold] – Execute all folds
[1Fold] – Execute selected fold
[Line] – Create fold lines by cutting elements
[Step] – Step through all folds one at a time
[Reset] – Reset model to original geometry
[Unfold] – Unfold airbag
[Done] – Exit the Airbag Folding interface
Quick Guide: Creating an airbag fold

1. Click create
2. Enter fold ID and name
3. Select fold type
4. Define fold line by picking 2 nodes
   Or by picking 1 node and a direction
5. Pick a node on the active side of the fold. (This active side will be folded up/down/in)
6. Enter fold thickness
7. Enter fold scale for thin/tuck
   Or Enter radius for thick/spiral
8. Select direction to fold active side (Up/Down) (Does not apply to tuck/scrunch)
9. Select ignore to ignore previous fold thicknesses
10. Enter iMove value for thin fold only. This will be the distance moved back by the upper (see diagram below)
11. Click [Apply] create fold
Quick Guide: How to simulate all folds
1. Click [Fold] to initiate all defined folds

Quick Guide: How to simulate a single fold
1. Select a fold
2. Click [1Fold]

Quick Guide: Animating folds
1. Enter number of steps to animate per fold
2. Check Anim and click [Step] to move through the folding step by step
3. Check [Loop] to automatically animate all the folds and to keep looping
**Dummy Positioning Interface**

**Introduction**

Use this interface to manipulate a HYBRID III dummy.

**Interface Description**

[Read] – Read in a Tree file  
[Write] – Open file write interface  
[Reset] – Reset model to last saved settings  

Sel – Select a rigid body component in current occupant  
Cre – Create a rigid body component in current occupant (Not implemented yet)  
Del – Delete selected rigid body component in current occupant (Not implemented yet)

Positioner interface  
H-Point operations – Operations about the H-Point for the entire occupant  
Limb operations – Operations affecting joints and limbs

H-point/Global operations  
Rotate – Rotate occupant  
Translate – Translate occupant  
Mirror – Mirror occupant about a plane

H-Point location  
X/Y/Z – X/Y/Z coordinates of the H-Point  
H-Pt N – Pick/Enter node ID for new H-Point location
Rotation Interface

Global X/Y/Z – H-Point rotation about global x/y/z axes
User Defined – H-Point rotation about a user defined direction
N1/N2 – Pick/Enter 2 nodes to define a direction
Rot. Ang. – Enter angle to rotate occupant
Div – Select the number divisions used for half the tuning range (i.e. 0 to X or 0 to –X)
Range – Select/Enter angle for the extremes of the fine tuning range
Slider – Drag slider to rotate the occupant about the rotation axis
Always update – Always update occupant’s position after operations
[Update] – Update occupant’s current position
[Reset] – Revert the last operation performed on the occupant

Translation Interface

Global X/Y/Z – Translate occupant along global x/y/z axes
User Defined – Translate occupant along a user defined direction
N1/N2 – Pick/Enter 2 nodes to define a direction
Distance Factor – Enter distance to translate occupant
Always update – Always update occupant’s position after operations
[Update] – Update occupant’s current position
[Reset] – Revert the last operation performed on the occupant
Mirror Interface

- Z(xy) plane – Mirror occupant with respect to z-plane
- X(yz) plane – Mirror occupant with respect to x-plane
- Y(zx) plane – Mirror occupant with respect to y-plane
- User Defined – Mirror with respect to user defined plane
- N1/N2/N3 – Pick/Enter nodes to define a plane
- [Apply] – Mirror occupant
- Always update – Always update occupant’s position after operations
- [Update] – Update occupant’s current position
- [Reset] – Revert the last operation performed on the occupant

Limb Operations interface

- Orient Limb – Orient the selected joint
- Joint Constraints – Set the joint’s orientation constraints
- Select limb in the occupant
Limb Orientation interface

Rot. Ang. – Enter angle to rotate limb
Div – Select number of divisions for half the fine tuning range
Range – Select/Enter angle for the extremes of the fine tuning range
Slider – Drag slider to rotate the limb about the rotation axis
N/A – ??
Always update – Always update occupant’s position after operations
Lock children – Lock children to rotate with current limb

Joint setting interface

CP:N – Assign control point to another node
X/Y/Z – Enter control point X/Y/Z coordinates
Global X/Y/Z axes – Set x/y/z-axis as the current rotation axis
User Defined – Assign a user defined axis as the current rotation axis
N1– Pick/Enter a node to define a rotation axis. (The CP will be used as the base point)
Lock x/y/z – Prevent the joint from rotating about the x/y/z-axis
[Default] – Reset joint settings to default values
File Write interface

Keyword file – Write data as a keyword file for the occupant model (*.k)
Intermediary file – Write data in keyword format with occupant information (*.inf)
Tree file – Write tree file only
[Filename] – Enter output filename including full path
[…] – Browse to find a directory/filename
[Write] – Write file using selected format
[Done] – Close File Write interface

Quick Guides

Quick Guide: How to load an occupant for dummy positioning
1. File/Open/BinaryPlot
2. Select a .inf file
3. Click [4]
4. Click [DmyPos]
5. Click [Read]
6. Select a tree file for occupant

Quick Guide: How to rotate the occupant about the H-point
1. Select H-Point Operations
2. Select Rotate
3. Select rotation axis X,Y,Z
   Or User defined (Pick 2 nodes to form axis of rotation)
4. Enter angle to rotate occupant (Rot. Ang.)
5. Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
6. Click arrows to rotate occupant
   Or Use fine tuning
      ➢ Select number of divisions for half the tuning range (This scale will be applied to full range)
      ➢ Select/Enter angle for the extremes of the fine tuning range
      ➢ Drag slider to adjust fine tuning
Quick Guide: How to translate the occupant
1. Select H-Point Operations
2. Select Translate
3. Select direction of translation X,Y,Z
   Or User Defined (Pick 2 nodes to form direction vector)
4. Enter distance to translate occupant
5. Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
6. Use arrows to move occupant

Quick Guide: How to mirror the occupant
1. Select H-Point Operations
2. Select Translate
3. Select plane of reflection XY,YZ,XZ
   Or User Defined (Pick 3 nodes to form a plane of reflection)
   Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
4. Click [Apply]

Quick Guide: How to rotate limbs
1. Select Limb operations
2. Select Joint settings
3. Select limb to orientate, either from the list or by right-clicking on the model
4. Lock/Unlock rotation axes
5. Select X/Y/Z axes (if unlocked)
   Or pick/enter a node ID to define a rotation axis (The CP is used as the other node)
6. Select Orient limb
7. Check Always Update for automatic model update with no undo action (The manual method allows 1 undo action)
8. Check Lock Children to move children limbs with the parent
9. Left-Click and drag to rotate limb
   Or Enter angle of rotation and click arrows to rotate limb
   Or Use fine tuning
      ➢ Select number of divisions for half the tuning range (This scale will be applied to full range)
      ➢ Select/Enter angle for the extremes of the fine tuning range
      ➢ Drag slider to adjust fine tuning
Introduction

This interface is used with many of the interfaces on Page 4. It allows the user to select entities from the drawing area to display them as well as for modification and deletion.

