# Table of Contents

**Introduction**  
11

**Model Data Reference**  
13

- **Properties**  
  17
  - **Materials**  
    - Material Name  
      19
    - Basic Isotropic Material Properties  
      19
    - Other Common Basic Properties  
      19
    - Other Material Properties for Time Dependent and Inelastic Analyses  
      20
    - Orthotropic Material Properties  
      20

- **Sections**  
  23
  - General Section Properties  
    23
  - Properties for Time-Dependent Analysis  
    23
  - Properties for Inelastic Analysis  
    24
  - Stress Recovery Points  
    24
  - Section Dimensions  
    24
  - Centroid Offset  
    25

- **Spring Property Definitions**  
  29
  - Nonlinear Elastic Spring Properties  
    29
  - 6x6 Stiffness Matrix Properties  
    29
  - Inelastic (Hysteretic) Spring Properties  
    29
  - Nonlinear Spring Curve  
    30

- **Isolator Property Definitions**  
  33

- **User Coordinate Systems**  
  35
  - The Global Coordinate System  
    35
  - Defining Coordinate Systems  
    35
  - Cylindrical Coordinate Systems  
    37
  - Spherical Coordinate Systems  
    38
  - Bridge Paths  
    39

- **Bridge Paths**  
  43
  - Bridge Axes  
    44
  - Horizontal Geometry  
    44
  - Vertical Geometry  
    46
  - Superstructure Rotation  
    47

- **Time-Dependent Material Property Definitions**  
  49
  - Creep and Shrinkage Properties for CEBFIP-78  
    49
  - Creep and Shrinkage Properties for CEBFIP-90  
    50
  - Other Properties  
    50
  - Material Curves  
    51
  - Relaxation Coefficients  
    51

- **Geometry**  
  53
  - Joints  
    55
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Properties</td>
<td>55</td>
</tr>
<tr>
<td>Translational and Rotational Degrees of Freedom (DOF)</td>
<td>55</td>
</tr>
<tr>
<td>Displacement User Coordinate System</td>
<td>56</td>
</tr>
<tr>
<td>Members</td>
<td>57</td>
</tr>
<tr>
<td>Member Properties</td>
<td>57</td>
</tr>
<tr>
<td>Connection Beam Properties</td>
<td>61</td>
</tr>
<tr>
<td>Properties for Time-Dependent Material Effects</td>
<td>61</td>
</tr>
<tr>
<td>Member Coordinate Systems (Local Axes)</td>
<td>62</td>
</tr>
<tr>
<td>Self-Weight Computation</td>
<td>66</td>
</tr>
<tr>
<td>Spans</td>
<td>69</td>
</tr>
<tr>
<td>Plates</td>
<td>71</td>
</tr>
<tr>
<td>Usage Notes</td>
<td>71</td>
</tr>
<tr>
<td>Element Formulation</td>
<td>71</td>
</tr>
<tr>
<td>Attributes of Plates</td>
<td>73</td>
</tr>
<tr>
<td>Plate Local Axes</td>
<td>74</td>
</tr>
<tr>
<td>Material Axes</td>
<td>75</td>
</tr>
<tr>
<td>Self-Weight Computation</td>
<td>76</td>
</tr>
<tr>
<td>Springs</td>
<td>79</td>
</tr>
<tr>
<td>Usage Notes</td>
<td>79</td>
</tr>
<tr>
<td>Grounded Spring Element</td>
<td>79</td>
</tr>
<tr>
<td>Two-Node Spring Element</td>
<td>79</td>
</tr>
<tr>
<td>Inelastic (Hysteretic) Spring Element</td>
<td>80</td>
</tr>
<tr>
<td>General Attributes</td>
<td>80</td>
</tr>
<tr>
<td>Stiffness Attributes</td>
<td>81</td>
</tr>
<tr>
<td>Isolators</td>
<td>83</td>
</tr>
<tr>
<td>Mass Elements</td>
<td>85</td>
</tr>
<tr>
<td>DOF Constraints</td>
<td>87</td>
</tr>
<tr>
<td>Tendons</td>
<td>89</td>
</tr>
<tr>
<td>About the Tendon</td>
<td>89</td>
</tr>
<tr>
<td>Friction Losses</td>
<td>89</td>
</tr>
<tr>
<td>Anchorage Slip Losses</td>
<td>90</td>
</tr>
<tr>
<td>Elastic Shortening of Concrete Losses</td>
<td>90</td>
</tr>
<tr>
<td>Other Losses</td>
<td>90</td>
</tr>
<tr>
<td>Attributes</td>
<td>90</td>
</tr>
<tr>
<td>Path Geometry</td>
<td>91</td>
</tr>
<tr>
<td>Example Path</td>
<td>96</td>
</tr>
<tr>
<td>Lanes</td>
<td>99</td>
</tr>
<tr>
<td>Basic Properties</td>
<td>99</td>
</tr>
<tr>
<td>Path Geometry</td>
<td>100</td>
</tr>
<tr>
<td>Lane Path Example</td>
<td>102</td>
</tr>
<tr>
<td>Loads</td>
<td>105</td>
</tr>
<tr>
<td>Static Load Cases</td>
<td>107</td>
</tr>
<tr>
<td>Load Classes for Code-Based Results</td>
<td>107</td>
</tr>
<tr>
<td>Load Combinations</td>
<td>109</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Including Response Spectra Cases</td>
<td>109</td>
</tr>
<tr>
<td>Including Moving Load Cases</td>
<td>109</td>
</tr>
<tr>
<td>Use in Nonlinear Analysis</td>
<td>109</td>
</tr>
<tr>
<td>Self Weight</td>
<td>111</td>
</tr>
<tr>
<td>Weight Factor and Gravity Load Directions</td>
<td>111</td>
</tr>
<tr>
<td>Self Weight Computation</td>
<td>111</td>
</tr>
<tr>
<td>In Staged Construction Analysis</td>
<td>111</td>
</tr>
<tr>
<td>Joint Loads</td>
<td>113</td>
</tr>
<tr>
<td>Support Displacements</td>
<td>115</td>
</tr>
<tr>
<td>Member Loads</td>
<td>117</td>
</tr>
<tr>
<td>Member Thermal Loads</td>
<td>119</td>
</tr>
<tr>
<td>Load Input</td>
<td>119</td>
</tr>
<tr>
<td>Nonlinear Thermal Gradient Curves</td>
<td>120</td>
</tr>
<tr>
<td>Plate Loads</td>
<td>121</td>
</tr>
<tr>
<td>Plate Load Fields</td>
<td>121</td>
</tr>
<tr>
<td>Point Load Coordinates</td>
<td>122</td>
</tr>
<tr>
<td>Moving Loads</td>
<td>125</td>
</tr>
<tr>
<td>Influence Loads</td>
<td>127</td>
</tr>
<tr>
<td>Time History Loads</td>
<td>129</td>
</tr>
<tr>
<td>Excitation Functions</td>
<td>129</td>
</tr>
<tr>
<td>Initial Conditions</td>
<td>130</td>
</tr>
<tr>
<td>Construction Activities</td>
<td>131</td>
</tr>
<tr>
<td>Construct and Deconstruct Activities</td>
<td>133</td>
</tr>
<tr>
<td>Deconstruction</td>
<td>133</td>
</tr>
<tr>
<td>Self Weight and Mass</td>
<td>133</td>
</tr>
<tr>
<td>Segmental Construction Methods</td>
<td>134</td>
</tr>
<tr>
<td>Load Activities</td>
<td>137</td>
</tr>
<tr>
<td>Creating Load Activities</td>
<td>137</td>
</tr>
<tr>
<td>Using Load Activities</td>
<td>137</td>
</tr>
<tr>
<td>Self-Weight Load Cases</td>
<td>137</td>
</tr>
<tr>
<td>Support and Hoist Activities</td>
<td>139</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>139</td>
</tr>
<tr>
<td>DOF Constraint Change Activities</td>
<td>141</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>141</td>
</tr>
<tr>
<td>Tendon Stressing and Slackening Activities</td>
<td>143</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>143</td>
</tr>
<tr>
<td>Displacement Initializations</td>
<td>145</td>
</tr>
<tr>
<td>What It Does</td>
<td>146</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>146</td>
</tr>
<tr>
<td>Cast Concrete Activities</td>
<td>147</td>
</tr>
<tr>
<td>Composite Section Construction Activities</td>
<td>149</td>
</tr>
<tr>
<td>Analysis Scenarios</td>
<td>153</td>
</tr>
</tbody>
</table>
Analysis Reference

Static Analysis

Linear Static Analysis
- Structure Model for Linear Static Analysis
- Analysis Results
- Assumptions in a Linear Static Analysis

P-Delta Analysis
- Using Load Combinations for the P-Delta
- Caveats
- Method
- Convergence Criteria

Nonlinear Static Analysis
- Why a Nonlinear Analysis
- Applications of the Nonlinear Analysis
- Equations of Equilibrium and Solution
- Caveats
- Incremental Loading
- Convergence Criteria
- Displacement Convergence and Unbalanced Forces
- Load Case Data for Nonlinear Static Analysis

Nonlinear Buckling Analysis
- Advantages of a Nonlinear Analysis
- Using the Nonlinear Buckling Analysis

Nonlinear Pushover Analysis

Nonlinear Static Analysis Options

Incremental Nonlinear and Nonlinear Buckling Analysis Options

Nonlinear Pushover Analysis Options

Eigenvalue Analysis

Eigenvalue and Stressed Eigenvalue Analysis
- Unstressed (Standard) Eigenvalue Analysis
- Stressed Eigenvalue Analysis

Response Spectra Analysis
- Response Spectra Curve
- Modal Damping
- Modal Combination
- Spatial Combination
- Ground Motion Directions
- Response Spectra Load Cases
- Caveat About Sign
- Analysis Options for Response Spectra

Eigenvalue Analysis Options

Stressed Eigenvalue Analysis Options
Time History Analysis 191
  Linear Time History Analysis 193
  Nonlinear Time History Analysis 195
    Overview 195
    Newmark-Beta with Newton-Raphson 195
    Sparse Solver Technology 195
    Nonlinear Time History Data 195
  Linear Time History Analysis Options 197
  Nonlinear Time-History Analysis Options 199
    Geometric Nonlinearity 199
Live Load Analysis 201
  Moving Load Analysis 203
    Preparing a Moving Load Analysis 203
    Linear Static Moving Load Analysis 203
    Linear Dynamic Rolling Stock Analysis 203
Influence Line & Surface Analysis 205
  Influence Analysis Overview and Options 207
    The Assumption of Linearity 207
    The Vehicle Loading Algorithm 207
    Performing an Influence Analysis 208
    General Loading Parameters 209
    Vehicular Loading Options 212
    Vehicular Options for the 7.01 Compatible Solution Method 216
    Uniform/Patch Loading Options 216
    Procedure: Getting Results 216
  Suggestions for Influence Analysis 219
    Suggestions for Speed 219
  Influence Surfaces in LARSA 4D Version 7.01 221
    Design Lane Options 221
    Vehicle Type and Placement Options 222
  AASHTO LFD Point Loading in LARSA 4D Version 7.01 223
  Standard Vehicles 225
    AASHTO and CALTRANS Vehicles 225
    AASHTO Load Patterns for Influence Analysis 225
    IRC Load Patterns for Influence Analysis 226
Staged Construction Analysis 229
  Overview of Staged Construction Analysis 231
Staged Construction Activities 233
  Setting Up the Model 235
    Preparation 235
    Creating Activities in the Explorer 236
    Creating Activities in the Stage Editor 238
Time Effects on Materials 243
  Material Time Effects 243
  Design Codes 245
Load Class Tracking 247
  Setting up the model 247
  The Load Classes of Result Cases 247
  Accessing Class-Based Results 248
Staged Construction Analysis Options 249
  Options for Analysis Scenarios 250
Solver Options 253

Analysis Results Reference 255
Joint Results 257
  Joint Displacements 259
    Definitions 259
  Joint Reactions 261
    Definitions 261
  Joint Velocities & Accelerations 263
    Definitions 263
Camber Adjustment 265
    Definitions 265
Member Results 267
Member End Forces 269
  Definitions - General 269
  Definitions - When Reported in Global Directions 269
  Definitions - When Reported in Local Directions 269
Member Sectional Forces 271
  Definitions 271
Member Stresses 273
  Computation of Normal Stresses 273
  Definitions 273
Member Displacements 275
  Definitions 275
Member Plastic Deformation 277
  Definitions 277
Member Yield and Strains 279
  Definitions 279
Span Displacements and Forces 281
  Span Displacements 281
  Span Sectional Forces 281
Analyzed Member Loads 283
  Definitions 283
Plate Results

Plate Forces on Center
Definitions 287
Plate Forces at Joints - External
Definitions 289
Plate Forces at Joints - Internal
Definitions 291
Plate Stresses on Center and at Joints
Definitions 293

Spring Results

Spring Forces
Local Axial/Local Torsional Springs 297
Translation/Rotation X/Y/Z Directions 297
Spring Deformations
Local Axial/Local Torsional Springs 299
Translation/Rotation X/Y/Z Directions 299
Spring Yield 301
Connection Beam Yield 303

Tendon Results 305

Eigenvalue Results
Modal Frequencies
Definitions 309
Mode Shapes 311
Definitions 311
Modal Reactions and Modal Member/Plate Forces 313
Introduction

LARSA 4D is the premier general purpose structural analysis and design software. In use throughout the world, LARSA 4D boasts advanced analytical features, from influence surface based analysis to nonlinear time history analysis, and an all-new user interface. LARSA 4D: 4th Dimension, the most advanced program in the LARSA 4D series, features staged construction analysis and time-dependent material properties.

Clients have turned to LARSA for over 25 years for their structural analysis needs. The LARSA structural analysis engine was originally developed to perform nonlinear static analysis of structures with large displacements, such as suspension and cable-stayed bridges. But, LARSA has come a long way since it was first available on the VAX super-mini computers decades ago. Today, LARSA 4D has the only truly 3D analysis engine providing all of the tools segmental bridge and large-scale structures engineers can no longer live without.

This is the LARSA 4D Reference Manual, a part of the series of manuals for LARSA 4D. This manual is split into three sections. In the first, Model Data, LARSA's element library and model definitions are explained. The second section, Analysis Reference, describes how the various types of analysis are performed by the LARSA analysis engine and explains analysis parameters. The last section, Analysis Results, explains how to interpret the results of an analysis.

Separate manuals are available that delve deeper into specific uses of LARSA 4D: staged construction analysis, bridge analysis, and pushover analysis. In addition, the User's Guide explains how to use LARSA 4D's user interface, and the Samples and Tutorials manual provides a hands-on method for learning about the program.
Part I
Model Data Reference

The Model Data Reference details the input data describing the structural geometry, element behavior properties, and loading of the structure.

Properties

<table>
<thead>
<tr>
<th>Section</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Materials</td>
<td>17</td>
</tr>
<tr>
<td></td>
<td>19</td>
</tr>
<tr>
<td>Material Name</td>
<td>19</td>
</tr>
<tr>
<td>Basic Isotropic Material Properties</td>
<td>19</td>
</tr>
<tr>
<td>Other Common Basic Properties</td>
<td>19</td>
</tr>
<tr>
<td>Other Material Properties for Time Dependent and Inelastic Analyses</td>
<td>20</td>
</tr>
<tr>
<td>Orthotropic Material Properties</td>
<td>20</td>
</tr>
<tr>
<td>Sections</td>
<td>23</td>
</tr>
<tr>
<td></td>
<td>23</td>
</tr>
<tr>
<td>General Section Properties</td>
<td>23</td>
</tr>
<tr>
<td>Properties for Time-Dependent Analysis</td>
<td>23</td>
</tr>
<tr>
<td>Properties for Inelastic Analysis</td>
<td>24</td>
</tr>
<tr>
<td>Stress Recovery Points</td>
<td>24</td>
</tr>
<tr>
<td>Section Dimensions</td>
<td>24</td>
</tr>
<tr>
<td>Centroid Offset</td>
<td>25</td>
</tr>
<tr>
<td>Spring Property Definitions</td>
<td>29</td>
</tr>
<tr>
<td></td>
<td>29</td>
</tr>
<tr>
<td>Nonlinear Elastic Spring Properties</td>
<td>29</td>
</tr>
<tr>
<td>6x6 Stiffness Matrix Properties</td>
<td>29</td>
</tr>
<tr>
<td>Inelastic (Hysteretic) Spring Properties</td>
<td>29</td>
</tr>
<tr>
<td>Nonlinear Spring Curve</td>
<td>30</td>
</tr>
<tr>
<td>Isolator Property Definitions</td>
<td>33</td>
</tr>
<tr>
<td>User Coordinate Systems</td>
<td>35</td>
</tr>
<tr>
<td></td>
<td>35</td>
</tr>
<tr>
<td>The Global Coordinate System</td>
<td>35</td>
</tr>
<tr>
<td>Defining Coordinate Systems</td>
<td>35</td>
</tr>
<tr>
<td>Cylindrical Coordinate Systems</td>
<td>37</td>
</tr>
<tr>
<td>Spherical Coordinate Systems</td>
<td>38</td>
</tr>
<tr>
<td>Bridge Paths</td>
<td>39</td>
</tr>
<tr>
<td>Bridge Paths</td>
<td>43</td>
</tr>
<tr>
<td></td>
<td>44</td>
</tr>
<tr>
<td>Bridge Axes</td>
<td>44</td>
</tr>
<tr>
<td>Horizontal Geometry</td>
<td>44</td>
</tr>
<tr>
<td>Vertical Geometry</td>
<td>46</td>
</tr>
<tr>
<td>Superstructure Rotation</td>
<td>47</td>
</tr>
<tr>
<td>Time-Dependent Material Property Definitions</td>
<td>49</td>
</tr>
<tr>
<td></td>
<td>49</td>
</tr>
<tr>
<td>Creep and Shrinkage Properties for CEBFIP-78</td>
<td>49</td>
</tr>
<tr>
<td>Topic</td>
<td>Page</td>
</tr>
<tr>
<td>------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Creep and Shrinkage Properties for CEBFIP-90</td>
<td>50</td>
</tr>
<tr>
<td>Other Properties</td>
<td>50</td>
</tr>
<tr>
<td>Material Curves</td>
<td>51</td>
</tr>
<tr>
<td>Relaxation Coefficients</td>
<td>51</td>
</tr>
<tr>
<td>Geometry</td>
<td>53</td>
</tr>
<tr>
<td>Joints</td>
<td>55</td>
</tr>
<tr>
<td>General Properties</td>
<td>55</td>
</tr>
<tr>
<td>Translational and Rotational Degrees of Freedom (DOF)</td>
<td>55</td>
</tr>
<tr>
<td>Displacement User Coordinate System</td>
<td>56</td>
</tr>
<tr>
<td>Members</td>
<td>57</td>
</tr>
<tr>
<td>Member Properties</td>
<td>57</td>
</tr>
<tr>
<td>Connection Beam Properties</td>
<td>61</td>
</tr>
<tr>
<td>Properties for Time-Dependent Material Effects</td>
<td>61</td>
</tr>
<tr>
<td>Member Coordinate Systems (Local Axes)</td>
<td>62</td>
</tr>
<tr>
<td>Self-Weight Computation</td>
<td>66</td>
</tr>
<tr>
<td>Spans</td>
<td>69</td>
</tr>
<tr>
<td>Plates</td>
<td>71</td>
</tr>
<tr>
<td>Usage Notes</td>
<td>71</td>
</tr>
<tr>
<td>Element Formulation</td>
<td>71</td>
</tr>
<tr>
<td>Attributes of Plates</td>
<td>73</td>
</tr>
<tr>
<td>Plate Local Axes</td>
<td>74</td>
</tr>
<tr>
<td>Material Axes</td>
<td>75</td>
</tr>
<tr>
<td>Self-Weight Computation</td>
<td>76</td>
</tr>
<tr>
<td>Springs</td>
<td>79</td>
</tr>
<tr>
<td>Usage Notes</td>
<td>79</td>
</tr>
<tr>
<td>Grounded Spring Element</td>
<td>79</td>
</tr>
<tr>
<td>Two-Node Spring Element</td>
<td>79</td>
</tr>
<tr>
<td>Inelastic (Hysteretic) Spring Element</td>
<td>80</td>
</tr>
<tr>
<td>General Attributes</td>
<td>80</td>
</tr>
<tr>
<td>Stiffness Attributes</td>
<td>81</td>
</tr>
<tr>
<td>Isolators</td>
<td>83</td>
</tr>
<tr>
<td>Mass Elements</td>
<td>85</td>
</tr>
<tr>
<td>DOF Constraints</td>
<td>87</td>
</tr>
<tr>
<td>Tendons</td>
<td>89</td>
</tr>
<tr>
<td>About the Tendon</td>
<td>89</td>
</tr>
<tr>
<td>Friction Losses</td>
<td>89</td>
</tr>
<tr>
<td>Anchorage Slip Losses</td>
<td>90</td>
</tr>
<tr>
<td>Elastic Shortening of Concrete Losses</td>
<td>90</td>
</tr>
<tr>
<td>Other Losses</td>
<td>90</td>
</tr>
<tr>
<td>Attributes</td>
<td>90</td>
</tr>
<tr>
<td>Path Geometry</td>
<td>91</td>
</tr>
<tr>
<td>Example Path</td>
<td>96</td>
</tr>
<tr>
<td>Lanes</td>
<td>99</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Basic Properties</td>
<td>99</td>
</tr>
<tr>
<td>Path Geometry</td>
<td>100</td>
</tr>
<tr>
<td>Lane Path Example</td>
<td>102</td>
</tr>
<tr>
<td>Loads</td>
<td>105</td>
</tr>
<tr>
<td>Static Load Cases</td>
<td>107</td>
</tr>
<tr>
<td>Load Classes for Code-Based Results</td>
<td>107</td>
</tr>
<tr>
<td>Load Combinations</td>
<td>109</td>
</tr>
<tr>
<td>Including Response Spectra Cases</td>
<td>109</td>
</tr>
<tr>
<td>Including Moving Load Cases</td>
<td>109</td>
</tr>
<tr>
<td>Use in Nonlinear Analysis</td>
<td>109</td>
</tr>
<tr>
<td>Self Weight</td>
<td>111</td>
</tr>
<tr>
<td>Weight Factor and Gravity Load Directions</td>
<td>111</td>
</tr>
<tr>
<td>Self Weight Computation</td>
<td>111</td>
</tr>
<tr>
<td>In Staged Construction Analysis</td>
<td>111</td>
</tr>
<tr>
<td>Joint Loads</td>
<td>113</td>
</tr>
<tr>
<td>Support Displacements</td>
<td>115</td>
</tr>
<tr>
<td>Member Loads</td>
<td>117</td>
</tr>
<tr>
<td>Member Thermal Loads</td>
<td>119</td>
</tr>
<tr>
<td>Load Input</td>
<td>119</td>
</tr>
<tr>
<td>Nonlinear Thermal Gradient Curves</td>
<td>120</td>
</tr>
<tr>
<td>Plate Loads</td>
<td>121</td>
</tr>
<tr>
<td>Plate Load Fields</td>
<td>121</td>
</tr>
<tr>
<td>Point Load Coordinates</td>
<td>122</td>
</tr>
<tr>
<td>Moving Loads</td>
<td>125</td>
</tr>
<tr>
<td>Influence Loads</td>
<td>127</td>
</tr>
<tr>
<td>Time History Loads</td>
<td>129</td>
</tr>
<tr>
<td>Excitation Functions</td>
<td>129</td>
</tr>
<tr>
<td>Initial Conditions</td>
<td>130</td>
</tr>
<tr>
<td>Construction Activities</td>
<td>131</td>
</tr>
<tr>
<td>Construct and Deconstruct Activities</td>
<td>133</td>
</tr>
<tr>
<td>Deconstruction</td>
<td>133</td>
</tr>
<tr>
<td>Self Weight and Mass</td>
<td>133</td>
</tr>
<tr>
<td>Segmental Construction Methods</td>
<td>134</td>
</tr>
<tr>
<td>Load Activities</td>
<td>137</td>
</tr>
<tr>
<td>Creating Load Activities</td>
<td>137</td>
</tr>
<tr>
<td>Using Load Activities</td>
<td>137</td>
</tr>
<tr>
<td>Self-Weight Load Cases</td>
<td>137</td>
</tr>
<tr>
<td>Support and Hoist Activities</td>
<td>139</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>139</td>
</tr>
<tr>
<td>DOF Constraint Change Activities</td>
<td>141</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>141</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Tendon Stressing and Slackening Activities</td>
<td>143</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>143</td>
</tr>
<tr>
<td>Displacement Initializations</td>
<td>145</td>
</tr>
<tr>
<td>What It Does</td>
<td>146</td>
</tr>
<tr>
<td>Activity Fields</td>
<td>146</td>
</tr>
<tr>
<td>Cast Concrete Activities</td>
<td>147</td>
</tr>
<tr>
<td>Composite Section Construction Activities</td>
<td>149</td>
</tr>
<tr>
<td>Analysis Scenarios</td>
<td>153</td>
</tr>
</tbody>
</table>
Properties