Interface Description

- **Pick Entity** – Pick a single entity
- **Area** – Select entities inside/outside a user defined area
- **Polygon** – Select entities inside/outside a user defined region
- **In** – Select entities inside the area/polygon
- **Out** – Select entities outside the area/polygon
- **Partial** – Select entities even if only partially within the area
- **Whole** – Only select entities completely within the area
- **Entity Type** – Select entity type to be operated upon

Using Show

When Show is selected on the entity interface the following options will be available.

- **Operation**:
  - **Show** – Show entity
  - **Blank** – Blank entity/part/element
  - **Kcard** – Show keyword form for entity

Quick Guides

**Quick Guide: How to Show entities on the drawing area**

1. Click Show
2. Select selection method (pick/area/polygon, in/out, partial/whole)
3. Clicking on the drawing area will show entities if they exist, it is advisable to use area selection to show entities in a certain region. Use Show all to display all entities.
**Quick Guide: How to Blank entities on the drawing area**
1. Click Blank
2. Select entity type, entity/part/element
   (Note: Parts / Elements can not be shown once blanked unless [Un Blank All] is used, other entities can be shown using the Show interface)
3. Select entity selection method (pick/area/polygon, in/out, partial/whole)
4. Make selections from the drawing area

**Quick Guide: How to Identify entities on the drawing area**
1. Click Identify
2. Select entity selection method (pick/area/polygon, in/out, partial/whole)
3. Select entity type, entity/part/element/node
4. Make selections from the drawing area

**Quick Guide: How to use KCard**
1. Select KCard
2. Select entity type
3. Pick visible entities to show their keyword data
   (Note to pick entities like Box data these must be visible first, so select them from the list or use the Show entity operation)
Using Modify/Delete

Once modify / delete is selected in the entity interface on page 4, the Entity Operations Interface will change. One can no longer show / blank entities. Select is used to highlight/un-highlight displayed entities for modification / deletion. In some cases only a single entity can be modified at a time. In this situation selecting entities using area/polygon selection will only highlight the first entity.

Operation:
Select – Select entities
Kcard – Show keyword form for entity

Quick Guides

Quick Guide: How to Select entities
1. Click Select
2. Select entity selection method (pick/area/polygon, in/out, partial/whole)
3. Select entity type
4. Make selections from drawing area

Quick Guide: How to use KCard
This operation is unaffected by the interface change, see above for operation details.
SET DATA

Introduction

Use this interface to show/create/modify/delete Set data.

Interface Description

Show – Show existing Set data
Create – Create new Set data
Modify – Modify existing Set data
Delete – Delete existing Set data

Label – Select label type

[Set_Node, etc] – Select entity type to operate on

Set ID – Enter Set ID. All node sets should have a unique set ID.

Title – Enter title for new/current set (optional).

DA1 – Fist nodal attribute (default value is 0.0)
DA2 – Second nodal attribute (default value is 0.0)
DA3 – Third nodal attribute (default value is 0.0)
DA4 – Fourth nodal attribute (default value is 0.0)

[All] – Select all sets
[None] – Deselect all sets
[Rev] – Reverse selection

[Apply] – Apply entries for creation/modification
[Cancel] – Cancel entries for modification
[Write] – Write selected sets to a file
[Card] – Open keyword input dialog for this set
[Done] – Exit Set Data interface

SET_SEGMENT extra options

Shell – Pick shell element to create segment
Solid – Pick solid face to create segment
2DPart – Pick shell part to create segment
3DPart – Pick solid part to create segment
Quick Guides

Quick Guide: Different ways of displaying set data
1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity

Quick Guide: How to create a Set
1. Select Create
2. Select set type
3. Enter unique SetID
4. Enter optional Title
5. Enter nodal attributes
6. Select nodes using General Selection Interface (see above)
7. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Set data
1. Select set data from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note: the list and drawing area will only show previously selected entities)
3. Select a Set from the list
   Or Use Select Operation on the Entity Operations Interface to select a Set from the drawing area
4. Modify nodal attributes
5. Use General Selection Interface to modify nodes in the set
6. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Set data
1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note: the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
Quick Guide: How to write out Set data

1. Select Show
2. Select set data
3. Click [Write]
4. Enter filename
5. Click [Save]
**DEFINE_BOX DATA**

**Introduction**

Use this interface to show/create/modify/delete *DEFINE_BOX entities.

**Interface Description**

Show – Show existing *DEFINE_BOX entities  
Create – Create new * DEFINE_BOX entities  
Modify – Modify existing * DEFINE_BOX entities  
Delete – Delete existing * DEFINE_BOX entities

Label – Select label type

[All] – Select all * DEFINE_BOX entities  
[None] – Deselect all * DEFINE_BOX entities  
[Rev] – Reverse selection  
[AList] – Select all entities within the selected range  
[Apply] – Apply entries for creation/modification or selections for deletion  
[Cancel] – Cancel entries for modification  
[Done] – Exit *Define_Box Data interface

**Create**

Title – Enter box title  
BOXID – Enter box ID  
Node1, 2 – Pick minimum and maximum nodes for extremes of box  
Areain – Select nodes inside a user defined area for the box  
Polyin – Select nodes inside a user defined region for the box

**Modify**

Title – Enter box title  
X – Translate/Scale box in x direction  
Y – Translate/Scale box in y direction  
Z – Translate/Scale box in z direction  
Translate – Translate box  
Enter distance to translate box  
[Tran-] – Translate in negative direction  
[Tran+] – Translate in positive direction  
Scale – Scale box  
Enter scale factor  
[Scale-] – Scale box down  
[Scale+] – Scale box up
Quick Guides

**Quick Guide: Different ways of displaying Box data**
1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity

**Quick Guide: How to create a Box**
1. Select Create
2. Select box type
3. Enter optional Title
4. Enter box ID
5. Enter min/max coordinates for extreme vertices
   (These can be 2 nodes picked from the model shown in RED)
Or Use Area/polygon to pick nodes to be contained by the box (Shown in blue)
6. Click [Apply] to save changes to memory
(Nota to save changes permanently use File/Save Keyword)
Quick Guide: How to modify existing Box data

1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a box from the list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Modify box parameters
5. Use Translate to translate the box
   ➢ Select Translate
   ➢ Select direction
   ➢ Enter distance using model units
   ➢ Click Trans +/- to translate box
6. Use Scale to change the dimensions of the box.
   ➢ Select Scale
   ➢ Select direction
   ➢ Enter scale factor
   ➢ Click Scale +/- to scale box
7. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Box data

1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note: the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
4. Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
5. Click [Apply] to delete selected entities
6. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
**DEFINE_COORDINATE DATA**

**Introduction**

Use this interface to show/create/modify/delete *DEFINE_COORDINATE entities.