Property data are specifications of element behavioral properties that are used for one or more elements in the structure. These include materials, sections, nonlinear spring behavior definitions, isolator and bearing property definitions, and time-dependent material property definitions. User coordinate systems are also defined in this section.

Materials
- Material Name 19
- Basic Isotropic Material Properties 19
- Other Common Basic Properties 19
- Other Material Properties for Time Dependent and Inelastic Analyses 20
- Orthotropic Material Properties 20

Sections
- General Section Properties 23
- Properties for Time-Dependent Analysis 23
- Properties for Inelastic Analysis 24
- Stress Recovery Points 24
- Section Dimensions 24
- Centroid Offset 25

Spring Property Definitions
- Nonlinear Elastic Spring Properties 29
- 6x6 Stiffness Matrix Properties 29
- Inelastic (Hysteretic) Spring Properties 29
- Nonlinear Spring Curve 30

Isolator Property Definitions 33

User Coordinate Systems
- The Global Coordinate System 35
- Defining Coordinate Systems 35
- Cylindrical Coordinate Systems 37
- Spherical Coordinate Systems 38
- Bridge Paths 39

Bridge Paths
- Bridge Axes 44
- Horizontal Geometry 44
- Vertical Geometry 46
- Superstructure Rotation 47

Time-Dependent Material Property Definitions
- Creep and Shrinkage Properties for CEBFIP-78 49
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Creep and Shrinkage Properties for CEBFIP-90</td>
<td>50</td>
</tr>
<tr>
<td>Other Properties</td>
<td>50</td>
</tr>
<tr>
<td>Material Curves</td>
<td>51</td>
</tr>
<tr>
<td>Relaxation Coefficients</td>
<td>51</td>
</tr>
</tbody>
</table>
Materials

Material properties set the elastic modulus, unit weight, and other isotropic, orthotropic, hysteretic, and time-dependent material properties for members, plates, and tendons.

Material properties most often define isotropic linear elastic behavior using elastic modulus, shear modulus, and unit weight. Materials can also hold orthotropic material properties when assigned to plate elements. Time-dependent material properties (page 243) can also be set when used with member and plate elements in a Time-Dependent Staged Construction Analysis. And when assigned to hysteretic beam elements (page 57), additional properties can be used to define inelastic behavior.

The following material properties can be set:

**Material Name**

**Name**

A material's name is used to refer to the material throughout the project.

**Basic Isotropic Material Properties**

The behavior of an isotropic material does not depend on the direction of loading or the orientation of the material. Shearing behavior is uncoupled from extensional behavior.

**Modulus of Elasticity**

Young's Modulus (Elastic Modulus) of the material.

**Poisson Ratio**

Poisson ratio of the material. The range from 0.0 to 0.5 is common, but the ratio must be less than 0.50.

**Shear Modulus**

Shear Modulus of the material. The Poisson ratio can be calculated using the Young's modulus and shear modulus. When any two of the Young's Modulus, Poisson ratio, and shear modulus are specified, the third is automatically calculated.

**Other Common Basic Properties**

**Unit Weight**

Weight density (weight per unit volume) of the material. The self-weight of elements in a static analysis and the mass due to self-weight in a dynamic analysis are computed using this entry. If the unit weight is not entered (zero), the element is assumed weightless.

**Coefficient of Thermal Expansion**

The coefficient of thermal expansion is used when thermal loadings are specified for the structure. If there are no thermal loads, this field is optional. The unit of this field is x10^6, meaning a value of 6.5x10^6 should be entered as simply 6.5.
Other Material Properties for Time Dependent and Inelastic Analyses

These fields are used for steel design, time dependent analyses and inelastic elements in nonlinear analyses.

**Yield Stress**

The yield stress is used to compute the plastic moment capacity for beams. The plastic moment capacity is computed as the plastic section modulus times the material yield stress. In pushover analysis, element stiffness is reduced by the Post-Yield to Initial Slope Ratio after this point is reached.

**Post-Yield to Initial Slope Ratio**

This entry defines the slope of the stress-strain curve after the material yields. The field is the ratio of the post-yield stress-strain slope to the pre-yield stress-strain slope.

**Concrete Strength Specimen**

The specimen type used to get the 28 day strength of the material. It can be either cylinder or cube.

**Concrete Fck or Steel Fu**

The meaning of this column depends on whether the material is concrete or steel.

When the material is concrete, this entry holds the characteristic (also known as specified) strength of concrete (Fck) and is used in Time-Dependent Staged Construction Analysis creep and shrinkage and in the Ultimate Strength tool.

When the material is still, this entry holds Fu, which is used in steel design.

**Concrete Cement Hardening Type**

Cement hardening type to be used in a time-dependent analysis. If the material is not subject to creep and shrinkage, select Not Concrete.

**Tendon GUTS (Guaranteed Ultimate Tensile Stress)**

Required data for tendons. This is the guaranteed ultimate yield stress.

**Material Time-Effect**

This entry determines long-term material time effects, such as relaxation in tendons, creep and shrinkage in a concrete members and also the elastic modulus variation of the all elements as a function of age, that members or tendons assigned this material are subject to during the Staged Construction Analysis (page 229) (see Time Effects on Materials (page 243)). Choices are drawn from time-dependent material property definitions (page 49). A time-dependent material property definition can also be assigned to a member via its section (page 23). It is invalid to have a member for which both its material and its section have been assigned a time-dependent material property definition.

**Orthotropic Material Properties**

Although the plate element typically uses an isotropic material, it can also be assigned an orthotropic material with properties that vary in the element’s two transverse dimensions.

Sheet metal formed by squeezing thick sections of metal between heavy rollers is an example of an orthotropic material. The squeezing flattens and stretches the grain structure of the metal, as a result making it anisotropic. Its properties differ between the direction it was compressed in and each of the two transverse directions. Some other orthotropic materials are wood, laminated plastics, and reinforced concrete. Orthotropic steel bridge deck plates can be modeled using orthotropic materials with plate elements.
For more information about how orthotropic materials apply to plate elements, see Plates (page 71). To create an orthotropic material, the following properties are used:

**Material UCS**
A user coordinate system that determines the material axes of the plates to which the material is assigned. If not set, the Global Coordinate System is used. See Plates (page 71) for more information about material axes.

**E11**
Modulus of Elasticity in the direction of the Material 1 axis.

**E22**
Modulus of Elasticity in the direction of the Material 2 axis

**m12**
Poisson ratio that describes the amount of deformation in the Material 2 direction when a unit deformation occurs in the Material 1 direction. The Poisson ratio in the orthogonal direction, i.e. m21, is automatically calculated from E11, E22, and m12.

**G12**
In-plane shear modulus.

**G13**
Out-of-plane shear modulus of the element’s face whose outward unit normal is parallel to Material 1 axis. Applies to the thick plate element only.

**G23**
Out-of-plane shear modulus of the element’s face whose outward unit normal is parallel to Material 2 axis. Applies to the thick plate element only.

For More Information, please refer to the following documentation.

- Members on page 57.
- Time Effects on Materials on page 243.
- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
Sections

Section data is for describing the geometric and analytic cross-sectional properties of members. Section properties are assigned to members.

The section property data is arranged in three groups:

**General Properties**
The first group is for the most often used property data in the analysis such as area and inertias.

**Stress Recovery Points**
The second group is for stress recovery points, which are the coordinates of the extreme points or edge points on the section. These points are also used for defining tendon and lane paths.

**Section Dimensions**
The last group is for physical section dimensions, which is used for graphical rendering and computation of section properties.

All section properties are entered with respect to the reference coordinate system of the member (see Members (page 57)).

**General Section Properties**

**Name**
A section's name is used to identify the section throughout the project.

**Section Area**
The gross cross-sectional area of the section for axial stiffness. Since truss and cable elements have only axial stiffness, the cross-sectional area is the only property needed for truss and cable elements.

**Shear Areas**
The shear areas $A_y$ and $A_z$ are for transverse shear in the xy- and xz-planes with corresponding transverse shear stiffness as $A_yG$ and $A_zG$.

**Torsional Constant**
Torsional constant is the area moment about the member x-axis for torsional stiffness.

**Moment of Inertias Iyy and Izz**
These are the area moments of inertia $I_{yy}$ and $I_{zz}$ about the member local y- and z-axes, respectively. These values are used for the bending stiffness of beam elements.

**Properties for Time-Dependent Analysis**

**Perimeter**
The perimeter of the section. This is used to determine the notional thickness of the section.

**Material Time-Effect**
This entry determines long-term material time effects, such as creep and shrinkage in a concrete members and the elastic modulus variation of the all elements as a function of age, that members assigned this section are subject to during the Staged Construction Analysis (page 229) (see Time Effects on Materials (page 243)). Choices are drawn from time-dependent material property definitions (page 49). A time-dependent material property definition can also be assigned to a member via its material (page 19). It is invalid to have a member for which both its material and its section have been assigned a time-dependent material property definition.

Properties for Inelastic Analysis

**Plastic Section Modulus Zyy and Zzz**

Plastic section moduli are used for calculating the plastic moment capacity of beam elements in nonlinear inelastic analysis. Plastic section modulus Zyy is for yield moment My where My(yield) = Zyy*Fy (bending about y) and plastic section modulus Zzz is for yield moment Mz where Mz(yield) - Zzz*Fy (bending about z).

**Ductility**

The ductility factor. This is used in an inelastic nonlinear analysis to determine hysteretic beam behavior.

**Residual Strength (%)**

The residual strength used in nonlinear analysis for the property of hysteretic beams.

Additionally, a yield surface can be set for the second. See below for setting the yield surface.

**Stress Recovery Points**

The stress recovery points are y and z coordinates in the member reference coordinate system (page 57) for points on the outer edges of the section where the user desires stresses (page 273) to be reported at. Six stress recovery points can be entered directly on sections spreadsheet. More stress recovery points can be specified using the special commands (see below).

The coordinates of these points are specified in the member reference coordinate system using the proper sign convention with respect to the member's axes. See Members (page 57). When the section has a centroid offset (see below), stress recovery points are relative to the reference axis, not the centroid.

If the coordinates for the stress recovery points are not specified, only the stresses at the centroid of the section (y=0 and z=0) are reported.

These points are also used in linear thermal gradient loads (page 119), locating tendons (page 89) in beams when tendon locations are specified with reference to the top/bottom or left/right edges of the section, and in lane (page 99) definitions with y/z offsets. The first stress point is assumed to be at the top right corner of the section, in the positive-y/positive-z quadrant, and the third stress points at bottom left, in the negative-y/negative-z quadrant.

The location of the stress recovery points are computed automatically when the Custom Section utility is used or when sections are imported from a database. When sections are imported from the Section Composer [see “LARSA Section Composer” in LARSA Section Composer Manual], stress recovery points are set in the Section Composer, not in LARSA 4D.

**Section Dimensions**

These are the dimensions used in the Custom Sections tool when computing properties from dimensions or when sections are imported from a database. These dimensions are used for graphically rendering the actual shape.
The types of dimension measurements for a section vary according to the shape.

**Centroid Offset**

Sections defined in the Section Composer [see “LARSA Section Composer” in *LARSA Section Composer Manual*] may not have their COG lined up with the member reference x-axis, which is the joint-to-joint line for the member (plus member end offsets). When this is the case, the section is said to have a centroid offset. In these cases, the member local axes, which are at the COG, do not match the member reference axes. The local axes are used in the placement of member loads (page 117) and in the reporting of member end forces (page 269), sectional forces (page 271), and stresses (page 273). The reference axes, however, are used in the definitions of tendons (page 89) and lanes (page 99).

**Additional Commands**

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Sections spreadsheet, the commands are available in the Sections menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

The additional commands are as follows:

**Calculate Properties**

When the section shape and dimensions are given in the Section Dimensions spreadsheet, this command computes the analytic properties of the cross-section based on the shape and dimensions provided. The properties computed are: area, Iyy, Izz, J, shear areas, perimeter, and plastic section moduli. This command must be activated whenever the shape or dimensions are changed to update the analytic properties.

**Edit Parameters**

When a section has been imported from the Section Composer, this command brings up a window where the geometric parameters used to define the section can be viewed and edited. Nonprismatic variation and the analytic properties at points along the span can also be inspected.

**Edit Yield Surface**

Opens a spreadsheet to edit the yield surface of the section, which is used to determine the behavior of inelastic hysteretic members (page 57). The yield surface is defined by planar surfaces given by the user. The resulting yield surface is the surface of the volume enclosed by the planes. Each plane is given as 1) a vector <a,b,c> normal to the plane and 2) an offset distance d from the origin to the plane, in the direction of the normal vector. This defines the plane ax + by + cz + d = 0. The component a corresponds to the bending moment about the local z axis of the member. The component b corresponds to the bending moment about the local y axis of the member. And the component c corresponds to the axial force in the member.

As a simplified example, a cube is made of six planar surfaces with the vectors and offsets shown in the image below.
More Stress Points

This command is available in the Section Stress Recovery Points spreadsheet and allows the user to enter more than six stress recovery points.

Section Fiber Width vs Depth

This will be used for nonlinear temperature gradients.

Section Fiber Depth vs Width

This will be used for nonlinear temperature gradients.

Rebars

This command opens a spreadsheet to define the rebars present in the section. Rebar information will be used for concrete design.

Moment (y) Curvature

For hysteretic connection beam members (see the member element (page 57)), a family of moment-curvature spring curves can be entered to specify the inelastic behavior of the beam. With this method, the user is responsible for determining the moment-curvature relation data for the member at different axial loads. Moment curvature relations are entered as spring curve definitions (page 29) with the type field set to Moment curvature. Additionally, the Axial Force field of the spring curve is set to the axial force at which the curve is applicable. The Moment (y) Curvature tool opens a spreadsheet where the family of moment curvature curves for y-moment can be chosen.

Moment (z) Curvature

See the description of the Moment (y) Curvature tool. This tool opens a spreadsheet where the family of moment curvature curves for z-moment can be chosen.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User's Manual.
- Members on page 57.
- Tendons on page 89.
• LARSA Section Composer in *LARSA Section Composer Manual*. 
Spring Property Definitions

Spring property definitions set the behavior of one or more spring elements used in the model. Spring property definitions are applied to spring elements (page 79) to give the elements their nonlinear, hysteretic, or 6x6 matrix-based material properties. Linear springs do not need to be assigned spring property definitions.

The material properties of nonlinear elastic springs and nonlinear hysteretic springs are load dependent with nonlinear behavior and are described using tabular functions. The (6x6) Stiffness property is defined using a matrix. These properties are entered here and then assigned to the springs.

Spring property data common to all types of spring definitions consists of the following:

**Name**

The name is used to refer to the spring property definition throughout the project.

**Type**

The spring property type can be one of the following:

- Nonlinear Elastic Curve - Translational (Displacement vs. Force)
- Nonlinear Elastic Curve - Rotational (Rotation vs. Moment)
- 6x6 Stiffness (K) Matrix
- Nonlinear Inelastic (Hysteretic) Curve - Translational (Displacement vs. Force)
- Nonlinear Inelastic (Hysteretic) Curve - Rotational (Rotation vs. Moment)
- Moment Curvature (Curvature vs. Moment)

**Nonlinear Elastic Spring Properties**

In addition to the above fields, a spring curve must be entered for nonlinear elastic springs. Spring curves are described below.

Nonlinear elastic spring property definitions are applicable for nonlinear grounded springs and nonlinear two-node springs.

**6x6 Stiffness Matrix Properties**

In addition to the above fields, a stiffness matrix must be entered for 6x6 Stiffness Matrix definitions.

6x6 stiffness matrix spring property definitions are applicable for linear grounded springs only.