**Interface Description**

**Interface Description**

Show – Show existing * DEFINE_COORDINATE entities
Create – Create new * DEFINE_COORDINATE entities
Modify – Modify existing * DEFINE_COORDINATE entities
Delete – Delete existing * DEFINE_COORDINATE entities
Label – Select label type
Coord Type – Select coordinate type
CID – Coordinate ID. A unique number has to be defined
[New ID] – Generate a new ID for keyword
Title – Enter a title for the coordinate entity
[All] – Select all * DEFINE_COORDINATE entities
[None] – Deselect all * DEFINE_COORDINATE entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion
[Cancel] – Cancel entries for modification
[Done] – Exit * Define_Coordinate Data interface

**Coordinate Type Options**

**Nodes**

N1 – Enter node ID for node at local origin
N2 – Enter node ID for node on local x-axis
N3 – Enter node ID for node on local x-y plane
Flag – Set to unity, 1, if the local system is to be updated each time step for the BOUNDARY_SPC nodal constraints and ELEMENT_BEAM type 6, the discrete beam element. Generally, this option when used with nodal SPC's is not recommended since it can cause excursions in the energy balance because the constraint forces at the node may go through a displacement if the node is partially constrained

**System**

XO – x-coordinate of origin
YO – y-coordinate of origin
ZO – z-coordinate of origin
XL – x-coordinate of point on local x-axis
YL – y-coordinate of point on local x-axis
ZL – z-coordinate of point on local x-axis
XP – x-coordinate of point in local x-y plane
YP – y-coordinate of point in local x-y plane
ZP – z-coordinate of point in local x-y plane

*Vector
XX – x-coordinate on local x-axis. Origin lies at (0,0,0)
YX – y-coordinate on local x-axis
ZX – z-coordinate on local x-axis
 XV – x-coordinate of local x-y vector
YV – y-coordinate of local x-y vector
ZV – z-coordinate of local x-y vector

Quick Guides

Quick Guide: Different ways of displaying Coordinate data
1. Select Show
2. Select entities from the list to display
3. Select label format
   ➢ None – No label is shown next to entity
   ➢ Symbol – Displays entity ID next to entity

Quick Guide: How to create Coordinate data
1. Select Create
2. Select coordinate type
3. Enter unique Coordinate System ID
   Or Click [NewID]
4. Enter Title
5. Enter coordinate type options
   (Note: Use check boxes to pick nodes from the drawing area)
6. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Coordinate data
1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note: the list and drawing area will only show previously selected entities)
3. Select an entity from the list
   Or Use Select Operation on the Entity Operations Interface to select one from the drawing area
4. Modify parameters
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)
Quick Guide: How to delete existing Coordinate data

1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note: the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
**CONSTRUCTED_GENERALIZED_WELD_SPOT DATA (GEWeld)**

**Introduction**

Use this interface to show/create/modify/delete *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities.

**Interface Description**

Show – Show existing *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities
Create – Create new *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities
Modify – Modify existing *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities
Delete – Delete existing *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities

Label – Select label type
[New ID] – Generate a new ID for keyword
NSID – Enter Node Set ID
[KCard] / [Entitylist] – Toggle between keyword card entry interface and entity list
[All] – Select all *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities
[None] – Deselect all *CONSTRUCTED_GENERALIZED_WELD_SPOT* entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range (Only applies when there are more than 20 entities in the list)
[Apply] – Apply entries for creation/modification or selections for deletion
[Done] – Exit Constrained Generalized Weld Spot Data interface
Quick Guides

**Quick Guide: Different ways of displaying Constrained Generalized Weld Spot data**
1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity

**Quick Guide: How to create a Constrained Generalized Weld Spot**
1. Select Create
2. Enter NSID, or click [NewID] to generate a new one
3. Select nodes using General Selection Interface
4. Click [EditParam] to modify optional keyword data
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

**Quick Guide: How to modify existing Constrained Generalized Weld Spot data**
1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select Sngl / Multi
   Select Single Modification Mode – Use to modify entity nodes and parameters. When an entity is
   selected from the drawing area the modification interface will open automatically.
   (Note: When selecting from the drawing area, once modification mode is active, entity selection is not
   possible)
   - Modify parameters and click [Apply] to save changes to memory
   - Or click [Cancel] and select another entity
   Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is
   selected it is only highlighted, so additional entities can be selected without modifying the first. Once all
   selections have been made click [Edit]
   - Modify parameters and Click [Apply] to save changes to memory
   - Or click [Cancel]
   (Note to save changes permanently use File/Save Keyword)

**Quick Guide: How to delete existing Constrained Generalized Weld Spot data**
1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note: the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
**CONSTRUCTED_SPOT_WELD_DATA (SPWELD)**

**Introduction**

Use this interface to show/create/modify/delete *CONSTRUCTED_SPOTWELD entities.