**Inelastic (Hysteretic) Spring Properties**

These parameters are for inelastic hysteretic springs only and specify the stiffness of the spring after yielding, during unloading, and in any following loading-unloading cycles. A simple bilinear material model has no stiffness or strength degradation and no pinching and can be achieved with appropriate values for these parameters.
To set values beyond for a simple bilinear model, you may need to create a simple one-spring model and graph its
force-displacement function to determine the effects of the parameters.

**Backbone**
Either bilinear or trilinear. Bilinear springs are defined using 4 points in a spring curve and trilinear springs are
defined using 6 points. Trilinear curves can accommodate cracking in addition to yielding.

**Polygonal Hysteretic Model**
The type of cyclic behavior: bilinear, vertex-oriented, or yield-oriented. If Backbone is bilinear, then this field
must also be bilinear.

**Alpha**
Stiffness degradation parameter. This describes the decrease of elastic stiffness during and after the first
unloading, as a function of the plastic deformation at the point of unloading. Values of 2,000 or greater
cause negligible stiffness degradation, as in a simple bilinear model. Smaller values imply higher stiffness
degradation, i.e. a smaller slope in the force-displacement curve.

**Beta 1**
Ductility-based strength degradation parameter. This describes the reduction in strength ("yield capacity," the
force at which yielding begins) as a function of the degree of (plastic) deformation. A value of zero causes no
stiffness degradation due to plastic deformation, as in a simple bilinear model. Values greater than zero cause
stiffness degradation, and the value 1.0 causes rapid degradation.

**Beta 2**
Energy-based strength degradation parameter. This describes the reduction in strength as a function of energy
dissipated throughout the cycles of loading and unloading. Roughly, then, this is the degradation as a function
of the number of loading and unloading cycles. A value of zero causes no stiffness degradation due to energy
dissipation, as in a simple bilinear model. Values greater than zero cause stiffness degradation, and the value
1.0 causes rapid degradation.

**Gamma**
Slip parameter. This controls the amount of pinching. A value of zero causes no slip of pinching, as in a simple
bilinear model. Values greater than zero cause pinching. The value 1.0 causes rapid pinching.

In addition to the above fields, a spring curve must be entered for inelastic springs. Spring curves are described below.
Do not enter point (0,0) in the curve data as the compressive and tensile stiffnesses must be equal in the pre-yield range.

Inelastic spring property definitions are applicable for inelastic (two-node) springs only.

For more information, refer to: M.V. Sivaselvan and A.M. Reinhorn. (1999). "Hysteretic Models for Cyclic Behavior
of Deteriorating Inelastic Structures." Technical Report MCEER-99-0018, University at Buffalo, State University of
New York.

**Nonlinear Spring Curve**
A tabular function describes the force-deformation curve of nonlinear elastic and inelastic springs.

These curves can be for both translational and rotational springs. The horizontal axis of the curve is the deformation,
either in linear or angular dimensions, and the vertical axis is the force or moment.

For nonlinear elastic springs, the curve data can be of any shape and with any number of data points, but hysteretic
spring curves have restrictions on their shape and number of points. These restrictions are described above. Generally,
springs have positive stiffness, which appears as an upward-sloping function.
All spring curves must have a point on both sides of (0,0). Hysteretic spring curves prohibit the (0,0) point, as described above.

For curves assigned to two-node springs acting in the axial direction, the sign convention is as follows: Extensions and tensile forces have positive signs; Shortening and compressive forces have negative signs.

For curves assigned to two-node springs acting in the torsional direction, positive displacement corresponds to positive rotation of the end joint relative to the start joint, following the right-hand rule, about the joint-to-joint line.

For curves assigned to two-node springs acting in the translational or rotational x, y, or z directions, positive displacement corresponds to positive displacement or rotation of the end joint relative to the start joint in that direction. That is, positive displacement does not necessarily correspond with tension or extension. Rather, it follows the relative displacement of the end joint compared to the start joint.

For curves assigned to grounded (i.e. one-node) springs, the sign convention is as follows: Positive displacements will produce positive forces and negative reactions. That is, if the joint connected to a grounded spring displaces in the positive-z direction, then the force in the spring will be determined from the positive side of the spring curve.

The stiffness $k$ of a spring in the tangent stiffness matrix is the slope of the curve and varies as a function of the deformations in the spring. The value is computed in LARSA in the following manner:

- Assume an iteration in nonlinear analysis is performed and joint displacements are known for this iteration.
- Deformation (elongation or rotational deformation) of the element is computed using the nodal displacements of the element.
- With deformation known, the spring force is determined from the tabular deformation-force curve assigned as the material property. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end.
- Note that the familiar rule of linear elastic spring $\text{FORCE} = K \times \Delta L$ does not apply to nonlinear springs.
- The stiffness is computed as the slope of the curve corresponding to the computed deformation where x-axis is the deformation.
- The spring force and the new stiffness is used in the next iteration of the nonlinear analysis to establish the unbalanced load vector and tangent stiffness matrix of the structure.

**Additional Commands**

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Springs spreadsheet, the commands are available in the Springs menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

The additional commands are as follows:

**Edit Curve**

This command opens a new spreadsheet window where the spring curve can be modified and viewed as a graph. The curve is entered as a list of points on the curve. The analysis engine will linearly interpolate the curve to find intermediate values. Special restrictions on the number and positions of the points may apply depending on the curve type (see above).
Edit Stiffness Matrix

When the spring property type is 6x6 Stiffness Matrix, this command opens a new spreadsheet window where the 6x6 stiffness matrix can be entered. The matrix must be symmetric, so only the upper half of the matrix is editable.

Edit Damping Matrix

When the spring property type is 6x6 Damping Matrix, this command opens a new spreadsheet window where the 6x6 damping matrix can be entered. The matrix must be symmetric, so only the upper half of the matrix is editable.

For More Information, please refer to the following documentation.

- Springs on page 79.
- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
Isolator Property Definitions

Isolator property definitions set the behavior for one or more isolator elements used in the model. These property definitions are assigned to isolator elements (page 83) to give the elements their behavioral properties.

An isolator element has the behavior of a dashpot, such as a viscous damping device. These devices are widely used as structural protective systems for extreme loading (wind and earthquake) cases. Due to their combined benefits in terms of overall displacement reduction and energy dissipation, dampers are considered to be one of the most effective structural protective systems.

The dashpot element in LARSA is suitable for modeling the behavior of fluid viscous dampers or other devices displaying viscous behavior. This element can only be used in the time history analysis (and only in the nonlinear time history analysis before LARSA 4D version 8.0) because the response of the element is velocity-dependent.

The dashpot is controlled by a damping coefficient $C$ and a velocity power $n$. The force in the dashpot, $F$, is a function of the velocity across the element ($v$, the difference in velocity at the end joints), where $F = Cv^n$.

Fluid dampers which operate on the principle of fluid orificing, produce an output force which is proportional to the power of the velocity. That power $n$ can take values in the range of 0.5 to 2.0. This element has linear behavior when $n=1$ (but note that the element cannot be used in a linear time history analysis before LARSA 4D version 8.0).

It should be noted that when the velocity exponent $n$ is not 1.0, then the damping coefficient and velocity power must be chosen with respect to a particular choice of units. Care should be taken for units of $C$ and $v$ when using nonlinear damping properties.

The following fields are required for dashpot elements:

**Name**
The name is used to refer to the property definition throughout the project.

**Class**
Currently only Dashpot is supported.

**Damping Coefficient**
The damping coefficient, $C$, used in time history analysis. The coefficient is in units of force-per-velocity. If the isolator property definition is used on an isolator element whose direction is rotational, the unit is given in units of moment per units of velocity.

**Velocity Power**
The power that the velocity is raised to, typically in the range of 0.5 to 1.2. If the value is 1.0, then the element has linear behavior (see above).

For More Information, please refer to the following documentation.

- Isolators on page 83.
- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
User Coordinate Systems

User coordinate systems (UCSs) are used to describe the geometry of the structure in alternative coordinate systems and also to specify the directions of joint degrees of freedom, loads and displacements applied at joints, and the orientation of springs and isolators. There are four types of UCSs: rectangular, cylindrical, spherical, and bridge path.

In most structures, the coordinates of the joints are specified in the Global Coordinate System and joint degrees of freedom, loads and displacements applied at joints, and the orientation of springs and isolators follow the global axes. However, when a user coordinate system is assigned to the Displacement Coordinate System property of a joint (see Joints (page 55)), then the directions of the joint's degrees of freedom, and loads and displacements applied at that joint, are with respect to the user coordinate system. In addition, the orientation of springs (page 79) and isolators (page 83) connected to that joint may be affected.

The use of the additional user coordinate systems is merely a user convenience. The structures with curve beams, tunnels, domes, or structures with inclined supports can be modeled much easier using multiple displacement coordinate systems. For example, it is more convenient to specify radial and tangential loads or supports on curved structures by using a cylindrical rather than rectangular coordinate system.

Coordinate systems can be edited using the Properties spreadsheets or with the Model Data Explorer [see “User Coordinate Systems” in LARSA 4D User’s Manual].

The Global Coordinate System

The structure model has always one Global Coordinate System. The Global Coordinate system is a rectangular coordinate system with the axes X, Y and Z. The axes are perpendicular and right-handed. The origin and directions are chosen arbitrarily by the user, however the direction of the Z-axis is used in interpreting the orientation angle of members.

Defining Coordinate Systems

A user coordinate system definition consists of the following:

Type
The coordinate system type can be rectangular, cylindrical, spherical, or bridge path. The type of a user coordinate system affects how coordinates are entered and displayed, but not the location or orientation of the user coordinate system.

Origin (X/Y/Z)
This is the origin of the user coordinate system, in global coordinates.

For rectangular UCSs, the origin is the (0, 0, 0) point in local coordinates; for cylindrical UCSs, this is the center of the cylinder at z = 0; for spherical UCS, this is the center of the sphere; and for bridge paths, this is the location of the first point on the path.

In the spreadsheets, the following two fields are used to define UCSs:

Axis Point (X/Y/Z)
The vector from the origin to this point defines the positive x-axis of the user coordinate system. The axis point is specified in global coordinates.
Point on XY Plane (X/Y/Z)

This is a point on the positive-y side of the x-y plane of the user coordinate system, given in global coordinates. It may, of course, be a point on the UCS's +y-axis itself, but it need not be because the y-axis can be determined with any point on the +y side of the x-y plane.

The x-axis of the UCS is the vector from the Origin to the Axis Point.

The z-axis of the UCS is determined by taking the normal to the plane defined by the Origin, Axis Point, and Point on XY Plane.

The y-axis of the UCS is determined by taking the cross product of the z-axis and the x-axis.

The right-hand-rule applies.
Cylindrical coordinate systems are three-dimensional extensions to polar coordinates. The concentric circles of polar coordinates are on the x-y plane of the UCS and are extruded through the UCS's z-axis. A cylindrical coordinate has the components r, theta, and z. The z-coordinate is the same as when the coordinate is expressed in rectangular form. R and theta are computed as in polar coordinates as if all points are on the x-y plane.

R is the perpendicular distance from a point to the z-axis of the user coordinate system.

Theta is the projected angle from the x-axis to a point, generally expressed in degrees.

Z is the perpendicular distance from a point to the x-y plane.

Cylindrical coordinate systems are often used to restrain joints in radial or tangential directions, or to apply joint loads in those directions. To apply radial loads, for instance, create a cylindrical UCS and set its origin to the center of
the circle about which the loads radiate. For each joint on which a radial load will be applied, set its displacement coordinate system to the cylindrical UCS. Loads applied to these joints will act in the local directions of the UCS, which means the loads x, y, and z components correspond to r, theta, and z. Applying loads with x-components set to 5 will result in loads of magnitude 5 directed radially away from the origin of the UCS. Using the y-component instead would produce tangential loads.

Using a Cylindrical Coordinate System to Apply Radial Loads

Spherical Coordinate Systems

Spherical coordinate systems have coordinates in r, phi, theta form. R and theta are the same as in cylindrical coordinate systems, the perpendicular distance to the z-axis and angle from the x-axis, respectively.

Phi is the angle between the z-axis vector and the vector from the origin to a point. It is usually expressed in degrees.
Bridge Paths

Bridge path coordinate systems are special "traveling" coordinate systems whose axes are station, transverse offset, and elevation. The station axis follows the curved path of the bridge. It is the arc-distance along the center of the path, like the distance traveled by a car going down the center of a bridge. The station axis is shown as a thick line in the image below.

The tranverse offset axis is perpendicular to the heading of the bridge at any station. That is, it is always the perpendicular distance from a point to the center line of the bridge, in the plane of the bridge. The transverse lines in the image are lines of the transverse offset axis.

The elevation axis is perpendicular to the station and offset axes, according to the right-hand rule. This axis is shown as the single arrow going up.
Bridge paths are defined in two planes. Geometry control points at stations along the bridge with their directional headings and curve fitting options define the path in the plan or horizontal view. The elevation or vertical path is defined by a series of elevation control points at stations along the path, with the elevation and grade at each point. The vertical path is parabolically curve-fit.

For more information, see Bridge Paths (page 43).

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the UCSs spreadsheet, the commands are available in the UCSs menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the Model Data Explorer, change to the display for UCSs. Then right-click an object to get this list of commands. You can also apply the command to multiple at once by clicking on the first object, then holding the SHIFT key and clicking on the last object, and last right-clicking any of the to get the list of commands.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object.

The additional commands are as follows:

**Edit UCS...**
This command opens the UCS editing window. See User Coordinate Systems [in LARSA 4D User’s Manual].

**Make This the Current UCS**

This command sets the UCS to be the active coordinate system. See User Coordinate Systems [in LARSA 4D User’s Manual].

**Make Global System the Current UCS**

This command sets the global coordinate system to be the active coordinate system. See User Coordinate Systems [in LARSA 4D User’s Manual].

**Move UCS To Joint**

This command lets the user change the origin of a UCS by clicking on a joint in the graphics window. A window titled *Move UCS to Joint* appears. Click a joint to choose a new origin. Otherwise, click *Done* to cancel choosing a new origin.

**Translate UCS in X/Y/Z**

After activating this command, the user is prompted for the distance to move the UCS in either the x, y, or z directions in global coordinates.

**Rotate UCS in X/Y/Z**

After activating this command, the user is prompted for an angle in degrees to rotate the UCS axes about either the x, y, or z global axes.

---

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*. 
Bridge Paths

Bridge path coordinate systems are special coordinate systems whose axes are station, transverse offset, and elevation. Like the cylindrical user coordinate system (page 35), the bridge path user coordinate system is a tool of convenience for modeling. Bridge paths are particularly useful in the setup of model geometry because they allow the user to work in very simple coordinates despite any curvature of the structure. This is accomplished by warping the usual x-axis into a curve that follows the curvature of the bridge. In bridge path UCSs, this is called the station axis, and it is the thick line down the center of the bridge in the image below.

Bridge Path Coordinate Systems

The curvature of the station axis is given by defining the path in two planes. Geometry control points at stations along the bridge with their directional headings and curve fitting options define the path in the plan or horizontal view. The elevation or vertical path is defined by a series of elevation control points at stations along the path, with the elevation and grade at each point.

Bridge path coordinate systems can also be set as the displacement coordinate system of joints in order to change the direction of supports, springs, and loads to be parallel or perpendicular to the heading of the bridge at the location of the joint. This saves the effort of creating a separate rectangular user coordinate system at each pier, for instance, to set the directions of the supports at the piers.

Coordinate systems can be edited using the User Coordinate Systems (page 35) Properties spreadsheets or with the Model Data Explorer [see “User Coordinate Systems” in LARSA 4D User’s Manual].
Bridge Axes

Like the usual x-axis, the station axis is measured in length units. But the station axis is according to arc length. That is, station axis coordinates are distances that a vehicle would travel as it follows the centerline of the bridge. The benefit of this system is that any coordinate (x, 0, 0) is on the centerline of the bridge. Without a bridge path, trigonometric functions would be needed to locate points on the centerline.

The second axis of bridge paths is the tranverse offset axis, which is perpendicular to the station axis at any station, in the horizontal plane. For example, at station 1000 on the curved bridge, two parallel girders 10 units apart are passing through. The bridge path coordinates of the girders at this station are (1000, -5, 0) and (1000, 5, 0), regardless of the curvature at this point.

The elevation axis is always straight up, with the station axis at zero elevation. This is useful for setting coordinates relative to a bridge deck. Sometimes it is also useful to set coordinates relative to a ground line, especially when the bridge deck's elevation is not constant. In that case it may be helpful to create two bridge path user coordinate systems, both with the same horizontal geometry, but one with elevations set to the elevation of the deck, and the other with elevations set flat to the ground line.

Horizontal Geometry

Station and Heading

Horizontal geometry is set through geometry control points and curve fitting options. Geometry control points set the directional heading at stations along the bridge, usually before and after curved segments. For instance, if a bridge goes through an arc from station 2500, where the bridge is headed toward the east, to station 3000, where the bridge is headed at N 30° E, two geometry control points are set:

<table>
<thead>
<tr>
<th>Station</th>
<th>Heading</th>
</tr>
</thead>
<tbody>
<tr>
<td>2500</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>N 30 E</td>
</tr>
</tbody>
</table>

Headings can be entered as counter-clockwise degree measures from east, in either decimal format or DMS format. For DMS format, you may omit the symbols, e.g. enter "30 15 5" for 30° 15' 5. You may also enter surveyor notation, such as N 30 15.5 E, for 30.251 degrees from north going toward east.
Curve Types

The segments between each geometry control point are connected by a curve. The options for curve fitting are entered into the spreadsheet on the righthand side of the Horizontal Curve tab. The curve specified in row 1 of that spreadsheet corresponds to the segment between the points in rows 1 and 2 in the lefthand spreadsheet, and so on.

Three types of curve fitting are available: straight line, circle, and spiral.

Straight lines are automatically fit between two points that have the same heading.

For circular curve fitting, choose the Largest Arc option. This will choose the radius of curvature automatically based on the formula $R = \frac{S}{T}$, where $S$ is the arc-length of the segment and $T$ is the change in heading from the start point to the end point in radians.

Choose Circular Arc to specify a particular radius. If the radius is smaller than the largest radius that can fit between the points, the circular segment will be placed symmetrically between two straight line segments on either end of the segment.
Spirals

Euler spirals (also known as clothoids or Talbot transition spirals) can also be fit between control points. The spiral is a smooth transition between a circular curve and a straight line. Four spiral options are available. “Spiral Off” is used for spirals from straight lines to circular segments. “Spiral On” is used for spirals from circular segments to straight lines. The mnemonic is getting “on” the highway (i.e. from circular on-ramp to straight highway) or getting “off” the highway (from straight highway to circular off-ramp). “Right” spirals go clockwise, while “left” spirals go counter-clockwise.

For example, the Spiral Off Right option creates a clockwise spiral between a straight line and a circle.

The radius of curvature at the circular end is computed automatically from the formula \( R = \frac{S}{2T} \), where \( S \) is the arc-length if the spiral and \( T \) is the change in heading from the start point to the end point in radians.

Vertical Geometry

The vertical geometry of the station axis is determined by a series of elevation control points. Each point specifies a station, the elevation at that station, and the grade of the deck at that station. The elevation is relative to coordinate set as the origin of the user coordinate system (page 35), in the direction of the z-axis of the UCS. If the UCS origin and directions are left as the default, elevations are measured from the global xy-plane.

Vertical Curve Definition

The grade and elevation are used to fit curves between the elevation control points. Where possible, parabolic curve fitting is used. Parabolic curve fitting is only applicable when a point of intersection exists for adjacent elevation control points. PI points won't exist between elevation control points with the same grade but different elevations, for instance. In these cases, cubic curve fitting is applied.
Rotation of the superstructure can be set using the "Bank Rotation" spreadsheet. Rotations are specified at control points (stations) along the bridge path, and are given in degrees. The degree angle specifies a rotation about the bridge's station axis using the right-hand-rule (clockwise rotation).

Rotation values are linearly interpolated between the control points.

The effect of a rotation is to skew the local cross-sectional coordinate system at that point on the bridge path. The transverse offset axis is rotated according to the bank rotation angle, but the elevation axis remains vertical. This is shown in the image below, which displays the transverse offset and elevation axes at a particular cross-section on a bridge path.
Bank Rotation

Although the coordinate system is skewed for the purposes of model geometry, a non-skewed (orthogonal) coordinate system must be used when a bridge path is used as a joint displacement coordinate system or as the reference system for compound element forces results. In these cases, the local directions ignore any bank rotation. This is shown as the "displacement directions" axes in the image.
Time-Dependent Material Property Definitions

Time-dependent material property definitions are used only in the time-dependent staged construction analysis. These definitions define both concrete creep and shrinkage as well as steel relaxation behavior.