**Interface Description**

Show – Show existing *CONSTRUCTED_SPOT_WELD entities
Create – Create new *CONSTRUCTED_SPOT_WELD entities
Modify – Modify existing *CONSTRUCTED_SPOT_WELD entities
Delete – Delete existing *CONSTRUCTED_SPOT_WELD entities

Label – Select label type
Auto Create – Auto create spot weld
N1,N2 – Pick/Enter Node ID 1 and 2
Show Parameters – Toggle view spot weld data parameters
[All] – Select all *CONSTRUCTED_SPOTWELD entities
[None] – Deselect all *CONSTRUCTED_SPOTWELD entities
[Rev] – Reverse selection
[AList] – Select all *CONSTRUCTED_SPOTWELD entities within the selected range (Only applies when there are more than 20 entities in the list)
[Apply] – Apply entries for creation/modification or selections for deletion
[Cancel] – Cancel entries for modification
[Done] – Exit Spot Weld Data interface

**Quick Guides**

**Quick Guide: Different ways of displaying Constrained Spot Weld data**

1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
   - Detail – Displays entity ID and SpWeld node IDs next to entity
Quick Guide: How to create a Constrained Spot Weld

(See keyword user’s manual section on *CONSTRAINED_SPOT_WELD for parameter details)

<table>
<thead>
<tr>
<th>Entity</th>
<th>SpotWelds Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>SN</td>
<td>SS</td>
</tr>
<tr>
<td>N</td>
<td>M</td>
</tr>
<tr>
<td>TF</td>
<td>EP</td>
</tr>
</tbody>
</table>

1. Select Create
2. Enter parameters
3. Autocreate enabled (Enables creation without having to click apply after every entry)
   - Pick/Enter 2 node IDs
4. Or Autocreate disabled
   - Pick/Enter 2 node IDs
5. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Spot Weld data

1. Select entities to modify from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select Sngl / Multi
   Select Single Modification Mode – Use to modify SpWeld nodes and parameters. When an entity is
   selected from the list or drawing area the modification interface will open automatically.
   (Note: When selecting from the drawing area, once modification mode is active, entity selection is not
   possible)
   - Modify parameters and click [Apply] to save changes to memory
   - Or click [Cancel]
Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is
selected from the drawing area it is only highlighted, so another entity can be selected without
modifying the first. Once all selections have been made click [Edit]
   - Modify parameters and Click [Apply] to save changes to memory
   - Or click [Cancel]
(Note to save changes permanently use File/Save Keyword)
**CONSTRAINED_SPOT_WELD DATA (SPWELD)**

**Introduction**

Use this interface to show/create/modify/delete *CONSTRAINED_SPOT_WELD* entities.

**Interface Description**

Show – Show existing *CONSTRAINED_SPOT_WELD* entities
Create – Create new *CONSTRAINED_SPOT_WELD* entities
Modify – Modify existing *CONSTRAINED_SPOT_WELD* entities
Delete – Delete existing *CONSTRAINED_SPOT_WELD* entities

Label – Select label type
Auto Create – Auto create spot weld
N1,N2 – Pick/Enter Node ID 1 and 2
Show Parameters – Toggle view spot weld data parameters
[None] – Deselect all *CONSTRAINED_SPOT_WELD* entities
[Rev] – Reverse selection
[All] – Select all *CONSTRAINED_SPOT_WELD* entities
[Rev] – Reverse selection
[Apply] – Apply entries for creation/modification or selections for deletion
[Cancel] – Cancel entries for modification
[Done] – Exit Spot Weld Data interface

**Quick Guides**

*Quick Guide: Different ways of displaying Constrained Spot Weld data*

4. Select Show
5. Select entities from the list to display
6. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
   - Detail – Displays entity ID and SpWeld node IDs next to entity
Quick Guide: How to create a Constrained Spot Weld

(See keyword user’s manual section on *CONSTRAINED_SPOT_WELD for parameter details)

<table>
<thead>
<tr>
<th>Entity SpotWData</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>SN</td>
<td>SS</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

6. Select Create
7. Enter parameters
8. Autocreate enabled (Enables creation without having to click apply after every entry)
   - Pick/Enter 2 node IDs
9. Or Autocreate disabled
   - Pick/Enter 2 node IDs
10. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Spot Weld data

4. Select entities to modify from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
5. Select Modify (Note the list and drawing area will only show previously selected entities)
6. Select Sngl / Multi
   Select Single Modification Mode – Use to modify SpWeld nodes and parameters. When an entity is
   selected from the list or drawing area the modification interface will open automatically.
   (Note: When selecting from the drawing area, once modification mode is active, entity selection is not
   possible)
   - Modify parameters and click [Apply] to save changes to memory
   - Or click [Cancel]

Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is
selected from the drawing area it is only highlighted, so another entity can be selected without
modifying the first. Once all selections have been made click [Edit]
- Modify parameters and Click [Apply] to save changes to memory
- Or click [Cancel]

(Note to save changes permanently use File/Save Keyword)
Quick Guide: How to delete existing Constrained Spot Weld data

1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
**CONSTRANGED_SPOT_WELD DATA (SPWELD)**

**Introduction**

Use this interface to show/create/modify/delete *CONSTRANGED_SPOTWELD entities.

**Interface Description**

Show – Show existing *CONSTRANGED_SPOT_WELD entities
Create – Create new *CONSTRANGED_SPOT_WELD entities
Modify – Modify existing *CONSTRANGED_SPOT_WELD entities
Delete – Delete existing *CONSTRANGED_SPOT_WELD entities

Label – Select label type
Auto Create – Auto create spot weld
N1,N2 – Pick/Enter Node ID 1 and 2
Show Parameters – Toggle view spot weld data parameters
[All] – Select all *CONSTRANGED_SPOTWELD entities
[None] – Deselect all *CONSTRANGED_SPOTWELD entities
[Rev] – Reverse selection
[AList] – Select all *CONSTRANGED_SPOTWELD entities within the selected range (Only applies when there are more than 20 entities in the list)
[Apply] – Apply entries for creation/modification or selections for deletion
[Cancel] – Cancel entries for modification
[Done] – Exit Spot Weld Data interface

**Quick Guides**

*Quick Guide: Different ways of displaying Constrained Spot Weld data*

7. Select Show
8. Select entities from the list to display
9. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
   - Detail – Displays entity ID and SpWeld node IDs next to entity
Quick Guide: How to create a Constrained Spot Weld

(see keyword user’s manual section on *CONSTRAINED_SPOT_WELD for parameter details)

<table>
<thead>
<tr>
<th>Entity</th>
<th>SpotWData</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>SN</td>
<td>SS</td>
<td>N</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

11. Select Create
12. Enter parameters
13. Autocreate enabled (Enables creation without having to click apply after every entry)
   - Pick/Enter 2 node IDs
14. Or Autocreate disabled
   - Pick/Enter 2 node IDs
15. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Spot Weld data

7. Select entities to modify from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
8. Select Modify (Note the list and drawing area will only show previously selected entities)
9. Select Sngl / Multi
   Select Single Modification Mode – Use to modify SpWeld nodes and parameters. When an entity is
   selected from the list or drawing area the modification interface will open automatically.
   (Note: When selecting from the drawing area, once modification mode is active, entity selection is not
   possible)
     - Modify parameters and click [Apply] to save changes to memory
     - Or click [Cancel]
   Or Select Multiple Modification Mode – Use to modify parameters of multiple entities. When an entity is
   selected from the drawing area it is only highlighted, so another entity can be selected without
   modifying the first. Once all selections have been made click [Edit]
     - Modify parameters and Click [Apply] to save changes to memory
     - Or click [Cancel]
   (Note to save changes permanently use File/Save Keyword)
**CONSTRANGED_NODAL_RIGID_BODY_DATA (CNRB)**

**Introduction**

Use this interface to show/create/modify/delete *CONSTRANGED_ NODAL_RIGID_BODY entities.