The use of time-dependent material property definitions depends on what type of material properties they represent.

**For Concrete Creep and Shrinkage**
Time-dependent material property definitions are assigned to materials ([page 19](#)) or to sections ([page 23](#)) when they are used to calculate creep and shrinkage for members ([page 57](#)).

**For Steel Relaxation**
Time-dependent material property definitions are assigned to materials ([page 19](#)) when they are used to calculate relaxation for tendons ([page 89](#)).

**Creep and Shrinkage Properties for CEBFIP-78**

When the CEBFIP-78 code is chosen for a time-dependent staged construction analysis, the following coefficients must be specified to determine creep and shrinkage behavior. The CEBFIP-78 code is used when the user wants to define custom creep and shrinkage models for each section instead of the built-in material models used with CEBFIP-90. When the CEBFIP-90 code is selected, these coefficients are not necessary, but the CEBFIP-90 models may not be applicable in extreme environmental conditions.

Creep and shrinkage are affected by time-dependent factors, such as stress, relative humidity, and ambient temperature, thus a knowledge of environmental history is required for accurate predictions of deformation. The creep coefficient also depends on:

- Dimensions of Element
- Composition of the Concrete
- Rate of Hardening of Concrete

The creep coefficient for CEBFIP-78 is defined as:

\[
\begin{align*}
\beta(t, t_0) &= b_\alpha(t_0) + j \beta_\alpha(t-t_0) + \{j_{11} j_{12}\} \beta(t) - \beta(t_0) \\
b_\alpha(t_0) &= 0.8(1 - \frac{f_{c(t_0)}}{f_{c(\infty)}})
\end{align*}
\]

For more on CEBFIP-90, please refer to CEB-FIP Model Code 90 Section 2.1.6.

See Time Effects on Materials ([page 243](#)) for more information on choosing the material model for an analysis and other time-dependent material options.

**Notional Thickness Coefficient**
A coefficient depending upon the relative humidity. Usually the coefficient is 1.0. The notional thickness is defined as:

\[
h_0 = (\text{Notional Thickness Coefficient}) \times \frac{(2 \times \text{Cross-Section Area})}{(\text{Perimeter in Contact with the atmosphere})}
\]

**Flow Coefficients**

**Flow Coefficient 1**
\( j_{f1} \). This coefficient is 0.80 for water, 1.00 for very damp atmosphere, 2.00 for outside in general, and 3.00 for a very dry environment.

**Flow Coefficient 2**

\( j_{f2} \). Depends on the notional thickness.

The flow coefficients are used in computing the irreversible delayed deformation (flow) which is very much affected by the age at which loading commences. Their definitions are in accordance with CEBFIP-78.

LARSA uses the product of these coefficients, \( j_{f1} \) and \( j_{f2} \).

Note: \( j_{f1} = 2 \) and \( j_{f2} = 1.4 \) will represent an average concrete setting and aging in average conditions with a notional thickness of 0.40m.

**Delayed Modulus of Elasticity**

\( j_d \). Used in computing the recoverable part of the delayed deformation (delayed elasticity). It is assumed to be independent of aging in its development and it is defined by a constant value.

**Shrinkage Coefficients**

**Shrinkage Coefficient 1**

\( e_{s1} \). Depends on the environment. This coefficient is +0.00010 for water, -0.00013 for very damp atmosphere, -0.00032 for outside in general, -0.00052 for very dry atmosphere.

**Shrinkage Coefficient 2**

\( e_{s2} \). Depends on the notional thickness. This coefficient is usually available in the form of a curve with horizontal axis as the notional thickness (mm) and vertical axis as the value of for the shrinkage coefficient 2.

The shrinkage coefficient definitions are in accordance with CEBFIP-78.

The strain due to shrinkage which develops in an interval of time \((t-t_0)\) is given by

\[ e_0 = e_{s1} \cdot e_{s2} \]  
(This is the basic shrinkage coefficient.)

\[ e_s(t, t_0) = e_0[b_s(t) - b_s(t_0)] \]

The basic shrinkage coefficient corresponds to \( EPS \) in the BC software program.

\( b_s \) is a user-supplied function corresponding to the change of shrinkage with time depending on the notional thickness of the section.

The recoverable creep function \( b_d \), flow function \( b_f \) and shrinkage function \( b_s \) are entered as functions of time (days). The value is computed from the given curve by interpolation. These functions are used in accordance with CEBFIP-78.

**Creep and Shrinkage Properties for CEBFIP-90**

No values need to be entered when using CEBFIP-90 for creep and shrinkage.

**Other Properties**

The following other properties can be specified:

**Creep Factor**
A factor to apply to the effect of creep for elements to which this time-dependent material property definition is relevant.

**Shrinkage Factor**
A factor to apply to the effect of shrinkages for elements to which this time-dependent material property definition is relevant.

**Relaxation Factor**
A factor to apply to the effect of relaxation for tendons to which this time-dependent material property definition is relevant.

**Material Curves**
See Time Effects on Materials (page 243) for more information on which of these curves are needed under different conditions, choosing the material model for an analysis, and other time-dependent material options.

**Concrete Shrinkage Curve (S)**
Time (in days) versus Strain, the change in shrinkage at this time. A function corresponding to the change of shrinkage with time, depending on the notional thickness \( h_0 \). This curve is only used with CEBFIP-78. This corresponds to the \( b_S \) function.

**Concrete Delayed Plastic Strain Curve (F)**
Time (in days) versus Plastic Strain. A function corresponding to the development of delayed plastic strain over time, depending on the notional thickness \( h_0 \). This curve is only used with CEBFIP-78. This corresponds to the \( b_F \) function.

**Concrete Delayed Elastic Strain Curve (D)**
Time (in days) versus Elastic Strain. A function corresponding to the development of delayed elastic strain over time, depending on the notional thickness \( h_0 \). This curve is only used with CEBFIP-78. This corresponds to the \( b_D \) function.

**Time versus Elastic Modulus Curve**
This curve is not currently used by LARSA. Instead, a built-in function is applied when called for by chosen code.

**Stress/GUTS vs. Relaxation Curve (for Tendons)**
Relaxation losses for different stress levels in the tendon. This curve is used for all creep and shrinkage codes. Stress/GUTS is the stress in the tendon divided by the guaranteed ultimate tensile strength. This value is usually less than 1.0. Higher stress values in the tendon cause higher relaxation losses. Relaxation is the total relaxation. For 6 percent relaxation, specify 0.06.

**Time vs. Relaxation Curve (for Tendons)**
Time (in hours) versus Relaxation (a proportion). A function corresponding to how relaxation varies over time. This curve is used for all creep and shrinkage codes. If a tendon experiences 100 percent loss of relaxation in 1,000 hours, then the value for Time = 1,000 should be 1.0.

**Relaxation Coefficients**
Relaxation is derived from the product of the coefficients from the Stress/GUTS vs. Relaxation and Time vs. Relaxation curves. The coefficients are the values of the curves with time being the number of days since the tendon was stressed.
Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Time Material spreadsheet, the commands are available in the Time Material menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

The additional commands are as follows:

**Edit Shrinkage Curve S**

This command opens a new spreadsheet window where the S curve can be edited. The use of this curve depends on the code being used.

**Edit Delayed Plastic Strain Curve F**

This command opens a new spreadsheet window where the F curve can be edited. The use of this curve depends on the code being used.

**Edit Delayed Elastic Strain Curve D**

This command opens a new spreadsheet window where the D curve can be edited. The use of this curve depends on the code being used.

**Edit Curve: Time VS Modulus of Elasticity**

This command opens a new spreadsheet window where the time versus modulus of elasticity curve can be edited. The use of this curve depends on the code being used.

**Edit Curve: Stress/GUTS VS Relaxation**

This command opens a new spreadsheet window where the stress/GUTS versus relaxation curve can be edited. The use of this curve depends on the code being used.

**Edit Curve: Time VS Relaxation**

This command opens a new spreadsheet window where the time versus relaxation curve can be edited. The use of this curve depends on the code being used.

For More Information, please refer to the following documentation.

• For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*. 
Geometry

In a finite element analysis, model geometry is defined by the locations of joints, support conditions at joints, constraints between joint degrees of freedom, and a set of elements interconnected at those joints. In addition to general FEM geometry elements, model geometry includes lanes, tendons, and other miscellaneous objects.

Joints

General Properties 55
Translational and Rotational Degrees of Freedom (DOF) 55
Displacement User Coordinate System 56

Members 57
Member Properties 57
Connection Beam Properties 61
Properties for Time-Dependent Material Effects 61
Member Coordinate Systems (Local Axes) 62
Self-Weight Computation 66

Spans 69

Plates 71
Usage Notes 71
Element Formulation 71
Attributes of Plates 73
Plate Local Axes 74
Material Axes 75
Self-Weight Computation 76

Springs 79
Usage Notes 79
Grounded Spring Element 79
Two-Node Spring Element 79
Inelastic (Hysteretic) Spring Element 80
General Attributes 80
Stiffness Attributes 81

Isolators 83

Mass Elements 85

DOF Constraints 87

Tendons 89
About the Tendon 89
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Friction Losses</td>
<td>89</td>
</tr>
<tr>
<td>Anchorage Slip Losses</td>
<td>90</td>
</tr>
<tr>
<td>Elastic Shortening of Concrete Losses</td>
<td>90</td>
</tr>
<tr>
<td>Other Losses</td>
<td>90</td>
</tr>
<tr>
<td>Attributes</td>
<td>90</td>
</tr>
<tr>
<td>Path Geometry</td>
<td>91</td>
</tr>
<tr>
<td>Example Path</td>
<td>96</td>
</tr>
<tr>
<td>Lanes</td>
<td>99</td>
</tr>
<tr>
<td>Basic Properties</td>
<td>99</td>
</tr>
<tr>
<td>Path Geometry</td>
<td>100</td>
</tr>
<tr>
<td>Lane Path Example</td>
<td>102</td>
</tr>
</tbody>
</table>
Joints

Joints, also known as nodes, are the connection points for stiffness-carrying elements. They are the primary locations where displacements are computed. Joints define the physical geometry of the structure and are the sites of support conditions.

Degrees of freedom are the components of displacement at a joint. Joints can have up to six degrees of freedom, corresponding to translation along the three axes and rotation about the three axes. The number of unrestrained joint degrees of freedom in a model determines the number of equations to be solved in an analysis.

General Properties

ID

Numbers are assigned to joints for identification purposes. They can be any integer, and they must be uniquely assigned among all joints in a project, but they need not be assigned consecutively.

Coordinates

Joint locations are generally specified using x-, y-, and z-coordinates in the global coordinate system. When a user coordinate system (page 35) is the active coordinate system, then the coordinates are specified with respect to that coordinate system. (See User Coordinate Systems [in LARSA 4D User’s Manual].)

Translational and Rotational Degrees of Freedom (DOF)

Each degree of freedom can be either free or fixed. A free degree of freedom indicates the joint is free to move, while a fixed degree of freedom indicates that the joint is not free to move because of the presence of a support.

Without any restraint, the structure will float in space. A free floating structure will have six rigid body degrees of freedom. The structure must be restrained against its rigid body motions. In general, the rule is that degrees of freedom without stiffness need to be restrained to prevent rigid body motions as well as ill-conditioning of the system stiffness matrix.

For example, 2D structures such as a 2D frame in the xz-plane must have y-translation and x- and z-rotations restrained for all joints. To eliminate the rigid body motion at least three additional degrees of freedom must be restrained in the xz-plane.

The restraint directions are always in the displacement coordinate system of the joint (see below). If the joint is assigned a cylindrical coordinate system as the displacement coordinate system, then X translation represents the radial and Y translation represents the tangential motion at the joint.

Universal Restraints [in LARSA 4D User’s Manual] is used to fix a set of degrees of freedom for all joints. Any degrees of freedom restrained universally will apply to all joints, regardless of the DOF selected for a joint.

If the displacement of a joint along any one of its degrees of freedom is known ahead of time, then the degree of freedom is restrained. It should be set to fixed, and a joint displacement load (page 115) should be applied.

Joints can be constrained using DOF Constraints (page 87). Constrained degrees of freedom on dependent joints are removed from the system of equations to be solved.

To model other support conditions, such as soil-structure interaction, see Springs (page 79).
Displacement User Coordinate System

The directions of motion (displacements) of a joint are specified and computed in a coordinate system identified as the joint displacement coordinate system. The default system is the Global Coordinate System. Any user coordinate system (including a Bridge Path Coordinate System) can be assigned to a joint as the displacement coordinate system.

Supports, spring elements, DOF constraints, joint loads, and support displacements all act in the directions specified by the joint displacement coordinate system.

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Joints spreadsheet, the commands are available in the Joints menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object only.

To apply the command to multiple joints at once, first select those joints and unselect everything else, then go to the Modify menu, find the Joints menu, and choose a command from within that menu. Activating these commands from the Modify menu will apply the action to all selected joints.

The additional commands are as follows:

**Renumbering Setup**

This command sets up renumbering options for joints. The user can choose the starting ID and the step.

**Renumber**

Renumbers the selected joints according to the renumbering setup options, which must be set first.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- User Coordinate Systems on page 35.
Members

Members are structural elements connecting two joints and are used to model beams, columns, cables, and trusses. The members can also be used to model yielding connections and plastic hinging within the element.

Member Properties

ID
Member numbers are assigned to beams for identification purposes. They can be any integer, and they must be uniquely assigned among all members in a project, but they need not be assigned consecutively.

Start (I) and End (J) Joints
A member's connectivity is described using two non-coincident joints. Members may not have zero length. The joints' IDs are used here.

Member Type
The member types that can be selected are Beam, Truss, Cable, Compression-Only Truss, Tension-Only Truss, and Hysteretic Beam. The types Inactive and Construction Line are available to have the analysis engine ignore the stiffness of the member in the analysis. There is no difference between Inactive and Construction Line, but the user may decide to make use of the distinction.

Cable, Compression-Only Truss, Tension-Only Truss, and Hysteretic Beam elements and a beam with member-end nonlinear springs are nonlinear elements when used in a nonlinear analysis. If cable or truss elements are used in a linear analysis, they are treated as simple truss elements. The hysteretic beam elements fall back to the standard beam element in a linear analysis.

Beam Element
The beam element has six degrees of freedom at each end joint: translational displacements in X, Y and Z directions and rotational displacements about X, Y and Z directions. This element includes axial and shear deformations, twisting about its x-axis, bending in two perpendicular planes, and associated shears. The beam element is capable of exactly representing constant axial deformation along the beam with constant torsional shear deformation and linear bending deformations within the element. This is sufficient for analyzing structures with loads applied at joint points. However, modeling these deformations requires a higher order representation due to axial loads, torsional loads, lateral loads, and moments along the elements. Use of the fixed end forces allows these higher-order deformations without the need of additional degrees of freedom for the beam elements.

This element has geometric nonlinearity and stress-stiffening or softening in all nonlinear analysis types. It also includes beam-column properties.

When member-end nonlinear springs are assigned, the beam becomes the Connection Beam Element (see below).

Truss Element
The truss element has only translational displacements in X, Y and Z directions at each end and it has no rotational stiffness. These elements can carry axial force only. The cross-sectional area determines the axial stiffness of the element, and it is the only stiffness required to be entered for sections assigned to truss element members. The values for Ix, Iy, Iz and shear areas are ignored for truss and cable elements.
This element has geometric nonlinearity and stress-stiffening or softening in all nonlinear analysis types.

**Cable Element**

The cable element is identical to the truss element except a cable element cannot resist compressive force. The cable element can only be used in a nonlinear analysis. If during the analysis, the axial force in a cable element becomes compressive, then the cable element is assumed to have no axial stiffness and cannot carry any load. It is kept in the model as an inactive element with no contributing stiffness. During the loading process of a nonlinear analysis, if the element can become tensile again, it is included in the model with contributing stiffness to the system.

This element has geometric nonlinearity and stress-stiffening in all nonlinear analysis types.

**Compression- and Tension-Only Truss Elements**

These elements are identical to the truss element except the compression-only element cannot resist tension forces and the tension-only element cannot resist compressive forces.

This element has geometric nonlinearity and stress-stiffening or softening in all nonlinear analysis types. This element cannot be used in any linear analysis.

**Inelastic Hysteretic Beam Element**

This advanced element is used in nonlinear analyses for plastic pushover and progressive collapse. The formulation of the element is based on spread plasticity to capture the variation of section flexibility and combine them to determine the element stiffness matrix. It has nonlinear material properties including hysteretic behavior and geometric nonlinearity for tension stiffening.

Instead of using the material stress-strain relations directly, the hysteretic beam element uses the section axial force-centroidal strain and moment-curvature relationships to reduce computation. Hence the term strain in the context of the hysteretic beam element refers to the vector consisting of centroidal axial strain, curvature about the element y-axis, and the curvature about the element z-axis.

The interaction between the axial force and the two bending moments at the sections located at the element Gauss points is taken into account using a yield surface and the rules of plasticity: (i) Normal Flow rule and (ii) Kinematic Hardening rule. Normal flow rule states that plastic strain increments are orthogonal to the yield surface. Shear deformations and material nonlinearity in torsion are ignored.

Conventional finite elements use the stiffness formulation where the nodal displacements are interpolated within the element using displacement interpolation functions. In contrast, the hysteretic beam element uses force interpolation functions, i.e., the element end forces are interpolated within the element. This is because, for beam-elements, while displacement interpolation functions are accurate only for prismatic members with linear material behavior, the force interpolation functions are always accurate.

This necessitates an element level state determination procedure to determine the element end forces and strains at the Gauss Points. The Newton-Raphson method is used within the element for this purpose. The number of equations for each hysteretic beam element depends on the number of Gauss points. This number is set to 10 and cannot be modified by the user.

For the consideration of geometric nonlinearity, the element deformation is split into two parts: (i) the rigid body rotation of the element coordinate system and (ii) the deformation of the element within this coordinate system. Only the former is included in geometric nonlinearity.
The ignoring of the deformations within the element coordinate system may lead to greater inaccuracies when the element is in single curvature and when there is significant deformation before yielding in double curvature. In such cases, to increase accuracy the element should be subdivided.

Member loads applied to hysteretic beam elements are non-consistent. These loads are transferred immediately to the I- and J-joints at the ends of the member. Mid-span point loads, for instance, will be treated instead as split between the end joints.

**Connection Beam Element**

The connection beam element consists of an elastic beam with built-in yielding springs at the ends. There can be up to four such end-springs, two springs at each end, one for each direction of bending. The properties of these springs can be assigned so as to model the elastic-plastic behavior of the beam itself and/or to model the flexibility and inelastic behavior of the end connections.

The spring properties are assigned using spring curves (page 29) representing the moment-rotation behavior and by selecting the appropriate parameters that determine hysteretic behavior.

This element is used by specifying member end nonlinear springs for members whose type is `beam`.

**Super Cable Element**

The Super Cable Element has multiple internal Gaussian points that simplifies cable modeling. Self-weight and other types of distributed and concentrated loads imposed within the element are analyzed as equivalent concentrated loads at internal Gaussian points. While simplifying the modeling of cables, the element provides a realistic and reliable solution considering geometric nonlinearity due to the sagging effect under arbitrary loading.

**Section: At Start, End**

The sectional properties of members are established by assigning a section type (page 23) from the list of sections defined in the project. The section's name is used here.

A member is assigned one or two sections (page 23), which give the cross-sectional properties of the member. If one section is assigned, the section applies throughout the member. If two sections are assigned, the sectional properties of the member vary linearly, with the start of the member taking sectional properties from the first assigned section, and the end of the member taking sectional properties from the end of the section.

Sections may have nonprismatic properties, in which the dimensions of the section change over the length of the member, either through assigning two section properties as above, or using a nonprismatic section defined in the Section Composer. When a Section Composer nonprismatic section is assigned to a member, a second section cannot also be assigned. When a nonprismatic section from the Section Composer is assigned, use Spans (page 69) to extend the nonprismatic variation over multiple connected members.

The beam element can carry axial forces, shear forces, bending moments and torsion. The data for cross-sectional area gives axial stiffness in the x-direction; the torsional moment of inertia gives stiffness for rotation about the element x-axis; the moment of inertia about the y-axis gives stiffness for translation in the z-direction and for rotation about the y-axis. The moment of inertia about the z-axis gives stiffness for translation in the y-direction and for rotation about the z-axis. The shear areas are for shear deformations. If you set any of the cross-sectional variables are set to zero on the section property data, the corresponding stiffness is zero and the corresponding force at both ends is released.

The orientation of the principal axis of a member is defined with an orientation angle (see below), which sets the relative orientation of the member's local axes. The relation of the principal axis to the local axes of a member is dependent on the definition of the member's sectional properties.
Material
The material properties of members are established by assigning a material type (page 19). The material's name is used here. Material properties are constant throughout the member, except when a composite section defined in the Section Composer has been assigned as the section of the member.

Prestress Force
The initial prestressing force can be set for beam, cable, and truss elements. This type of prestressing can only be used in a nonlinear analysis. For prestressing with tendons, see Tendons (page 89). Member loads (page 117) can also be used.

Rigid Zones
The structural model is normally idealized representing the center line of the beams, columns, and braces. This technique of center-line modeling ignores the thickness of elements and results in greater lengths than is actually the case because a portion of element thickness is included in the idealized length of other elements. For instance, the thickness of a column is not accounted for in the lengths of beams attached to the column.

When the joint thickness is small compared to the actual member length, errors due to ignoring the thickness are insignificant. If the joint thickness is large compared to the actual member length, the error may be significant.

Rigid end zones can be specified to include the actual characteristics of the element end condition. A rigid end is specified as the length along the member from the start or end joint, or both, which is to be considered infinitely stiff, specified normally as a relative length from zero (i.e. no rigid end) to 0.5 (i.e. half the length of the member is rigid). It is also possible to specify rigid zones in absolute length units. See the Additional Commands section below.

Use of rigid end zones produces more realistic displacements and element forces. The element loads are considered to be acting only on the elastic length of the beam. The moment and shear force distribution along the member is computed accordingly. Loads placed in rigid zones are ignored.

While the total member length for the purposes of self-weight computation is from joint to joint, the member length used in stiffness computation is the length of the elastic portion of the member (i.e. excluding the rigid zones). To shorten the length of the member for the purposes of automatic self-weight computation, use End Offsets.

Orientation Angle
Members can be situated in any angle about their centroidal axis, such as placing flanges horizontal or vertical. The orientation angle of a member determines this angle. It is crucially important to define the member orientation properly so that the strong axis of the member is in the right direction.

The orientation angle is used to determine the orientation of the local coordinate system of the member, which is explained below. Roughly speaking, the orientation angle sets the angle between the global Z-axis and the member's local z-axis. Member sectional properties are given in member local directions. A member whose section's Izz is greater than its Iyy, which is the case for standard I-shape sections, has its strong axis in its local z-axis direction. If the global Z-axis is "up" and the member is a horizontal beam, the the strong axis needs to be made horizontal by specifying an orientation angle of 90 degrees. For vertical members, the orientation angle specifies the angle between the global X-axis and the local z-axis.

See the section on member coordinate systems below for a detailed explanation of member local axes and the orientation angle.

End Offsets
The centerline of a member is normally the straight line between the member's two joints. End offsets can be applied to move the end-points of the member, relative to the locations of the joints. Offsets can be applied at the start and end of a member and are given in global coordinate system directions.

The actual length of the member is computed after the end offsets are applied. This length is used for stiffness computation and load application (including self-weight).

An end offset is equivalent to a rigid link between the offsetted end of the member and the joint it is connected to at that end.

See the section on member coordinate systems below.

End Releases

The end connections of beam elements are by default perfectly rigid. Beam-joint connections can be also specified as pinned or semi-rigid. If a member end release is specified, then the corresponding force is set to zero. The effect of the hinge is to free the force at the end of the beam, not at the joint.

The release components are in the member local coordinate system. All directions except Fy and Fz may be released. Semi-rigid releases can be applied only for Mx and Mz. The semi-rigid end connections are specified as a percent of fixity. A value of 0 represents a perfect hinge (free) and 100 represents full fixity. Semi-rigid connections are ignored when applied to hysteretic and connection beams. Fy and Fz can be effectively released by making use of DOF constraints (page 87).

Note that a rigid/fixed member-joint end connection does not prevent the joint itself from translating or rotating, subject to its own degrees of freedom restraint conditions. Instead, a fixed member-joint connection ensures that all other member rigidly connected to the same joint undergo the same displacement at the joint.

Structure/Construction Group

Structure groups can be assigned directly to members. It is possible to select an existing structure group here or specify a new one. Structure groups are used to modularize the geometry of the model and plays a key role in staged construction analysis. For more information see Structure Groups Explorer [see “Structure Groups Explorer” in LARSA 4D User’s Manual].

Connection Beam Properties

Member End Nonlinear Springs

Application of these springs makes the member a connection beam element. The connection beam element consists of an elastic beam with built-in yielding springs at the ends. There can be up to two springs at each end, one for each direction of bending. The properties of these springs can be assigned so as to model the elastic/plastic behavior of the beam itself and/or to model the inelastic behavior of the end connections. The type of connection beam members must be beam.

Properties for Time-Dependent Material Effects

In the Time-Dependent Staged Construction Analysis (page 229), members can be subject to time-dependent material effects including creep, shrinkage, and the time-effect on elastic modulus. The following fields are relevant to this analysis:

Casting Day
The day the member is cast, for use in the Time-Dependent Staged Construction Analysis (page 229). The age of the member when constructed is determined by subtracting the casting day from the day of construction set for the construction stage being analyzed. The casting day of a member must be less than the day of construction when the member is assembled.

Starting in LARSA 4D version 7.2, there is also a cast concrete activity (page 147) in Staged Construction Analysis which can be used to set the casting day of the member. It is also possible to cast individual shapes within a built-up section created in the Section Composer. In this case, the cast concrete staged construction activity must be used rather than the Casting Day attribute of the member element as a whole.

Additionally, a time-dependent material property definition (page 49) must be assigned to either the material or the section that is assigned to the member, but not both. The time-dependent properties are thus indirectly associated with the member element.

### Member Coordinate Systems (Local Axes)

Members have two "local" coordinate systems for the purposes of the alignment of cross-section geometry and properties, loading, the alignment of tendons and lanes, and reporting forces, stresses, and other results. The local coordinate systems of members determine both the location of the member centroid and the orientation of a member about its centroid: which way the wedges and flanges are pointing, for instance. Because these coordinate systems define the position of the centroid and the orientation of the member's strong axis, it is crucially important to understand and set up these systems properly.

The two systems are the **reference coordinate system** and the **local coordinate system**. In most cases, the two systems are the same. They differ only when a member has been assigned a section from the Section Composer [see “LARSA Section Composer” in *LARSA Section Composer Manual*] with a centroid offset from the reference coordinate system.

The **reference coordinate system** is used to determine the placement of the member's cross-sectional geometry and tendons (page 89). Its origin is located at the I-Joint of the member, plus any end offset specified at the I-Joint. The x-axis of the reference coordinate system is directed toward the J-Joint of the member, plus any end offset specified at the J-Joint.
The y- and z-axes of the reference coordinate system are oriented according to the orientation angle of the member, as follows:

The direction of the reference coordinate system z-axis is determined as follows:

Call the global coordinate axes X, Y, Z and the reference member axes x, y, z. The orientation angle is defined as follows:

- Let A be a plane containing the reference x-axis and a line parallel to the global Z-axis. Plane A will therefore be perpendicular to the XY plane.
- Let z' be a point on a line connected to the origin of the reference coordinate system in the plane A and perpendicular to the reference x-axis. The location of z' must be taken so that the projection of the line from reference origin to z' is in the positive Z direction.
- Then, the orientation angle is defined as the angle from z' to z, positive by the right-hand rule about the member local x-axis.

In other words, the orientation angle is the angle between the global Z-axis and the reference z-axis

This definition is valid for all members except when the reference x-axis is parallel to the global Z-axis. In such a case, the plane A cannot be defined as described above. If the reference x-axis is parallel to the global Z-axis, the following rules are used:

- When the reference x-axis is in the same direction as the global Z-axis, the orientation angle is defined as the angle from the minus X-axis to the z-axis.
- When the reference x-axis is in the direction opposite to the global Z-axis, the orientation angle is defined as the angle from positive X-axis to the z-axis.
The member local coordinate system x-axis corresponds to the member centroid, with the y- and z-axes the axes about which the sectional properties (page 23) Iyy and Izz are given and the directions in which member loads (page 117) and results such as Member End Forces (page 269) are entered and reported in. For standard sections (i.e. those not created in the Section Composer), the local x-axis (i.e. the centroid of the member) is placed along the x-axis of the reference coordinate system, and the member local coordinate system coincides with the reference coordinate system.

Note that the local member axes are not necessarily principal axes. If the product of inertia Iyz is zero for the member, the local axes coincide with the principal axes of the member.

For sections created in the Section Composer [see “LARSA Section Composer” in LARSA Section Composer Manual] when the centroid of the section is not placed at the member reference coordinate system origin, the centroid is offset from the reference axis according to the section definition. For sections without nonprismatic variation, this merely offsets the member local coordinate system from the reference axes. The local coordinate system y- and z-axes are directed the same way as (are parallel to) the reference axes.
If the section has nonprismatic variation, the end points of the centroidal local x-axis are positioned in the reference coordinate system according to the section definition. The local coordinate system y- and z-axes are oriented so that the local z-axis is parallel to the reference z-axis. In this case, the local y-axis will not be parallel with the reference y-axis.

When a user coordinate system is active, the orientation angle is displayed relative to the active local coordinate system's axes rather than the global axes. For members parallel to the user coordinate system's z-axis, the orientation angle is the angle from the coordinate system's positive x-axis. This differs from the convention described above which uses the negative global x-axis.
Self-Weight Computation

The self-weight of members used in static load cases with self-weight (page 111) activated, is applied as a uniform load with magnitude equal to the member's material unit weight multiplied by the member's cross-sectional area. It is applied in the direction specified in the load case.

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Members spreadsheet, the commands are available in the Members menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object only.

To apply the command to multiple members at once, first select those members and unselect everything else, then go to the Modify menu, find the Members menu, and choose a command from within that menu. Activating these commands from the Modify menu will apply the action to all selected members.

The additional commands are as follows:

**Edit Section**

When the member is assigned to a standard section, this command opens the sections spreadsheet to the member's section. When the member is assigned a nonprismatic section, this command opens a new window to edit the parameters of the section and to view its nonprismatic variation.

**See Section Stresses**

When analysis results are available and there is an active result case, this command opens a new window showing the stresses throughout the member's cross-section, at any point along the length of the member.

**Rotate Principal Axis**

The member's orientation angle is increased by 90 degrees.

**Enter Relative Positions**

Choosing this option sets the rigid zone fields to be entered in relative dimensions, where 0 is no rigid zone and 1 is the entire length of the member.

**Enter Absolute Positions**

Choosing this option sets the rigid zone fields to be entered in true length dimensions.

**Break Member to Existing Joints**
If the member passes through joints but is not connected to the joints, the member is broken into pieces so that it connects to each joint it passes through.

**Reverse Direction**

The member's start (I) and end (J) joints are reversed, which reverses the direction of the local x-axis of the member.

**Select Chain**

This command starts with the chosen member, or the selected members, and selects all members on the same chain. A chain is a series of members connected by joints all going in the same direction, like a physical member or girder modeled as many members.

**Orient Principal Axis to Joint...**

This command prompts the user to choose a joint, and then orients the member's local z-axis so that it points toward the chosen joint.

**Renumbering Setup**

This command sets up renumbering options for members. The user can choose the starting ID and the step.

**Renumber**

Renumbers the selected members according to the renumbering setup options, which must be set first.

**Renumber Chain**

This command renumbers the members in the chosen member's chain according to the renumbering options. If the member is selected, only selected members on the chain will be renumbered. Otherwise, all members on the chain will be renumbered.

**Span Properties**

If the member is part of a span (page 69), the span's properties are shown.

---

For More Information, please refer to the following documentation.

- Spans on page 69.
- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User's Manual*.
- Joints on page 55.
- Materials on page 19.
- Sections on page 23.
- Member Reference Line and Local Coordinate System in *LARSA Section Composer Manual*. 
Spans

Spans are special entities that represent physical members in the real world. Spans are collections of one or more members that usually form a straight line. They must be used in certain cases of sections with nonprismatic variation.

A beam in the real world must often be modeled as multiple members in LARSA so that connections along the member can be made. Such members are called analytical members, as opposed to physical members, because they only exist for the sake of the analysis.

It is sometimes necessary to treat groups of members that represent one physical member in a special way, such as for code checks and design [see “Steel Design” in LARSA 4D User’s Manual], for obtaining analysis results (page 281), or when the physical member's cross-section has nonprismatic variation [see “Nonprismatic Variation” in LARSA Section Composer Manual].

All members in a span must have the same section and material properties and must be of the same type. For more information on span properties, see member properties (page 57).

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Spans spreadsheet, the commands are available in the Spans menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

The additional commands are as follows:

**Properties**

This command opens a window which allows the user to add and remove members from the span, and to change the span's material and section properties.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
- Members on page 57.
- Nonprismatic Variation in LARSA Section Composer Manual.
Plates

The plate is a planar element with constant thickness, either triangular or quadrilateral in shape, with isotropic or orthotropic material properties. It has both plate bending (out-of-plane action) and plane stress (membrane actions, in-plane action) stiffness.

Usage Notes

The plate element is a linear element with linear material properties (page 19). Geometric nonlinearity is only introduced through the updating of the deformed shape that occurs in the nonlinear static analysis (but not in Staged Construction Analysis).

The use of the triangular element should be limited to transition zones where quadrilateral elements cannot be easily used. Triangular elements with interior angles close to 60 degrees each are ideal, and interior angles in excess of 120 degrees will distort results around the element.

The aspect ratio for quadrilateral plates, defined as width-to-length, should not be so large as to skew the element. The interior angles for quadrilateral elements should not be much greater than 90 degrees.

Plates must have non-zero area and thickness. When the thickness is large in comparison to the width of the plate, a thick-plate element behavior must be chosen.

Element Formulation

Several bending (out-of-plane, also called plate) and membrane (in-plane) behaviors are available for the plate element. The correct choice of bending and membrane behavior will depend on the model according to factors explained below.

Bending (Out-of-Plane) Behaviors

**Inactive**

The element has no plate stiffness.

**Thin Plate**

The thin plate (DKQ-DKT) bending behavior has two rotational ($\theta_x$ and $\theta_y$) and one translational ($w$) degrees of freedom at each node. Shear deformations within the thickness of the element are assumed to be equal to zero. Factors to consider:

- Can be used to model any type of plates where shear deformations are not important such as slabs, decks, retaining walls, abutments, etc.
- Shear forces are only computed at the center of the element using the first derivatives of the internal element moments.
- Gives more accurate internal moments when compared with thick plate elements.

**Thick Plate**

The thick plate bending behavior has two rotational ($\theta_x$ and $\theta_y$) and one translational ($w$) degrees of freedom at each node. Shear deformations within the thickness of the element are assumed to have a linear variation.
The element has quadratic displacement field definition for bending and shear deformations. Factors to consider:

- Can be used to model plates where shear deformations are important such as pile caps and mat foundations.
- Does not give correct moment results just under point loads. The moments elsewhere are accurate. Therefore, the thick plate bending behavior should be avoided in coarse meshes.
- Shear forces are only computed at the center of the element using the shear strains.

In order to determine whether shear deformations are important or not, the parasitic shear parameter can be computed with the following equation for the direction in interest: \( \Phi = \frac{12EI}{A_sGL^2} \), where \( E \) and \( G \) are the elastic and shear modulus, respectively, \( I \) is the moment of inertia, \( A_s \) is the equivalent shear area, and \( L \) is the clear span length. Having a value equal to 0.05 means that shear deformations make the plate approximately 5% flexible. Thus, based on the value of parasitic shear parameter, one can decide whether to consider shear deformations or not.

Membrane (In-Plane) Behaviors

The following in-plane membrane behaviors are available:

**Inactive**

The element has no membrane stiffness.

**Bilinear (Quadrilateral Plates Only)**

The bilinear membrane behavior for quadrilateral elements has two translational (ux, uy) and one rotational (θz) degrees of freedom at each node. The rotational degree of freedom has a fictitious stiffness to provide stability in shell structures. The element has a bilinear displacement field definition. Factors to consider:

- The element has a linear displacement field and so suffers from shear locking (very stiff behavior) in bending problems.
- Can be used for cases where axial deformations are more important and bending deformations are negligible.
- Element stresses are less sensitive to mesh distortions when compared with higher order elements.
- The effect of rotation at the nodes (θz) cannot be computed accurately, and this may be important when the nodes connect to other elements such as beams with significant end rotations.

When the Bilinear / CST option is chosen for triangular elements, the Constant Strain Triangle (CST) behavior is used instead.

**Incompatible (Quadrilateral Plates Only)**

The membrane with incompatible modes behavior has two translational (ux, uy) and one rotational (θz) degrees of freedom at each node. The rotational degree of freedom has a fictitious stiffness to provide stability in shell structures. The element has additional internal general displacement parameters named as incompatible modes. With these additional parameters, the element has a higher order displacement definition which avoids shear locking (i.e. better bending behavior when compared with bilinear elements). Factors to consider:

- Can be used for modeling flat members.
- Due to the definition of general displacement terms within the elements, the displacement at the element edges are not continuous (compatible) between elements.
- Element stresses are more sensitive to mesh distortions when compared with bilinear membrane element.
• The effect of rotation at the nodes (θz) cannot be computed accurately, and this may be important when the
nodes connect to other elements such as beams with significant end rotations.

When the Incompatible / CST option is chosen for triangular elements, the Constant Strain Triangle (CST)
behavior is used instead.

Drilling (Quadrilateral Plates Only)

The membrane with a drilling degree of freedom behavior has two translational (ux, uy) and one rotational
(θz) degrees of freedom at each node. The rotational degree of freedom is defined from the displacement
of the mid-points of the edges, thus they have actual stiffness values for drilling degrees of freedom. The
element has a quadratic displacement field definition. Factors to consider:
• Can be used for modeling shell structures such as folded plates, curved roofs, etc.
• When connected to beam elements at a node, the effect of rotation (θz) is better represented.
• Defining a restraint for the drilling degree of freedom means that the mid displacements of the neighbor
edges in the outward normal direction is also restrained.

When the Drilling / CST option is chosen for triangular elements, the Constant Strain Triangle (CST) behavior
is used instead.

Constant Strain Triangle (CST)

The CST membrane behavior is used for triangular plate elements when Bilinear, Incompatible, or Drilling
are selected. This behavior has two translational (ux, uy) and one rotational (θz) degrees of freedom at each
node. The rotational degree of freedom has a fictitious stiffness to provide stability in shell structures. The
element assumes constant axial strain between the nodes. Factors to consider:
• Not a good element for bending.
• Should be used with a fine mesh size for acceptable results.
• The effect of rotation at the nodes (θz) cannot be computed accurately, and this may be important when the
nodes connect to other elements such as beams with significant end rotations.

In addition, there is a bending and membrane behavior called Classic which was the plate element formulation available
in LARSA 4D through version 7.1.12. It is no longer recommended to use these behaviors because time-dependent
material effects such as creep and shrinkage and orthotropic material properties are not supported for plates with
these behaviors. They can only be used when both the bending and membrane behaviors are set to Classic. These
behaviors were recommended for thin plates when shear deformations are negligible. For the bending behavior, a
cubic polynomial is used to describe the displacement of the mid-plane. The membrane behavior is similar to the
Incompatible membrane behavior described above: It has no stiffness for rotation about the axis perpendicular to the
element surface, and it is an incompatible element (bilinear shape functions with incompatible modes). For triangular
elements, a constant strain formulation is used.

Attributes of Plates

ID
Plate numbers are assigned for identification purposes. They can be any integer, and they must be uniquely
assigned among all plates in a project, but they need not be assigned consecutively.

Bending Type
The out-of-plane plate behavior, as described above.
**Membrane Type**

The in-plane membrane behavior, as described above.

**Joints (I, J, K & L)**

The element connectivity of the plate elements is described using 3 joints for triangular elements and 4 joints for quadrilateral elements. These joints must not be coincident or collinear and must be specified in clockwise or counter-clockwise order. The order will determine the element's local coordinate system. For triangular plates, Joint L should be set to *none*.

**Material**

The material properties of the element are established by assigning the material type to the element. Both isotropic and orthotropic materials can be used (see Materials [page 19]).