**Interface Description**

Show – Show existing *CONSTRANGED_ NODAL_RIGID_BODY entities
Create – Create new *CONSTRANGED_ NODAL_RIGID_BODY entities
Modify – Modify existing *CONSTRANGED_ NODAL_RIGID_BODY entities
Delete – Delete existing *CONSTRANGED_ NODAL_RIGID_BODY entities

Label – Select label type
[New ID] – Generate a new ID for keyword
PID – Enter Part ID of the nodal rigid body
CID – Enter Coordinate ID/Open link dialog by clicking on the label
NSID – Enter Node Set ID
Pick PNODE – Pick node from model to be PNODE
PNODE – Enter PNODE/Open link dialog by clicking on the label
IPRT – Select print flag

[All] – Select all *CONSTRANGED_ NODAL_RIGID_BODY entities
[None] – Deselect all *CONSTRANGED_ NODAL_RIGID_BODY entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion
[Done] – Exit Constrained Nodal Rigid Body Data interface

**Quick Guides**

**Quick Guide: Different ways of displaying CNRB data**

1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity

**Quick Guide: How to create a CNRB**

1. Select Create
2. Enter parameters
3. Enter a unique NSID or click New ID to generate a new one
4. Select nodes using General Selection Interface
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)
Quick Guide: How to modify existing CNRB data

1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a single entity from the list
   Or Use Select Operation on the Entity Operations Interface to select an entity from the drawing area
4. Modify parameters
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing CNRB data

1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area. Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
**INITIAL VELOCITY DATA (INIVEL)**

**Introduction**

Use this interface to show/create/modify/delete *INITIAL_VELOCITY entities.

**Interface Description**

Show – Show existing *INITIAL_VELOCITY entities
Create – Create new *INITIAL_VELOCITY entities
Modify – Modify existing *INITIAL_VELOCITY entities
Delete – Delete existing *INITIAL_VELOCITY entities

Label – Select label type
Vx/Vy/Vz – Enter initial velocity in x/y/z directions
Vrx/Vry/Vrz – Enter initial rotational velocity in x/y/z directions

[All] – Select all *INITIAL_VELOCITY entities
[None] – Deselect all *INITIAL_VELOCITY entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion
[Done] – Exit Initial Velocity Data interface

[Done] – Exit Constrained Nodal Rigid Body Data interface

**Quick Guides**

**Quick Guide: Different ways of displaying Initial Velocity data**

1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
   - Detail – Displays entity ID and linear velocity components next to entity

**Quick Guide: How to set an Initial Velocity**

1. Select Create
2. Enter linear velocity in X/Y/Z directions
3. Enter rotational velocity in X/Y/Z directions
4. Select nodes using General Selection Interface
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)
Quick Guide: How to modify existing Initial Velocity data
1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Modify parameters
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Initial Velocity data
1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
SPC DATA

Introduction

Use this interface to show/create/modify/delete SPC data.

Interface Description

Show – Show existing SPC nodes
Create – Create new SPC nodes
Modify – Modify existing SPC nodes
Delete – Delete existing SPC nodes

Label – Select label type
X,Y,Z – Toggle translational constraint in local X,Y,Z directions respectively
RX,RY,RZ – Toggle rotational constraint about the local X,Y,Z axes respectively
CID – Open Define_Coordinate_{Option} link dialog

[All] – Select all SPC nodes
[None] – Deselect all SPC nodes
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion
[Done] – Exit SPC Data interface

Quick Guides

Quick Guide: Different ways of displaying SPC data
1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
   - Detail – Displays entity ID and the number of the active constraints next to entity

Quick Guide: How to create an SPC keyword
1. Select Create
2. Set translational constraint directions
3. Set rotational constraint directions
4. Select/Enter CID value
5. Select nodes using General Selection Interface
6. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)
Quick Guide: How to modify SPC data

1. Select SPC data from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area. Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select SPC data from the list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Modify translational constraint directions (X, Y, Z)
5. Modify rotational constraint directions (RX, RY, RZ)
6. Modify CID value
7. Modify node selection using General Selection Interface
8. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing SPC data

1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area. Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
**CONSTRANGED RIVET DATA (RIVET)**

**Introduction**

Use this interface to show/create/modify/delete Constrained Rivet Data.

**Interface Description**

Show – Show existing *CONSTRANGED_ RIVET entities
Create – Create new *CONSTRANGED_ RIVET entities
Modify – Modify existing *CONSTRANGED_ RIVET entities
Delete – Delete existing *CONSTRANGED_ RIVET entities

Label – Select label type
RID – Enter Rivet ID (Optional)
N1 – Pick/Enter node ID 1
N2 – Pick/Enter node ID 2
TF – Enter failure time for nodal constraint set

[All] – Select all *CONSTRANGED_ RIVET entities
[None] – Deselect all *CONSTRANGED_ RIVET entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion
[Done] – Exit Initial Velocity Data interface

[Done] – Exit Constrained Rivet Data interface

**Quick Guides**

**Quick Guide: Different ways of displaying Constrained Rivet data**

1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
   - Detail – Displays entity ID and CR node IDs next to entity
Quick Guide: How to create a Constrained Rivet

1. Select Create
2. Enter RID and TF
3. Autocreate enabled (Enables creation without having to click apply after every entry)
   - Pick/Enter 2 node IDs
Or Autocreate disabled
   - Pick/Enter 2 node IDs
   - Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Rivet data

1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a single entity from the list
   Or Use Select Operation on the Entity Operations Interface to select one from the drawing area
4. Modify RID, TF and CR nodes
5. Autocreate enabled (Enables creation without having to click apply after every entry)
   - Pick/Enter 2 node IDs
Or Autocreate disabled
   - Pick/Enter 2 node IDs
   - Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Constrained Rivet data

1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
(Note to save changes permanently use File/Save Keyword)
CONSTRAINED NODE SET DATA (CONSTN)

Introduction

Use this interface to show/create/modify/delete Constrained Node Set Data.