**Thickness**

The thickness of the element. The value entered must be in the units specified for section properties. The thickness is constant throughout the plate.

**Structure/Construction Group**

Structure groups can be assigned directly to plates. User have option to select an existing structure group or specify a new one. Structure groups are used to modularize the geometry of the model and plays a key role in staged construction analysis. For more information see Structure Groups Explorer [see “Structure Groups Explorer” in LARSA 4D User’s Manual].

**Material Angle**

For plates with orthotropic material properties, this field reports the angle between the local x-axis of the plate and its Material 1 axis. It cannot be set by the user. Instead, set the Material UCS property [page 19] of the material.

**Plate Local Axes**

Plate loads, plate results, and plate material properties are defined with respect to up to two different element-specific coordinate systems. The first is called the plate local axes. Plate local axes define the directions of plate loads and the directions that internal plate forces and stresses are reported in.

The local x-axis and local y-axis are on the surface of the plate. The plane defined by the local x- and y-axes represent the in-plane directions. The local-z direction is the out-of-plane direction of the element and is perpendicular to the surface of the plate. The diagram below shows the definition of the plate local coordinate system:
Plate Local Axes (left) and Material Axes (right)

The local x-axis is defined as the vector from the location of the plate element’s first joint (plus any plate end offsets at that joint) to the location of the plate element’s second joint (plus any plate end offsets at that joint). The positive local-z axis points to the side of the plane from which the joints appear in counter-clockwise order. The direction of the local y-axis is determined from right hand rule using the local x- and z-axes.

It is important to note that the element’s local axis directions depend on the way the element connectivity is defined. The local axis is always determined by using the global coordinates of the first three nodes, in order, and the plate element’s corresponding end offsets, if any. Thus, depending on the element geometry and element connectivity definition, the local axes of these elements in a mesh may be different.

Formally, the local axes of plates are determined using the sequence of the joints entered (I, J, K, and, for quadrilateral plates, L, defined in counter-clockwise order):

- The origin of the plate's coordinate system is at the I joint.
- The x-axis is along the line from the I joint to the J joint.
- The z-axis is perpendicular to the plane of the plate, in the direction of the right-hand rule.
- The y-axis is normal to x- and z-axes.
- The thickness of the plate extends \( \text{thickness}/2 \) into the positive-z direction and \( \text{thickness}/2 \) into the negative-z direction.

**Material Axes**

Material axes are used when an orthotropic material is applied to the plate element. (See Materials (page 19).) Orthotropic material properties are applied to the plate element according to its material axis directions, not the element’s local axis directions.

The material axes are named 1, 2 and 3, so as not to be confused with the local axes x, y, and z. While Material 3 axis is always identical to the out-of-plane local z-axis, the Material 1 and 2 axes may not be identical to the local y- and z-axes. These material axes are determined according to the material’s Material UCS, which is either either the Global Coordinate System or a user coordinate system. The Material 1 axis is the projection of the Material UCS’s x-axis onto the surface of the plate element, and the Material 2 axis is the projection of the Material UCS’s y-axis onto the surface of the plate. The Material UCS determines the material axes for all plate elements that the material is applied
to. Therefore, to set a plate element’s material axes, set the Material UCS on the plate’s material to a User Coordinate System whose x-axis is in the direction of the desired Material 1 axis.

The “material angle” is the angle between the element local x-axis and the Material 1 axis measured in the counterclockwise direction. The material angle is not entered by the user. It is computed automatically and reported in the Plates spreadsheet.

See the plate axes figure above.

Self-Weight Computation

Automatic self-weight computation (page 111) uses the plate's material unit weight (page 19) and the plate's thickness to determine the magnitude of the uniform distributed load placed in the given direction of self-weight.

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Plates spreadsheet, the commands are available in the Plates menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object only.

To apply the command to multiple plates at once, first select those plates and unselect everything else, then go to the Modify menu, find the Plates menu, and choose a command from within that menu. Activating these commands from the Modify menu will apply the action to all selected plates.

The additional commands are as follows:

**Renumbering Setup**

This command sets up renumbering options for plates. The user can choose the starting ID and the step.

**Renumber**

Renumbers the selected plates according to the renumbering setup options, which must be set first.

**Reverse Joint Order**

The order of the I, J, K, and L joints are reversed. The effect is to keep the origin of the plate's local axes at the I joint, but to direct the local x-axis not toward the second joint but instead toward the last joint, and to reverse the direction of the local z-axis.

**Shift Joint Order**

The order of the I, J, K, and L joints are shifted so that the origin of the plate's local axes moves to the J-joint.

**Put Local Axes in Same Direction**
When many plates are selected, this command ensures that all of the plates' local z-axes are directed in the same direction, i.e. all up or all down.

**Break Plates to Existing Joints**

The plate is meshed so that any joints within the plate are connected to the meshed surface.

For More Information, please refer to the following documentation:

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- Materials on page 19.
- Joints on page 55.
Springs

The spring is a very general element that can be used to model foundations, soil-structure interaction, gaps, bearings, and isolators, and it can be used as a general stiffness element. The spring element can be grounded (one-node) or two-node and can have linear, nonlinear, elastic, and inelastic material properties. All springs have one degree of freedom at the joints they are connected to, except when a 6x6 matrix is assigned to the spring.

Usage Notes

For coupled springs, see Isolators (page 83).

The spring element is weightless. Joint loads (page 113) for static analysis or mass elements (page 85) for dynamic analysis can be used to model the weight or mass of springs, if necessary.

Grounded Spring Element

The grounded spring element (i.e. the one-node spring) is defined using a single joint and there is no length associated with the element. The element represents a spring connecting a joint in the structure to the ground at its undeformed location.

The linear grounded spring element has the same stiffness in both tension and compression. Either a single spring constant is specified, or a 6x6 stiffness matrix (page 29) is applied to the spring. If a spring constant is given, the element can act in one direction only: either translation along or rotation about an axis of the global coordinate system, or the joint displacement directions (page 55) assigned to the joint the spring is connected to if a joint displacement system has been assigned to the joint.

The nonlinear grounded spring element is a general foundation element. This element is a versatile element for use in mat foundations, beams on elastic foundations and in structures supported by contact only. It is used to model foundations where soil has stiffness in compression and no resistance in tension or to model piles with varying resistance in tension versus compression. For nonlinear grounded springs, k-tension, k-compression, maximum tension, maximum compression, hook, and gap values can be specified. Alternatively, a spring curve (page 29) can be assigned describing the force-deformation curve of the element. The spring acts in a single direction only.

Two-Node Spring Element

This element is defined using two joints. The element represents a spring connecting the joints, or represents arbitrary stiffness between the two joints in the direction specified. The length of the element may be zero, depending on the spring's direction property.

This element can act in one of several directions:

1) It can act in either the axial or torsional direction along or about the straight line connecting the spring's ends, provided the length of the element is non-zero. An axial spring acts according to the lengthening or shortening of the straight-line distance between the joints. For torsional springs, "tension" is rotation about the joint-to-joint line following the right-hand rule by the end joint, relative to the start joint.
2) It can also act in a direction given by either the global coordinate system or a user coordinate system. In these cases, the length of the element may be zero. The directions are along or about the start-joint's displacement coordinate system (The end-joint's displacement coordinate system is ignored for determining the direction of the spring). The start joint's displacement coordinate system can be a user coordinate system (page 35) or the global coordinate system if it has not been assigned a joint displacement coordinate system (page 55). The spring deformations (page 299) are reported as the displacement of end joint relative to the start joint and the spring forces (page 297) are reported as the forces at the start of the spring element, in the direction chosen. In other words, the positive displacement or force does not necessarily correspond to extension or tension.

The linear two-node spring element has the same stiffness in both tension and compression and acts in a single direction only. Only one spring constant can be specified (see Springs (page 79)). The 6x6 stiffness matrix option for grounded springs is not available for two-node springs.

The nonlinear two-node spring is defined using a spring curve (page 29) describing the force-deformation curve of the spring. The nonlinear elastic spring element loads and unloads along the same path. Its stiffness can only be specified in this manner.

Gap and hook distances, yield force, and different behavior for compression and tension can be included in the spring material curve with the proper definition of the tabular function representing the behavior.

The two-node spring is an uncoupled element, meaning the behavior of the spring in one direction does not affect other directions. Isolators and bearings are coupled elements. (page 83)

The nonlinear two-node spring element can be elastic or inelastic. The inelastic (hysteretic) spring element is described below.

**Inelastic (Hysteretic) Spring Element**

This element is defined using two joints. The element represents an inelastic, nonlinear connection between the joints. The length of the element may be zero, depending on the spring's direction property.

Compared to the elastic nonlinear two-node spring, selection of material properties is more restrictive for the hysteretic spring. The spring curves are restricted based on the selected backbone and polygonal hysteretic models. The backbone type of the hysteretic spring may be one of bilinear, trilinear, or fractural. The polygonal hysteretic model may be bilinear, yield-oriented, or vertex-oriented.

The material model is based on polygonal hysteretic models, which are models with piece-wise linear behavior. These are also referred to as multi-linear models. This material model can include stiffness and strength degradation, pinching, and slip.

**General Attributes**

**ID**

Spring numbers are assigned for identification purposes. They can be any integer, and they must be uniquely assigned among all springs in a project, but they need not be assigned consecutively.

**Start Joint (I Joint)**

The starting connectivity of the spring.

**End Joint (J Joint)**

The end connectivity of the spring. For grounded springs, leave this field unset.
Type
The behavior of the spring: linear, nonlinear, or hysteretic. For more information on the different types, see above.

Direction
Defines the direction of force or moment to which the spring reacts.

Local Axial & Local Torsional - Only available for two-node springs. The axis from joint I to joint J establishes the direction of the spring. For Local Axial, the motion is in the I-J direction. For Local Torsional, the motion is about the axis joining the joints. The length of the spring must be nonzero.

Translation X/Y/Z or Rotation X/Y/Z - Used to specify that the direction of the spring is relative to the global coordinate system or the displacement coordinate system of the spring's start joint. The directions are along or about the global coordinate system axes if the spring's start-joint has not been assigned a joint displacement coordinate system (page 55), or along or about a user coordinate system (page 35) if it has been assigned as the start-joint's displacement coordinate system. The end-joint's displacement coordinate system is ignored for determining the direction of the spring.

Structure/Construction Group
Structure groups can be assigned directly to the springs. User have option to select an existing structure group or specify a new one. Structure groups are used to modularize the geometry of the model and plays a key role in staged construction analysis. For more information see Structure Groups Explorer [see “Structure Groups Explorer” in LARSA 4D User’s Manual].

Stiffness Attributes
The stiffness of linear grounded springs, nonlinear grounded springs, and linear two-node springs can be given using the following properties. For linear springs, only K Tension may be used. K Compression is automatically given the same value. A 6x6 stiffness matrix may be provided instead for linear grounded springs, and a nonlinear spring curve may be provided instead for nonlinear grounded springs (see below).

These fields are not used for nonlinear two-node springs, which require a nonlinear spring curve. To model the nonlinear properties for a nonlinear two-node spring, modify the nonlinear spring curve property definition assigned to the spring accordingly.

K Tension
The elastic stiffness for the spring in tension in the specified spring direction. The spring stiffness for translation is force per linear translational displacement, such as kip/in. The spring stiffness for rotation is moment per unit rotation, such as kip-ft/radian. This number must be positive or zero.

K Compression
The elastic stiffness for the spring in compression in the specified spring direction. The nonlinear spring element can have compressive stiffness that differs from its tensile stiffness, but the linear spring cannot. This number must be positive or zero.

Maximum Tension/Compression
The spring element force can be restricted to a maximum specified value. Once this capacity is reached, the element will deform without any increase in the element force. The element can have a different force capacity for tension and compression.

Hook Distance
The element can be restricted to react to tension after the displacement exceeds the specified hook distance.
Gap Distance

The element can be restricted to react to compression after the displacement exceeds the specified gap distance.

The following can be used to define the nonlinear behavior of nonlinear elastic grounded springs and is used to define the behavior of all nonlinear two-node springs.

Spring Properties Definition

The string property definition specifies other spring properties (page 29).

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Springs spreadsheet, the commands are available in the Springs menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object only.

To apply the command to multiple springs at once, first select those springs and unselect everything else, then go to the Modify menu, find the Springs menu, and choose a command from within that menu. Activating these commands from the Modify menu will apply the action to all selected springs.

The additional commands are as follows:

Edit Spring Curve

This command opens the spring property spreadsheet on the spring property definition assigned to the spring.

Renumbering Setup

This command sets up renumbering options for springs. The user can choose the starting ID and the step.

Renumber

Renumbers the selected plates according to the renumbering setup options, which must be set first.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
- Spring Property Definitions on page 29.
- Joints on page 55.
Isolators

Isolators are two-node elements often used for protection from seismic forces. They have the behavior of a dashpot.

An isolator element has the behavior of a dashpot, such as a viscous damping device. These devices are widely used as structural protective systems for extreme loading (wind and earthquake) cases. Due to their combined benefits in terms of overall displacement reduction and energy dissipation, dampers are considered to be one of the most effective structural protective systems. For more details, see Isolator Property Definitions (page 33).

This element can only be used in the time history analysis (and only in the nonlinear time history analysis before LARSA 4D version 8.0) because the response of the element is velocity-dependent.

The fields required for isolator elements are given below:

**ID**
Isolator numbers are assigned to these elements for identification purposes. They can be any integer, and they must be uniquely assigned among all isolators in a project, but they need not be assigned consecutively.

**Start (I) and End (J) Joints**
An isolator's connectivity is described using two non-coincident joints. Isolators must have two joints and may not have zero length. The joints' IDs are used here.

**Properties Definition**
The behavior of the isolator is determined by the isolator property definition (page 33) selected for this field.

**Direction**
Defines the direction of velocity to which the isolator reacts.

- **Local Axial** - The isolator reacts to velocities parallel to the axis from the I-Joint to the J-Joint of the isolator.
- **Trans. X/Y/Z** - The isolator reacts to velocities parallel to one of the three global directions, or if the isolator's I-Joint has a joint displacement coordinate system (page 55) assigned then parallel to the X, Y, or Z local direction of that displacement coordinate system. The J-Joint's displacement coordinate system is ignored for determining the direction of the isolator.
- **Local Torsional** - The isolator reacts to angular velocities about the axis from the I-Joint to the J-Joint. The damping coefficient must be given in units of moment per units of velocity.
- **Rot. X/Y/Z** - The isolator reacts to angular velocities about one of the three global directions, or if the isolator's I-Joint has a joint displacement coordinate system (page 55) assigned then about the X, Y, or Z local direction of that displacement coordinate system. The J-Joint's displacement coordinate system is ignored for determining the direction of the isolator. The damping coefficient must be given in units of moment per units of velocity.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
- Isolator Property Definitions on page 33.
Mass Elements

Mass elements represent concentrated masses or mass moments of inertia at joints for dynamic analyses. They are not used to apply static weight loads.

The mass of the structure is used to compute inertial forces in a dynamic analysis. The mass matrix of the structure is obtained by computing the contribution of each element connected to that joint using the weight density of the material and the volume of the element. If the weight density is zero, then the contribution of that element to the mass of the structure becomes zero.

If it is necessary to place additional concentrated masses or mass moments of inertia at the joints, the lumped masses and rotational mass moments inertia can be defined at the joints of the structure using Mass Elements. The mass at a joint includes the mass assigned to that joint using the mass element data plus the contributions from the weight of all elements connected to the joint.

Mass at restrained degrees of freedom are ignored in the analysis. The masses are always specified in the directions of the displacement coordinate systems assigned to the joints, if one has been assigned. If the joint is not assigned a displacement coordinate system, then the masses are in the global directions.

Mass values are entered as weights. The mass is automatically calculated by dividing the weight by gravitational acceleration.

**Joint**

The ID of the joint (page 55) to which this mass will be applied.

**Translational Mass (X/Y/Z)**

The applied translational mass.

**Rotational Mass (X/Y/Z)**

The applied rotational mass.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- Joints on page 55.
DOF Constraints

Degree of Freedom (DOF) Constraints can be used to enforce equal-displacement behavior of two joints in any degree of freedom.

A practical use of DOF constraints is for connecting different parts of a model or in reducing the number of equilibrium equations for certain types of structures.

The use of this feature can be illustrated where a bridge girder is supported vertically on a pier but is free to move laterally. To model such behavior, the joint on the beam and the joint on the column are defined as two separate joints sharing the same location. To simulate the interfacing with equal vertical displacement, the degree of freedom in the vertical direction of the column joint is linked to the vertical displacement direction of the joint on the beam. The joints will move independently in the horizontal direction while they will have the same vertical displacement.

The directions of motion are with respect to the displacement coordinate systems of the dependent and master joints. When no displacement coordinate systems are specified, the global coordinate system is used.

When the displacement directions of the dependent and master joints are not the same, bear in mind that degrees of freedom, and not particular directions of displacement, are being linked. For instance, if the X-Translation degree of freedom for the dependent joint is in the direction of the global y-axis, and the X-Translation degree of freedom for the master joint is in the direction of the global z-axis, then applying a DOF constraint for X-Translation between these joints will cause the dependent joint's global y-displacement to be equal to the master joint's global z-displacement.

**Dependent Joint**
- The ID of a joint to constrain. The dependent joint will move together with equal displacement as of the master joint specified for each direction (degree of freedom). The dependent joint will move independently in the degrees of freedom where a master joint is not specified.

**Master Joint for X-Translation**
- If a joint is specified, then the X-Translation of the dependent joint will be equal to X-Translation of this joint.

**Master Joint for Y-Translation**
- If a joint is specified, then the Y-Translation of the dependent joint will be equal to Y-Translation of this joint.

**Master Joint for Z-Translation**
- If a joint is specified, then the Z-Translation of the dependent joint will be equal to Z-Translation of this joint.

**Master Joint for X-Rotation**
- If a joint is specified, then the X-Rotation of the dependent joint will be equal to X-Rotation of this joint.

**Master Joint for Y-Rotation**
- If a joint is specified, then the Y-Rotation of the dependent joint will be equal to Y-Rotation of this joint.

**Master Joint for Z-Rotation**
- If a joint is specified, then the Z-Rotation of the dependent joint will be equal to Z-Rotation of this joint.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
- Joints on page 55.
Tendons

Tendons can be included in a structure for modeling the pre- and post-tensioning of members. Tendons are generally used in the Staged Construction Analysis (page 229) and include both short-term and long-term losses.

Tendons are groups of strands, wires, or rods that are stressed against a concrete member after the concrete is set. The tendons are typically given a profile to enhance their load resisting characteristics.

Tendons are defined in two parts. First, the basic properties like material and jacking force are set in the Tendons spreadsheet. Then, the tendon path geometry through the structure is set.

About the Tendon

The tendon modeling technique used in LARSA is known as Equivalent Load through Discretization of Tendon Force. The tendon force is discretized along its length to account for the following:

- Variation of force along the tendon length caused by the short-term effects of friction and seating of the tendon at stressing.
- Long-term losses in prestressing due to relaxation, creep, shrinkage, and elastic shortening.

This method has the advantage of implicitly accounting for the change in geometry of nonprismatic sections, which are commonly used in bridge construction projects. An added advantage is that when the primary moment at each section is computed using the adjusted force at the section, the prestress losses along the tendon are included.

For an internal tendon idealized as a series of straight segments there is a gradual stress loss along each segment due to wobble friction and curvature. External tendons are subject to losses due to curvature at the deviators. The component of friction due to curvature (angle change) is concentrated at the intersection of the segments. Hence the force distribution is represented by a series of sloping lines with steps at the discretization points.

Using this method, both the immediate prestress losses and the long-term losses are rigorously accounted for. With the iterative solution method employed in the nonlinear analysis, at each load step (construction stage) the prestress losses are adjusted based on the current prestressing force, deformation caused by subsequently stressed tendons, applied loads, creep, and shrinkage. The current prestressing force is then used to compute the equivalent forces which will in turn change the long-term losses.

Friction Losses

For post-tensioned members, tendons are usually anchored at one end and stretched with the jacks at the other end. As the steel tendon slides through the duct, frictional resistance is developed. The tension at the anchored end is less than the tension at the jack. The total friction loss is the sum of the wobble friction due to unintentional misalignment, and the curvature friction due to the intentional curvature of the tendon. These effects are considered separately and then combined. The following is the relationship between the jacking force and the reduced force due to friction at a distance on the tendon:

\[ \ln\left(\frac{P_s}{P_x}\right) = KL + ma \]

where \( P_s \) is the prestress force at the jack and \( P_x \) is the reduced value at a distance \( L \) from the jack. \( K \) is the wobble coefficient. \( m \) is the curvature friction and \( a \) is the change in angle.
External tendons are not subject to friction loss, and since they do not have curvature between deviators, loss due to angle change occurs only at deviators.