Interface Description

Show – Show existing *CONSTRAINED_NODE_SET entities
Create – Create new *CONSTRAINED_NODE_SET entities
Modify – Modify existing *CONSTRAINED_NODE_SET entities
Delete – Delete existing *CONSTRAINED_NODE_SET entities

Label – Select label type
RID – Enter Rivet ID (Optional)
N1 – Pick/Enter node ID 1
N2 – Pick/Enter node ID 2
TF – Enter failure time for nodal constraint set

[All] – Select all *CONSTRAINED_NODE_SET entities
[None] – Deselect all *CONSTRAINED_NODE_SET entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Constrained Node Set Data interface

Quick Guides

Quick Guide: Different ways of displaying Constrained Node Set data

1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
Quick Guide: How to create a Constrained Node Set
1. Select Create
2. Enter Constrained Node Set ID and Node Set ID
   Or Click [New ID]
3. Select nodes using General Selection Interface
4. Select DOF value
5. Enter TF value
6. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Constrained Node Set data
1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Modify (Note the list and drawing area will only show previously selected entities)
3. Select a single entity from the list
4. Or Use Select Operation on the Entity Operations Interface to select one from the drawing area
5. Modify parameters
6. Select nodes using General Selection Interface
7. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Constrained Node Set data
1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
DATABASE HISTORY DATA

Introduction

Use this interface to assign database history nodes/beams/shells/solids/sph nodes/tshells.

Interface Description

Show – Show existing *DATABASE_HISTORY_{OPTION} entities
Create – Create new *DATABASE_HISTORY_{OPTION} entities
Modify – Modify existing *DATABASE_HISTORY_{OPTION} entities
Delete – Delete existing *DATABASE_HISTORY_{OPTION} entities

Type – Select database history data type
Label – Select label type

[All] – Select all *DATABASE_HISTORY_{OPTION} entities
[None] – Deselect all *DATABASE_HISTORY_{OPTION} entities
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion

[Done] – Exit Database History Data interface

Quick Guides

Quick Guide: Different ways of displaying Database History data
1. Select Show
2. Select entities from the list to display
3. Select label format
   - None – No label is shown next to entity
   - Symbol – Displays entity ID next to entity
Quick Guide: How to create Database History data
1. Select Create
2. Enter Constrained Node Set ID and Node Set ID
   Or Click [New ID]
3. Select nodes using General Selection Interface
4. Select DOF value
5. Enter TF value
6. Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify existing Database History data
1. Select entities from the list
   (Use Select/Blank Operation on the Entity Operations Interface to select entities from the drawing area.
2. Note these options are only available when Show is selected)
3. Select Modify (Note the list and drawing area will only show previously selected entities)
4. Select/Deselect nodes using General Selection Interface
   (Note: existing entities may also be selected / deselected)
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

Quick Guide: How to delete existing Database History data
1. Select entities from the list
   (Use Blank Operation on the Entity Operations Interface to blank unwanted entities from the drawing area.
   Note these options are only available when Show is selected)
2. Select Delete (Note the list and drawing area will only show previously selected entities)
3. Select entities from the reduced list
   Or Use Select Operation on the Entity Operations Interface to select entities from the drawing area
4. Click [Apply] to delete selected entities
5. Click [Ok] to confirm deletion
   (Note to save changes permanently use File/Save Keyword)
MASS DATA

Introduction

Use this interface to show/create/modify/delete Mass data.

Interface Description

Show – Show existing mass data
Create – Create new mass data
Modify – Modify existing mass data
Delete – Delete existing mass data

Label – Select label type
Mass Value – Enter mass value

[All] – Select all mass nodes
[None] – Deselect all mass nodes
[Rev] – Reverse selection
[AList] – Select all entities within the selected range
[Apply] – Apply entries for creation/modification or selections for deletion
[Done] – Exit Mass Data interface

Quick Guides

Quick Guide: How to create Mass nodes
Select Create
Enter mass value
Select nodes using General Selection Interface
Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)

Quick Guide: How to modify Mass nodes
Select an SPC keyword from the list
Or use Selection on Entity Operations Interface
(This interface is only shown in Show/Delete modes)
Select Modify
Modify mass value
Modify node selection using General Selection Interface
Click [Apply] to save changes to memory
(Note to save changes permanently use File/Save Keyword)
PART DATA

Introduction

Use this interface to show/create/modify/delete Part data.

Interface Description

Show – Show existing part data
Create – Create new part data
Modify – Modify existing part data
Sear – Search for parts using user defined parameters
Assi – Assign entered data to selected parts
Part Type – Select Part Entity type
Pick – Pick a single part
Area – Select entities inside/outside a user defined area
Poly – Select entities inside/outside a user defined region
In – Select entities inside the area/polygon
Out – Select entities outside the area/polygon
[All] – Select all parts
[None] – Deselect all parts
[Rev] – Reverse selection
[Del] – Delete selected parts
[Write] – Write selected parts to a file
[Done] – Exit Part Data interface
**NEW ID** – Generate a new ID for keyword

**Add** - Generate a new ID for keyword and set all fields to default values

**Accept** - Accept entries for current part

**PID** – Part ID

**SECID** – Enter Section ID /Open link dialog by clicking on the label

**MID** – Enter Material ID /Open link dialog by clicking on the label

**EOSID** – Enter Equation of state ID /Open link dialog by clicking on the label

**HGID** – Enter Hourglass/bulk viscosity ID /Open link dialog by clicking on the label

**GRAV** – Select part initialization for gravity loading

**ADPOPT** – Select part adaptivity

**TMID** – Enter Thermal material property ID

**Search**

<table>
<thead>
<tr>
<th>SECID</th>
<th>MID</th>
<th>EOSID</th>
<th>HGID</th>
<th>TMID</th>
<th>GRAV</th>
<th>ADPOPT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

**[Load RefBy]** – Load Part IDs stored in selected reference buffer

**[Buffer n]** – Select buffer to load Part IDs

**Assign**

**Assign Part**  **Apply**
Quick Guides

**Quick Guide: How to create Parts**
1. Select Create
2. Enter Part ID
   Or Click [New ID]/[Add]
3. Select part type
4. Enter keyword data
5. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

**Quick Guide: How to modify Parts**
1. Select a part from the list
   Or use Selection on Entity Operations Interface
   (This interface is only shown in Show/Delete modes)
2. Select Modify
3. Modify Part ID
4. Modify part type
5. Modify keyword data
6. Click [Apply] to save changes to memory
   (Note to save changes permanently use File/Save Keyword)

**Quick Guide: How to use Search**
1. Enter values in fields to search for parts containing those IDs
2. Click [Apply]
   Or
1. Select Buffer
2. Click Load Ref By – Material/EOS/Hourglass keywords that are used in Part keywords can be saved into buffers using the appropriate dialog window on Page 3. (See Page3 instructions) These buffers can be reloaded here to display the parts.

**Quick Guide: How to assign new keyword values to multiple parts**
1. Select parts from list / model
2. Enter new keyword data
3. Click [Apply] to assign new values
   (Note to save changes permanently use File/Save Keyword)
PAGE D: DISPLAY INTERFACE

INTRODUCTION

Use this interface to display entities that are present in the current keyword file.

INTERFACE DESCRIPTION

[All] – Turn on all entities
[None] – Turn off all entities
Select entity type to be displayed

[All] – Turn on all entities in the current list
[None] – Turn off all entities in the current list
[Rev] – Reverse selection
[AList] – Turn on all entities in the selected range only
Label – Select label type. Note: not all entities have a detailed label, in this situation selecting Detail will only display a symbol style label.

The Entity Operation interface can be used, see description under Page 4 chapter.

BOUNDARY

Prescribed motion – Display *BOUNDARY_PRESCRIBED_{OPTION} data
SPC – Display *BOUNDARY_SPC_{OPTION} data
Node_SPC – Display nodal boundary SPC data (See *NODE_NODE data)

CONSTRAINED

ExNodes – Display *CONSTRAINED_EXTRA_NODES_{OPTION} data
GenWeld – Display *CONSTRAINED_GENERALIZED_WELD_SPOT data
SpotWeld – Display *CONSTRAINED_SPOT_WELD data
RgBodies – Display *CONSTRAINED_RIGID_BODY_{OPTION} data
NodalRDB – Display *CONSTRAINED_NODAL_RIGID_BODY data
Joint – Display *CONSTRAINED_JOINT_{OPTION} data
Rivet – Display *CONSTRAINED_RIVET data
Points – Display *CONSTRAINED_POINTS data
NodeS – Display *CONSTRAINED_NODE_SET data
**CONTACT**

General – Display *CONTACT_GENERAL_{OPTION}* data  
Entity – Display *CONTACT_ENTITY* data  
Gebod – Display *CONTACT_GEBOD_{OPTION}* data  
Interior – Display *CONTACT_INTERIOR* data  
RigidSurface – Display *CONTACT_RIGID_SURFACE* data  
1D – Display *CONTACT_1D_{OPTION}* data  
2D – Display *CONTACT_2D_{OPTION}* data

**DEFINE**

Box – Display *DEFINE_BOX_{OPTION}* data  
Coordinate – Display *DEFINE_COORDINATE_{OPTION}* data  
Curve – Display *DEFINE_CURVE_{OPTION}* data  
SD_orientation – Display *DEFINE_SD_ORIENTATION* data  
Vector – Display *DEFINE_VECTOR* data

**INITIAL**

Velocity – Display *INITIAL_VELOCITY_{OPTION}* data

**LOAD**

Beam – Display *LOAD_BEAM_{OPTION}* data  
Body – Display *LOAD_BODY_{OPTION}* data  
Node – Display *LOAD_NODE_{OPTION}* data  
Rigid Body – Display *LOAD_RIGID_BODY* data  
Segment – Display *LOAD_SEGMENT_{OPTION}* data  
Shell – Display *LOAD_SHELL_{OPTION}* data
SET
Beam – Display *SET_BEAM_(OPTION) data
Discrete – Display *SET_DISCRETE_(OPTION) data
Node – Display *SET_NODE_(OPTION) data
Part – Display *SET_PART_(OPTION) data
Segment – Display *SET_SEGMENT_(OPTION) data
Shell – Display *SET SHELL_(OPTION) data
Solid – Display *SET SOLID_(OPTION) data
Tshell – Display *SET TSHELL_(OPTION) data
SPH – Display *SET SPH data

SEAT BELT
Accelerometer – Display *ELEMENT SEATBELT ACCELROMETER data
Pretensioner – Display *ELEMENT SEATBELT PRETENSIONER data
Reactor – Display *ELEMENT SEATBELT REACTOR data
Sensor – Display *ELEMENT SEATBELT SENSOR data
Slipring – Display *ELEMENT SEATBELT SLIPRING data

RIGID WALL
Rigidwall – Display *RIGID WALL_(OPTION) data

QUICK GUIDES
Quick Guide: Displaying entities using Page D
1. Select main entity type (using choice box)
2. Select sub entity group from the top list
3. Select members of that sub-entity group from the bottom list to display
(See Entity Interface for instructions regarding the lower interface)
APPENDIX I

CONFIGURATION FILE OPTIONS

Introduction

The .lspostrc file stores information regarding the LS-Pre/Post configuration. This file is only present if the user saves a configuration that is different from the defaults within LS-Pre/Post.

LS-Pre/Post will look for this file in the following locations:

1. LSTC_FILE – An environment variable defining the directory where the LSTC license key resides
2. $HOME – User home directory
3. WD – working directory where the model resides

LS-Pre/Post searches through these three locations in order, options set in location 1 will be overwritten if found in location 2 and overwritten again if found in location 3. Note this is at the option level and so if different options are set in different locations LS-Pre/Post will use them all instead of defaults. Any options not found in user defined .lspostrc files will be set to default automatically by Ls-Pre/Post.