**Anchorage Slip Losses**

In post-tensioned members, when the jacking force is released, the steel tension is transferred to the concrete by special anchorages. Inevitably, there is small amount of slip at the anchorages upon transfer, as the wedges seat themselves into the tendons or as the anchorage hardware deforms.

Losses due to anchor slip are computed and displayed along with short-term losses prior to an analysis.

**Elastic Shortening of Concrete Losses**

The stress loss in each tendon due to elastic shortening of concrete will depend on the total number of tendons in the concrete member and the sequence of stressing.

When the prestressing force is transferred to a concrete member, there will be elastic shortening of the concrete as it is compressed. For post-tensioned members having a single tendon, the elastic deformation of the concrete takes place when the jacking force is applied, and there is automatic compensation for elastic shortening loss, which therefore need not be calculated. If all the steel tendons in a post-tensioned beam are tensioned at the same time, there will be no loss due to elastic shortening.

However, for the common case where multiple tendons are used, with the tendons tensioned in sequence, there will be losses due to elastic shortening of the concrete. As each tendon is stressed, the compression in concrete members increases. The elastic shortening of the concrete due to the increase in compressive stress causes a loss of prestressing force in tendons which were previously stressed and anchored. For example, the first tendon will lose stress when the second is tensioned, and the first and second will lose stress when the third tendon is stressed. The elastic shortening loss is calculated for each tendon starting with the last tendon, for which there will be no loss due to elastic shortening.

This type of loss is included with long-term losses, calculated in the time-dependent staged construction analysis.

**Other Losses**

Creep and shrinkage of concrete and the application of other loads result in stress loss in post-tensioning. These are computed and accumulated through the Staged Construction Analysis.

There is a gradual reduction of stress in post-tensioning resulting from relaxation even though the length of the tendon is held constant. The amount of relaxation depends on the intensity of steel stress as well as time.

**Attributes**

These properties are in the Tendons tab of the Geometry Spreadsheets, which is under the Input Data menu. Alternatively, double-click the name of the tendon in the tendons section of the Model Data Explorer [in LARSA 4D User’s Manual].

**Tendon Name**

Names are assigned to identify tendons.

**Design Group**
This is a name assigned by the user and does not affect the analysis.

**Material**
The material (page 19) of the tendon.

**Strand Area**
Cross-sectional area of each strand.

**# of Strands**
Total number of strands in the tendon.

**Proposed Jacking Force**
The proposed jacking force applied to the tendon from which the applied jacking forces are computed.

**Applied Jacking Force (at Start/End)**
The jacking force applied to the tendon in the analysis. It is automatically computed based on the proposed jacking force, anchor set, and peak stress ratios. This field must be manually updated by the Compute Jacking Force command (see below).

**Jacking End**
The application point or sequence of the jacking force: At Start, At End, or At Start Then End.

**Anchor Set**
The length of the anchor set.

**Wobble Coefficient**
The wobble coefficient. Wobble friction is due to unintentional misalignment. It is specified per unit length. External tendons may not have a wobble coefficient.

**Curvature Friction Coefficient**
The curvature friction coefficient, which is due to the intentional curvature of the tendon. This coefficient is unitless. Since external tendons may only have curvature at deviators, the wobble coefficient is only applied at deviators.

**Peak Stress Ratio - Interior**
The limiting stress for the interior points of the tendon as a proportion from 0.0 to 1.0 of the guaranteed ultimate tensile strenght (GUTS) specified in the tendon's material, used to compute the Applied Jacking Force.

**Peak Stress Ratio - Ends**
The limiting stress for the start and end points of the tendon as a proportion from 0.0 to 1.0 of the guaranteed ultimate tensile strenght (GUTS) specified in the tendon's material, used to compute the Applied Jacking Force.

**Elongation**
The elongation of the tendon after each pull and at each stationary, automatically computed by LARSA for informational purposes.

**Path Geometry**
Path geometry for a tendon is accessed from the tendons spreadsheet above by right-clicking a tendon and choosing *Edit Path & View*. Alternatively, right-click the name of the tendon in the tendons section of the Model Data Explorer [in *LARSA 4D User’s Manual*] and choose the same command.
Paths consist of a list of geometry points that determine the points in space that the path follows. A path can contain any number of geometry points, and they are specified with reference to the member that the point falls within. The force in the tendon will be applied to the structure only within the members specified in the path.

By default, the path will consist of straight line segments connecting each of the geometry points. Parabolic, cubic, and circular curve fitting may be applied at geometry points to smooth out transitions between segments, as described below.

The following describes the data used to define each point in a path.

**Point Type**

Not every member that a tendon passes through will need a geometry point, but each member passed through must be specified in the path, in the order in which the tendon passes through them. For those members the tendon passes through but in which no geometry control point is needed, use the path only point type. Otherwise, use the geometry point type. The span break point type is not used for tendons.

**Reference Object Type**

Only the member option is applicable for tendons.

**Reference Object or Range**

- For geometry points, specify the ID of the member that this point is within. The offset coordinate for this point is specified in the local coordinate system of this member. Specify a range of IDs, such as 12-36, or even 36-12, to repeat this point for each member in the range, in the order specified.
- For path only points, specify an ID or range of IDs for the members that this point is acting as a placeholder for.

The remaining fields are for geometry point only.

**Offset (X, Y and Z)**

The offset of this point from some reference position within the member given above.

Offset directions are in the member's reference axis directions, rather that in global axis directions. The reference axis is generally the same as the member local axis, along the joint-to-joint line of the member. However, if member end offsets (page 57) or section centroid offsets (page 23) are applied to the member, the reference axis may not match either the joint-to-joint line or the member local axis. If there are member end offsets (page 57), the reference line follows the member local axis, shifted from the joint-to-joint line by the end offsets. The reference axis is not affected by section centroid offsets (page 23), which means the reference line is constant despite any nonprismatic variation in the section affecting the location of the centroid. If there are section centroid offsets (page 23), which do affect the member local axis, then the reference axis and the member local axis will not match.

The meaning of the x, y, and z offset fields depends on the choices made in the x, y, and z reference fields, described next. The X offset is specified in coordinate units, for convenience, while the Y and Z offsets are specified in section units.

**X Reference**

The position that the X Offset starts from.

Start: The X Offset is the x-distance from the start of the member, going toward the end of the member.

End: The X Offset is the x-distance from the end of the member, going toward the start. Positive values for X Offset are toward the start of the member.

**Y Reference**

The position that the Y Offset starts from.
**+Y1 Local Edge:** The **Y Offset** is the y-distance from the 1st stress recovery point of the member's section, which is generally used to specify the top of a member. Positive values are "down" into the member, in the negative-y direction. (Note that this differs from the Z Offset.)

**Reference Line:** The **Y Offset** is the y-distance from the reference line of the member, which is the joint-to-joint line of the member. For standard sections, the reference line is the same as the COG. Positive values are "up" or in the positive direction of the member y-axis.

**-Y3 Local Edge:** The **Y Offset** is the y-distance from the 3rd stress recovery point of the member's section, which is generally used to specify the bottom of a member. Positive values are "up" into the member, in the positive-y direction. (Note that this differs from the Z Offset.)

**Z Reference**

The position that the **Z Offset** starts from.

**-Z3 Local Edge:** The **Z Offset** is the z-distance from the 3rd stress recovery point of the member's section, which is generally used to specify the left side of a member. Positive values are "left" away from the member, in the negative-z direction. (Note that this differs from the Y Offset.)

**Reference Line:** The **Z Offset** is the z-distance from the centerline of the member (the center of gravity, generally speaking). Positive values are "toward the right" or in the positive direction of the member z-axis.

**+Z1 Local Edge:** The **Z Offset** is the z-distance from the 1st stress recovery point of the member's section, which is generally used to specify the right side of a member. Positive values are "right" away from the member, in the positive-z direction. (Note that this differs from the Y Offset.)

The figure below shows three geometry points in a tendon path. The first two points have X Reference set to "start", so their X Offset is the distance from the start of the member; for the third point, whose X Reference is "end", the X Offset is the distance from the end of the member. Each point also has a different Y Reference, changing how the Y Offset is applied.

![Member Side View](Image)

**Using Offsets and References**

In a nonprismatic section, the reference axis does not match the member local axis at the centroid. The figure below shows two geometry points and that they are offset perpendicularly from the joint-to-joint line of the member, which is the reference axis.
Offsets and Reference in a Nonprismatic Section

Curvature Type
Determines how curve fitting is applied at this point.

No Curve: No curve fitting will be used at this point. There will be a sharp corner at this point if the tendon turns.

Parabolic:
Parabolic curve fitting is used on either side of this point. The direction of a tangent line to the path at this point is specified as tangency angles in the XY and XZ planes of the member reference axis. Zero degrees specifies the curve's tangent at this point is parallel to the member reference axis.

Note that each segment between consecutive rows in the path spreadsheet is fit as a separate curve. Since each segment is between two points, and not three, LARSA 4D does not do the typical process of fitting a parabola between three points. Instead, it fits the curve between two points according to the tangent directions at each point. The largest curve that fits between the two points will be used. Two independent parabolic equations are solved in separate planes and then combined to produce the three-dimensional curve.

To solve the parabolic equation, the tangent lines must intersect. If no such point exists in one of the two planes, cubic curve fitting may be used in that plane. If no such point exists in either plane, cubic curve fitting will be used in an arbitrary plane instead.
Tangent directions perpendicular to the reference axis (plus or minus 90 degrees), or perpendicular to the lines connecting this point with the previous or next point, will not be accepted. To use parabolic curve fitting, specify the **parabolic** curve type for the two points on either side of the curve.

![Parabolic Curve Fitting](image)

**Circular Radius.** A sharp corner at this point is smoothed using the arc of a circle. The radius of the arc is specified, and the tendon will not actually pass through this point. The coordinates of this point give the PI point (point of intersection) for the curve. The PT and PC points are automatically determined from the radius. This method is similar to parabolic curve fitting, except the PI point is given to determine the tangent directions, and the radius of the arc is given by the user. Circular radius curvature cannot be used at the start or end of a tendon path.

The circular arc is created in the plane defined by the PI point and the geometry points immediately before and after the PI point.

![Circular Radius Curve Fitting](image)

**Cubic:** Cubic curve fitting is used on either side of this point. The direction of a tangent line to the path at this point is specified as tangency angles as with parabolic curve fitting. Two independent cubic equations are solved in separate planes and then combined to produce the three-dimensional curve. To use cubic curve fitting, specify the **cubic** curve type for the two points on either side of the curve.
Example Path

The image below shows a tendon passing through five members. From members 1 to 3, the tendon is straight. From members 3 to 5, the tendon follows a parabolic curve. Three geometry points are used to create the path, the first two at the ends of the linear section, and the last two around the parabolic section.

An Example Tendon Path

This tendon would be defined as follows. The Z offset, reference, and angle columns have been omitted for brevity. Note that members 2 and 4 are not relevant for the path geometry, but they are included at path points in the appropriate locations.

<table>
<thead>
<tr>
<th>Point Type</th>
<th>Ref. Type</th>
<th>Ref. Object</th>
<th>Offset X</th>
<th>Offset Y</th>
<th>X Ref</th>
<th>Y Ref</th>
<th>Curve Type</th>
<th>Angle XY</th>
</tr>
</thead>
<tbody>
<tr>
<td>geometry</td>
<td>member</td>
<td>1</td>
<td>0</td>
<td>8</td>
<td>Start</td>
<td>+Y1 Edge</td>
<td>No Curve</td>
<td>0</td>
</tr>
<tr>
<td>path</td>
<td>member</td>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>3</td>
<td>0</td>
<td>8</td>
<td>Start</td>
<td>+Y1 Edge</td>
<td>Parabolic</td>
<td>0</td>
</tr>
<tr>
<td>path</td>
<td>member</td>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>5</td>
<td>0</td>
<td>2</td>
<td>End</td>
<td>+Y1 Edge</td>
<td>Parabolic</td>
<td>60</td>
</tr>
</tbody>
</table>
Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Tendons spreadsheet, the commands are available in the Tendons menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the Model Data Explorer, change to the display for Tendons. Then right-click an object to get this list of commands. You can also apply the command to multiple at once by clicking on the first object, then holding the SHIFT key and clicking on the last object, and last right-clicking any of the to get the list of commands.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object.

The additional commands are as follows:

**Edit Path and View**

This command opens a new spreadsheet window for editing the tendon path and a new graphics window for viewing the path.

**Edit Material**

This command opens the materials spreadsheet to the material assigned to the tendon.

**Duplicate Tendon**

This command duplicates the tendon.

**Generate Equivalent Loads**

This command creates a new load case (page 107) with equivalent member loads based on the forces in the tendon, including anchor set.

**Generate Equivalent Loads (before Anchor Set)**

This command creates a new load case (page 107) with equivalent member loads based on the forces in the tendon before anchor set is applied.

**Compute Jacking Force**

Based on the limiting peak stress ratio, proposed jacking force, and short term losses, the applied jacking force is computed. After changing any values that alter the computation of the applied jacking force, this command must be re-run.

The applied jacking force is determined as follows. The limiting force is calculated as the product of the peak stress ratio, material GUTS, strand area, and number of strands. The maximum proposed force is calculated as the maximum force in the tendon jacked with the proposed jacking force, immediately after anchor set. If the maximum proposed force does not exceed the limiting force, the applied jacking force is set to the proposed jacking force. Otherwise, the applied jacking force is taken as the proposed jacking force multiplied by the ratio of the limiting force to the maximum proposed force. This procedure is done twice, first using the interior
peak stress ratio and the forces all along the tendon, then using the end peak stress ratio and only the force at the start and end of the tendon. The minimum of the two applied jacking forces found is used.

**Mirror Y Offsets**

This tool changes all y-offsets in the tendon path to the negative of the offset provided

**Mirror Z Offsets**

This tool changes all z-offsets in the tendon path to the negative of the offset provided

---

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual.*
Lanes define loading paths for vehicles in the moving load analysis (page 203) and for influence-based analysis (page 205).

A lane is a line or surface that travels through a structure. In moving load and influence analysis, the load pattern applied to the structure follows a defined lane path, starting at one end of the path and moving toward the other end. The path of a lane is determined by geometry control points between which straight lines or curves are fit.

Geometry control points are given as offsets relative to members or plates in the model. A lane's geometry points may reference either all members or all plates. When a lane is defined with reference to members, every member that the lane passes through must be present in the path. Loads will be applied directly to the members listed in the path, with added moment loads as needed if the lane does not follow the member centroid. If the path references plates, only plates containing geometry control points need to be included in the path. Loads will be applied to any plates that the path passes through.

For influence surfaces, the lane must be defined with reference to plates, and not members. Only in this case may a lane width be specified. For a standard moving load analysis and for influence line analysis when the lane reference members, the lane width is ignored for the purposes of the analysis. It may be used by design tools, however.

The type of vehicle to march on the lane is set independently of the lane definition. Use Moving Loads (page 125) or Influence Loads (page 127) to set vehicle parameters.

Basic Properties

These properties are in the Lanes tab of the Geometry Spreadsheets, which is under the Input Data menu. Alternatively, double-click the name of the lane in the lanes section of the Model Data Explorer [in LARSA 4D User’s Manual].

Lane Name
A name is assigned by the user to identify the lane.

Lane Width
The width of the lane where loads can be positioned. This field is used only for influence surface analysis. Note that for influence surfaces, the lane path must reference only plates and not members, and must have a nonzero width.

Skew at Start/End
When the lane width is nonzero, skews the start and end of the lane so that the lane is trapezoidal rather than rectangular. The skew value represents the slope of the edge, with 0 being perpendicular to the direction of the lane and 1 being a 45-degree skew. To compute the skew, find the values of $x$ and $y$ in the figure below. The value of skew-at-start is $x/y$. In an influence-based live load analysis with UDL patch loading, skew-at-start and skew-at-end must be equal.
Path Geometry

Path geometry for a lane is accessed from the lanes spreadsheet above by right-clicking a lane and choosing Edit Path & View. Alternatively, right-click the name of the lane in the lanes section of the Model Data Explorer [in LARSA 4D User's Manual] and choose the same command.

Paths consist of a list of geometry control points and path-only points. Geometry points determine the points in space that the path follows. A path can contain any number of geometry points, and they are specified with reference to the member or plate that the point falls within. Geometry points can be offset from member centroids to model lanes with eccentricity.

A path may reference either members or plates, but not both. When a path references members, it must list every member the path passes through in the right order. Although every member must be listed, there does not need to be a geometry control point in each member. For instance, a straight line lane needs only two geometry points, one at the start and one at the end, but must list every member that the lane passes through between the geometry points. Include those members with path-only points mixed in with the geometry points so that the members appear in the order in which the lane passes through them.

When a path is used to create an influence surface, bear in mind that the geometry of the surface (a rectangle in 3D space) is distinct from the set of plates to which influence surface unit loading will be applied. It is typically not necessary to list all of the plate elements to which influence surface loading will be applied in the path spreadsheet. LARSA 4D will automatically determine which plates to load. However, if the surface is elevated above the plate elements or if the model has other plate elements below the surface that do not intersect with the surface, then it may be necessary to list the plate elements that the influence surface loading should be applied to as path-only rows in the path spreadsheet. If no path-only rows are in the path spreadsheet, LARSA 4D may attempt to load any plate in the model that is close to the surface. If any path-only rows are given, then LARSA 4D will only attempt to load plates that are included in the path spreadsheet (either as geometry points or as path-only points).

The following describes the data used to define each point in a path.
**Point Type**

Not every member that a lane passes through will need a geometry point, but each member passed through must be specified in the path, in the order in which the lane passes through them. For those members the lane passes through but in which no geometry control point is needed, use the *path only* point type. Otherwise, use the *geometry* point type.

For lanes used for influence surfaces, if any path-only rows are given, then LARSA 4D will only attempt to load plates that are included in the path spreadsheet (either as geometry points or as path-only points). If no path-only rows are present, LARSA 4D may attempt to load any plate elements in the model.

The *span break* point type is a marker indicating that the lane has crossed into another span on a bridge, for design purposes. This does not affect the computation of the influence coefficients, but the location of span breaks are used in influence-based results (page 205) for compliance with the AASHTO LFD and BS5400 codes. The span break marker indicates that the previous geometry point gives the coordinates of the end of a span; thus, span break markers should immediately follow a geometry point positioned over a support.

**Reference Object Type**

Select whether this point references a *member* or *plate*.

**Reference Object or Range**

- For geometry points, specify the ID of the member or plate that this point is within. The offset coordinate for this point is specified in the local coordinate system of this member or plate. Specify a range of IDs, such as 12-36, or even 36-12, to repeat this point for each member or plate in the range, in the order specified.
- For path only points, specify an ID or range of IDs for the members that this point is acting as a placeholder for.

The remaining fields are for geometry point only.

**Offset (X, Y and Z)**

For geometry points, the offset of this point from some position within the referenced object. Offset directions are in the element's reference axis directions, rather than in global axis directions. The offset is by default relative to the start of the member or the I-joint of the plate. Y- and z-offsets in lanes on members will create moments. For influence surfaces, a z-offset will create a moment if the load direction is either Local X or Local Y.

**Reference (X, Y and Z)**

For points referencing members, set the X Reference to *End* to position the point relative to the end of the member. Positive values for X Offset are then toward the beginning of the member.

The y and z reference fields are not normally changed from *Reference Line* so that the y and z offset fields are in the member reference axes. See the documentation for tendon paths (page 89) for more on the other choices for the y and z reference fields.

By default, geometry points on members will fall at the start of the member on its reference line, which normally is at its centroid (except see section centroid offsets (page 23)). Loading at this position will be applied directly to the member without any eccentricity. To apply eccentricity, use the offset fields described above.

Points that reference plates by default will fall at the I-joint of the plate. Use the X and Y offsets, which are in the plate's local coordinate system, to position the geometry point.
The geometry points on the path will by default be connected by straight lines, meeting at hard corners if the lane turns. For descriptions of the curve fitting options, see the documentation for tendon paths (page 89).