List of Configuration File Options

max_physical_memory = 1867392K – assign maximum available physical memory for LS-Pre/Post
texture = off – Switch texture mode off/on (only applicable when in shade mode)
twoside_lighting = on – Switch between Two-sided (on) /One-sided (off) lighting
background_color = 0.000, 0.000, 0.000 – Set indicated color using RGB range (0.0 to 1.0)
text_color = 1.000, 1.000, 1.000 – Set indicated color using RGB range (0.0 to 1.0)
mesh_color = 0.000, 0.000, 0.000 – Set indicated color using RGB range (0.0 to 1.0)
label_color = 1.000, 1.000, 1.000 – Set indicated color using RGB range (0.0 to 1.0)
hilite_color = 0.500, 0.500, 0.500 – Set indicated color using RGB range (0.0 to 1.0)
outline_color = 0.000, 0.000, 0.000 – Set indicated color using RGB range (0.0 to 1.0)
triad_color = 1.000, 1.000, 1.000 – Set indicated color using RGB range (0.0 to 1.0)
edgelwidth = 1 – Set edge view line width
meshwidth = 1 – Set mesh view line width
fringe_light = on – Switch fringe component lighting on/off
plotunode = off – Switch plot un-referenced nodes on/off
plotmode = shade – Switch model plot mode shade/view/wire/feat/edge/grid
outlining = edge – Switch outlining type edge/feat/off
shade_mode = flat – Switch shading mode flat/smooth
highlight_node = off – Assign default setting for highlighting nodes under Node Trace, on/off
result_onscreen = off – Switch results on screen off/on (Only applicable if a model entity is identified with show results toggled on)
highlight = on – Switch highlight on/off
mat_ambient = 0.200000 – Assign default value for material light ambience (0.0 to 1.0)
mat_diffuse = 0.200000 – Assign default value for material light diffusion (0.0 to 1.0)
mat_specular = 0.700000 – Assign default value for material light speculance (0.0 to 1.0)
mat_shininess = 50.000000 – Assign default value for material shininess (0.0 to 128.0)
light_ambient.1 = 0.300, 0.300, 0.300 – Assign ambient light RGB values from (0.0 to 1.0)
light_diffuse.1 = 0.700, 0.700, 0.700 – Assign diffuse light RGB values from (0.0 to 1.0)
light_specular.1 = 0.400, 0.400, 0.400 – Assign specular light RGB values from (0.0 to 1.0)
light_position.1 = 0.000, 0.000, 1.000 – Assign light 1 xyz co-ordinates
transparency_value = 0.850 – Assign default transparency value (0.0 = opaque to 1.0 = transparent)
matt_color.1 = 0.769, 0.004, 0.110 – Assign up to 29 different material colors for the color map
fringe_level = 10 – Set default number of fringe levels to display 2 to 30
vect_plotscale = 1.000 – Assign default value for vector arrow scale factor
sph_spherescale = 0.300 – Assign default value for sph sphere scale factor
sph_spheredivs = 12 – Assign default value for number of segments to draw an sph sphere
sph_spherestyle = 4 – Assign default value for sph sphere style,
(1 = dot, 2 = flat, 3 = wire frame polygon, 4 = shiny sphere)
sph_fixedradius = 0 – Assign default radius for sph sphere
shrink_factor = 0.800 – Assign default value for shrink scale factor
rotate_angle = 10.0 – Assign default value for model rotation
feature_angle = 30.0 – Assign angle limit for feature line generation
default_light = 0.300, 0.700, 0.400 – Assign default light color with RGB values from (0.0 to 1.0)
help_label = off – Switch help bubbles off/on
mainmenu_on = right – Set position of main menu right/left
mainwindow_x = 1423 – Set default window size
mainwindow_y = 1068 – Set default window size
plotwindow_x = 0 – Set XY-Plot window start x-coordinate
plotwindow_y = 0 – Set XY-Plot window start y-coordinate
xyplotlegend = on – Switch default setting for XY-Plot window legend on/off
init_path = c:\dynafiles\ – Assign default directory for files to be read (Note: directory must exist)
session_file = lspost.cfile – Assign file name for command file
message_file = lspost.msg – Assign file name for message file
message = off – Switch default setting for message recording off/on
ic_gravity = 9.81 – Assign the value for gravity
ic_timeunit = 1 – Assign the value for standard time unit
drawscene = on – Assign default model drawing option, (on = model drawn upon opening file,
off = model is not drawn upon opening file)
shell_ipt = maxima – Assign default position of shell surface integration points maxima/upper/middle/lower
beam_tubesegs = 8 – Assign number of segments used to draw a beam as a tube
beam_prismon = off – Display beam as prism on/off
beam_sizefactor = 500 – Assign beam prism scale factor
green_bricity = 0 – Assign default value for strain
animate_mode = 0 – Animate mode (0 = loop animation, 1 = animate once)
duplicate_action = 2 – Assign default action when duplicated data is found when reading a keyword file
(0 = ignore, 1 = replace all, 2 = ask user for action)
The following menu page options show current interface button assignments, these can be changed here, however it is easier to use the Assign Menu Button dialog within LS-Pre/Post. (One can assign up to 7 pages):

- menu_page1 = 1,2,3,4,5,6,7,8,9,10,11,12,13,14,15,16,17,18,19,20,21,22,23,24,25,26,27
- polygon_offset_factor = 1 – Assign default polygon outline offset factor
- polygon_offset_unit = 1 – Assign default polygon outline offset unit
- override_screen_aspect = 0 – Assign default window aspect ratio
- fld_engstrain = 1 – Assign default value for FLD engineering strain
- fld_tvalue = 0.8 – Assign default value for FLD t
- fld_nvalue = 0.21 – Assign default value for FLD n
- def_limit_flc = 0 – Assign default value for FLD FLC limit
- def_allow_thinning = 0.3 – Assign default value for FLD allowable thinning
- def_safety_margin = 20 – Assign default value for FLD safety margin
- def_essential_thinning = 0.02 – Assign default value for FLD required thinning
- def_allow_thickening = 0.01 – Assign default value for FLD allowable thickening
- def_r_value = 1.82 – Assign default value for FLD ‘r’
- def_failure = on – Assign default value for FLD failure on/off
- def_risk_fracture = on – Assign default value for FLD risk of fracture on/off
- def_severe_thinning = on – Assign default value for FLD severe thinning on/off
- def_inadequate_stretch = on – Assign default value for FLD lack of stretching on/off
- def_wrinkle_tendency = on – Assign default value for FLD wrinkle tendency on/off
- def_wrinkles = on – Assign default value for FLD wrinkles on/off

* use a backslash at end of line to continue points list on next line

* ______10________20________30________40________50________60________70________80

def_flc = DEF 800521501
-0.3 0.909, -0.2 0.666, -0.1 0.475, 0 0.357, 0.1 0.44, \ 
0.2 0.493, 0.3 0.523, 0.4 0.547, – Define FLC with paired values (separate individual values with a space, separate pairs by comas)

- pscompression_mode = none – Assign default postscript compression mode none/gz/irle/lzw
- entitylabel_size = 4 – Assign default entity label size
- primitive_size = 0.6 – Assign default primitive size factor
- thickenedshell = 0 – Assign default setting for drawing a shell with thickness as a solid (0 = off, 1 = on)