**Lane Path Example**

A straight lane that passes through members 1 through 10 should be defined as follows:

<table>
<thead>
<tr>
<th>Point Type</th>
<th>Ref. Type</th>
<th>Ref. Object or Range</th>
<th>Offset X</th>
<th>Offset Y</th>
<th>Offset Z</th>
<th>X Ref</th>
<th>Y Ref</th>
<th>Z Ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>geometry</td>
<td>member</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>Start</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
<tr>
<td>path</td>
<td>member</td>
<td>2-9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>End</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
</tbody>
</table>

Note that the last point has its X Reference set to **End** so that the lane goes through all of member 10, rather than stopping at its start joint.

If members 1 through 10 are on a curve, the lane above will not follow the curve. It will only be a straight line between the two geometry points defined. To have the lane follow the curve, there should be a geometry point at the start and end of every member. Without these geometry points, the lane will not exactly following the centroids of the members, leading to inadvertent eccentricities. That lane is defined as follows:

<table>
<thead>
<tr>
<th>Point Type</th>
<th>Ref. Type</th>
<th>Ref. Object or Range</th>
<th>Offset X</th>
<th>Offset Y</th>
<th>Offset Z</th>
<th>X Ref</th>
<th>Y Ref</th>
<th>Z Ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>geometry</td>
<td>member</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>Start</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>Start</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>3</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>Start</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
<tr>
<td></td>
<td></td>
<td>...</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>Start</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>End</td>
<td>Reference Line</td>
<td>Reference Line</td>
</tr>
</tbody>
</table>

Note that member 10 is referenced twice so that a point is placed at its start and end. This can be abbreviated as:
<table>
<thead>
<tr>
<th>Point Type</th>
<th>Ref. Type</th>
<th>Ref. Object or Range</th>
<th>Offset X</th>
<th>Offset Y</th>
<th>Offset Z</th>
<th>X Ref</th>
<th>Y Ref</th>
<th>Z Ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>geometry</td>
<td>member</td>
<td>1-10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>Start</td>
<td>Line</td>
<td>Line</td>
</tr>
<tr>
<td>geometry</td>
<td>member</td>
<td>10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>End</td>
<td>Line</td>
<td>Line</td>
</tr>
</tbody>
</table>

**Additional Commands**

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Lanes spreadsheet, the commands are available in the Lanes menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

In the Model Data Explorer, change to the display for Lanes. Then right-click an object to get this list of commands. You can also apply the command to multiple at once by clicking on the first object, then holding the SHIFT key and clicking on the last object, and last right-clicking any of the to get the list of commands.

In the graphics window, activate the Pointer mouse tool from the toolbar, and then right-click an object in the graphical display to get the list of commands to apply to that object.

The additional commands are as follows:

**Edit Path and View**

This command opens a new spreadsheet window for editing the lane path and a new graphics window for viewing the path.

**Transfer to Plates**

This command duplicates the lane and modifies all member-based geometry points in the path that fall within a plate element to reference the plate element instead. The offset fields are updated appropriately.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- Moving Loads on page 125.
- Influence Loads on page 127.
- Moving Load Analysis on page 203.
Structure loading is defined in load cases. There are several types of load cases, the most commonly used of which is the static load case. In static load cases, joint, member, plate, thermal, and other types of static loads can be applied to the structure. The other load case types include moving load cases, time history cases, and response spectra cases.

The type of analysis chosen determines how load cases are used. Static analyses (Linear Static Analysis (page 161), P-Delta Analysis (page 163), and Nonlinear Static Analysis (page 165)) require static load cases and load combinations of static load cases. Eigenvalue analyses (page 181) do not make use of load cases at all. The moving load analyses (Moving Load Analysis (page 203) and Influence Line & Surface Analysis (page 205)) require moving load cases with moving loads. The time history analyses (page 193) require a time history load case with time history excitation loads and, for some models, initial conditions.

The Response Spectra Analysis (page 183) can be performed with or without static load cases, but it requires a response spectra case with setup information.

In Staged Construction Analysis (page 229), static load cases are used to define the patterns of loading that will be applied on the structure at points throughout construction, but they are not applied to the structure until they are assigned to construction stages.

### Static Load Cases

- Load Classes for Code-Based Results
- Load Combinations
- Self Weight
- Joint Loads
- Support Displacements
- Member Loads
- Member Thermal Loads
- Plate Loads

### Load Combinations

- Including Response Spectra Cases
- Including Moving Load Cases
- Use in Nonlinear Analysis

### Self Weight

- Weight Factor and Gravity Load Directions
- Self Weight Computation
- In Staged Construction Analysis

### Joint Loads

### Support Displacements

### Member Loads

### Member Thermal Loads

- Load Input
- Nonlinear Thermal Gradient Curves

### Plate Loads

- Plate Load Fields
- Point Load Coordinates
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moving Loads</td>
<td>125</td>
</tr>
<tr>
<td>Influence Loads</td>
<td>127</td>
</tr>
<tr>
<td>Time History Loads</td>
<td>129</td>
</tr>
<tr>
<td>Excitation Functions</td>
<td>129</td>
</tr>
<tr>
<td>Initial Conditions</td>
<td>130</td>
</tr>
</tbody>
</table>
Static Load Cases

A static load case represents one scenario of static loading.

The following loads may be applied in static load cases.

- **Self Weight** (page 111): for automatic computation of gravitational loading based on section and material properties of elements.
- **Joint Loads** (page 113): Forces and moments applied at joints
- **Support Displacements** (page 115)
- **Member Loads** (page 117) on beams (point loads, uniformly or partially distributed loads, and triangular loads), including pre- and post-tensioning of members such as cables
- **Member Thermal Loads** (page 119): Uniform thermal loads for beam, truss, and cable elements, and thermal gradients for beam elements
- **Plate Loads** (page 121): Concentrated and uniform loads applied on plate and shell elements
- **Thermal Plate Loads** (page 121): Temperature change through the thickness of plate and shell elements resulting in uniform and thermal gradients

Load cases can be marked *Inactive*. Inactive load cases are excluded from analysis runs.

Load cases also have the option "Is Dynamic Mass?" This option affects the dynamic analysis types only, including eigenvalue, RSA, and time history. When this option is Yes, the loads in this case are treated as representing mass. All load cases with the option set to Yes are included as dynamic mass, in addition to any other mass specified in the model, in all dynamic analysis types.

**Load Classes for Code-Based Results**

Load cases can be assigned a load class, such as dead load, rain, and vehicular live load. These classes have been gathered from various codes and are used to pick out class-based results and to automate the process of creating class-based load combinations. Load classes can be chosen from the Load Cases Explorer [in *LARSA 4D User’s Manual*].

To create code-based load combinations, the Automatic Code-Based Load Combinations [in *LARSA 4D User’s Manual*] tool is useful. Class-based cumulative results can be obtained in the results spreadsheets [see “Results Spreadsheets” in *LARSA 4D User’s Manual*] in staged construction analysis.

For More Information, please refer to the following documentation.

- For help creating and editing load cases, see Load Cases Explorer in *LARSA 4D User’s Manual*.
- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- Analysis Reference on page 155.
Load Combinations

Load combinations represent the combined application of load cases. They are defined before an analysis is run.

In a linear analysis, load combinations are computed by linearly combining the results of the static cases. However, in the nonlinear analysis types, load combinations are computed by analyzing the load combination's load cases consecutively, carrying the effect from each case into the next.

Load combinations are most often used for combining static load cases with other static load cases and combining static cases with seismic cases. They can also combine static cases with moving load cases, but the behavior of load combinations varies slightly when non-static load cases are included. Use Linear Result Combinations [in LARSA 4D User’s Manual] to combine the results of a Time History Analysis with other results.

Load combinations can be analyzed in static, moving load, and response spectra analyses.

Each load case in a combination can be assigned a Load Factor which determines its contribution to the combination.

Load combinations are primarily needed for codes, to check or design the structure under various loading conditions. Linear Result Combinations [in LARSA 4D User’s Manual] can also be used to combine results. Automatic Code-Based Load Combinations [in LARSA 4D User’s Manual] can be used to automatically create code-based combinations.

Including Response Spectra Cases

Response Spectra Analysis (page 183) results generally do not have a sign convention. When an RSA case is included in any combination, its contribution is included with positive sign multiplied by the load factor.

To account for both the positive and negative directions of an earthquake, create two load combinations using the RSA case, one with a positive load factor for the RSA case and the other with a negative load factor.

Including Moving Load Cases

Up to two Moving Load Analysis (page 203) cases can be included in a Load Combination. If one moving load case is included in a combination, then LARSA creates a group of result cases, one for each generated case for the moving load case, each combining the effects of the static loads in the combination and the corresponding moving load result case.

If two moving load cases are included in a combination, then LARSA generates a group of generated load cases, one for each combination of a generated case from the first moving load case and a generated case from the second moving load case. If there are 1,000 generated cases in the first group and 1,000 in the second group, then one million cases will be generated. Because the number of generated load cases can quickly grow out of hand, using the Influence Line & Surface Analysis (page 205) is recommended instead.

Use in Nonlinear Analysis

In a nonlinear static analysis, load combinations are treated differently than in a linear static analysis, and their behavior can vary between analysis types.
Nonlinear Static Analysis (page 165)
Load cases within a Load Combination represent increments. The analysis is performed for each case in the combination as the load vector of the case multiplied by the corresponding load factor. The results of each increment are not saved and cannot be viewed later. Only the results after all cases have been applied are reported.

Nonlinear Buckling (page 169) and Restart Analysis
Load combinations themselves are treated as incremental load steps. Results are accumulated from one load combination to the next. Only static load cases can be included in the combinations.

For More Information, please refer to the following documentation.
• For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
• For help on using the Load Cases Explorer, see Load Cases Explorer in LARSA 4D User’s Manual.
• Linear Result Combinations in LARSA 4D User’s Manual.
Self Weight

The automatic computation of self-weight (gravity loading) can be included in static load cases (page 107) by specifying a weight factor in the direction of gravity. The weight factor for X, Y and Z are specified with the load case description data.

Weight Factor and Gravity Load Directions

The gravity load directions are the directions of the Global Coordinate System. A positive factor implies that self-weight is acting in the positive direction of the axes. A negative factor implies that self-weight is acting in the negative direction.

The user may choose to apply the gravity loading in one direction at a time or in any combination of X, Y and Z directions.

A value of -1.00 for Weight Factor Z would instruct the program to apply 100 percent of the computed self weight of the structure in the negative Z-axis direction, which properly simulates gravity for a structure where the Z-axis is up. A value of 1.00 for Weight Factor X and 0.50 for Weight Factor Y would instruct the program to apply 100 percent of the computed self weight in the direction of X-axis and 50 percent in the direction of Y-axis simultaneously.

The self weight factors are used in static analyses only. For dynamic analysis, the mass due to the weight of an element is computed and applied automatically in all translational directions X, Y and Z when the weight density of the material is specified as nonzero. See also Mass elements (page 85). You can create additional mass with the dynamic mass option on static load cases (page 107).

Self Weight Computation

The weight of members (page 57) are computed using the cross sectional area, element length and weight density. The weight of plates (page 71) are computed using the element volume and the weight density.

The self-weight of beam elements are modeled using uniformly distributed loads. This approach generates a consistent load vector considering the bending and shear effects due to the self-weight. The program computes the equivalent uniform load using the gravitational factors, direction, section area and weight density. Nonprismatic members use the average of the cross-sectional area at the member ends.

For plates, self-weight is computed based on material weight and thickness. Uniform pressure loads are used.

Springs, isolators, and tendons do not contribute to the weight of the model.

In Staged Construction Analysis

In a Staged Construction Analysis (page 229), self-weight loading must be applied to include the self-weight of elements in the analysis. Place a load case with self-weight turned on in every step in which new elements are constructed if self-weight is desired for those elements.
Self-weight load cases only apply self-weight to elements constructed in the same construction step as the self-weight load case. Therefore you must place a self-weight load case in the same construction step in which elements are constructed.

If a self-weight load case is placed in a construction step without a construction activity, self-weight loading will be applied (possibly again) to all elements in the model constructed up to that point.

To omit self-weight for elements in Staged Construction Analysis, there are two options. You may omit the self-weight load case in the construction step in which the elements are constructed. Alternatively, you may construct the elements with the stiffness-only option. In the Construction Stages Explorer [in LARSA 4D User’s Manual] right-click the structure group of elements and choose Stiffness Only. With this option, self-weight will be skipped even if a self-weight load case is present.

When a dynamic analysis is performed as an analysis scenario (page 153) in Staged Construction Analysis, element mass is computed automatically for elements constructed normally. Elements constructed with the stiffness-only option will not have mass. (If an element is constructed normally, that is with stiffness and weight, but a self-weight load case is not applied, mass will still be included.)

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
Joint Loads

Joint loads can be applied at the joints of the structure. The applied forces are specified in the displacement coordinate system of the joint, which is by default the Global Coordinate System.

Multiple joint loads can be applied to the same joint in a load case.

Joint loads are static loads and are applied in static load cases (page 107).

**Joint**

The ID of the joint to which this joint load will be applied.

**Force X/Y/Z**

The applied translational force in the joint's displacement coordinate system.

**Moment X/Y/Z**

The applied moment in the joint's displacement coordinate system.

If the joint is assigned a cylindrical coordinate system, the X, Y, and Z components of the load correspond to the R, Theta, and Z axes of the UCS. Specifying an x-component results in a load directed radially from the center of the UCS, with positive loads directed away from the origin. Specifying a y-component results in a tangentially-directed load, with positive loads directed counter-clockwise about the origin.

If the joint is assigned a spherical coordinate system, the X, Y, and Z components of the load correspond to the R, Phi, and Theta axes of the UCS. Specifying an x-component results in a load directed radially from the center of the UCS, with positive loads directed away from the origin. Specifying a y- or z-component results in a tangentially-directed load in the phi and theta directions, respectively.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- Joints on page 55.
Support Displacements

Support displacement loads can be applied at the joints of the structure. The applied displacements are specified in the displacement coordinate system of the joint, which is by default the Global Coordinate System.

Support displacement loads can only be applied to joints in degrees of freedom that are fixed at the joint. If a support displacement load in any load case is applied to a joint in a direction in which the joint is not restrained, the joint will become restrained in that direction in all load cases in the project.

Support displacement loads are static loads and are applied in static load cases. A joint can have only one support displacement in a load case.

Support displacements are also referred to as support settlements and joint displacement loads.

**Joint**

The ID of the joint to which this joint load will be applied.

**Translation X/Y/Z**

The translational displacement in the joint's displacement coordinate system.

**Rotation X/Y/Z**

The rotational displacement in the joint's displacement coordinate system.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*. 
Member Loads

Member loads encompass a wide range of loads that can be applied to members, including concentrated forces and moments and distributed forces and moments.

Member loads can be specified in the local coordinate directions of the member or in the Global Coordinate System. Distributed loads can also be specified as projected loads in the Global Coordinate System directions.

These loads are positive when acting on the member in the positive direction of the specified reference axis. The loads are always assumed to act on the centroid axis of the member.

Because truss and cable elements have only axial stiffness, laterally applied loads are transferred to the joints the member is connected to, unlike in beam elements for which the fixed-end-force concept is used.

Multiple member loads may be applied to the same member in a load case.

**Member**
- The ID of the member to which the load will be applied.

**Type**
- The type of the load.
  - **Point Force/Point Moment** - A force or moment applied at a single point on the member.
  - **Uniform Force/Moment** - A uniform force or moment applied along a part or all of the length of the member.
  - **Trapezoidal Force/Moment** - A force or moment applied along a part of the length of the member with varying magnitude at the beginning and at the end of the applied load.
  - **Pre-Tension** - This load type is usually used for cable elements in shape finding or for construction analysis of cable structures. The pre-tension load is the initial cable force at the beginning of the analysis step, but the cable force may be reduced or increased due to applied loads or other structural changes as they are applied.
  - **Post-Tension** - This load type is usually used for cable elements in shape finding or for construction analysis of cable structures. The post-tension load will force the rest of the structure to adjust in order to keep the cable force constant at the specified magnitude, simulating the jacking operation at a constant force on a cable.
  - **Jacking Elongation** - Simulates the jacking operation in cable tensioning. The jacking elongation load specifies the change in the length of the member, which is used only in nonlinear analyses.

The pre- and post-tension load types are only used in a nonlinear analysis, both static and Staged Construction Analysis.

**Direction**
- Loads can be applied in the local or the global coordinate system. Positive magnitude loads are in the direction of the positive axis selected. Specify negative magnitudes to apply the load in the negative axis direction.
  - **Local X/Y/Z.** The load acts in the direction of the local axis of the member.
  - **Global X/Y/Z.** The load acts in the direction of the global axis.
  - **Projection on Global X/Y/Z.** Applies to uniform or partially distributed loads. The direction of the load is in the direction of the global axis, and the magnitude of the load is measured per projected length. The projected length is in the plane perpendicular to the direction of loading.
The direction is not applicable for pre/post-tension loads or jacking elongation, which are applied axially.

**Magnitude at Start**
For point and uniform loads, the magnitude of the load. For trapezoidal loads, the magnitude at the beginning of the load application.

**Magnitude at End**
For trapezoidal loads, the magnitude at the end of the load application. Not applicable for other load types.

**Start Position**
For point loads and pre-/post-tension loads, the location of the load. For distributed loads, the location of the start of the load application. Specified relative to the member length (e.g. 0.0 is the start of the member at the I-Joint and 1.0 is the end of the member at the J-Joint), or in absolute length units, measured along the length of the member. If the member has an end offset, the position is not measured from the joints but instead from the offsetted end positions.

**End Position**
For distributed loads, the location of the end of the load application, specified in the same way as the Start Position, relative to the start of the member. Not applicable for other load types.

**Shortening**
The shortening length. Only applicable for Jacking Elongation loads.

### Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Member Loads spreadsheet, the commands are available in the Member Loads menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

The additional commands are as follows:

**Enter Absolute Positions**
Choosing this option sets the start and end position fields to use absolute length units. 0.0 represents the start of the member, and the length of the member is used to give the position at the end of the member.

**Enter Relative Positions**
Choosing this option sets the start and end position fields to use relative units. 0.0 represents the start of the member, and 1.0 represents the end of the member.

For More Information, please refer to the following documentation.

- Members on page 57.
- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
Member Thermal Loads

The loading due to temperature change in a member can be specified using Member Thermal Loads.

With uniform thermal loading, a constant temperature change is applied to the entire member cross-section. Uniform thermal loading normally produces axial strain only. Linear and nonlinear thermal gradients create a temperature change that is uniform along the length of the member but that varies either from section top to bottom or from section left to section right. All thermal loads are constant along the length of the member.

When applied to composite sections, thermal loading will use the material assigned to each shape in the section if a material is assigned to section shapes. If either the material modulus of elasticity or coefficient of thermal expansion differ between shapes in the section, rotational strain may result and self-equilibrating stresses (also known as eigenstresses) will be accumulated and included in member stress results (page 273). Nonlinear thermal gradients may also create self-equilibrating stresses that will be included in member stress results. (Prior to LARSA 4D version 8, the material assigned to the member was assumed constant throughout the cross-section and, therefore, no self-equilibrating stresses were possible. Nonlinear thermal gradients became available in LARSA 4D version 8.)

For linear and nonlinear thermal gradients, temperature change is specified at (or, for nonlinear thermal gradients, relative to) the + y/z and - y/z edges of the member cross-section. The location of the + y/z and - y/z edges are determined by the maximum and minimum y and z coordinates of the points that define the section geometry of the member (either from Section Composer or section dimensions (page 23)). If a member has no section geometry (i.e. its section has Unspecified shape, see Section Dimensions (page 23)), the first and third section stress recovery points (page 23) determine the section extents instead. (Prior to LARSA 4D version 8, the first and third stress recovery points were always used to determine the section positive and negative extents, respectively.)

Nonlinear thermal gradients must be used with sections that have geometry either from Section Composer or section dimensions (page 23) because the results depend on the distribution of mass in the cross-section.

The coefficient of thermal expansion specified in the material data of the element (or shapes) must have a nonzero value, otherwise there will be no thermal effects for the element. Only uniform thermal loads should be applied to truss and cable elements.

Multiple member thermal loads can be applied to a member in a load case.

Load Input

Member
The ID of the member to which the load will be applied.

Type
The type of the load: Uniform, Linear, or Nonlinear.

Variation Direction
For linear or nonlinear gradient thermal loading, specifies whether the variation changes along the local-y or local-z axis of the member.

Temperature Change at Top
For uniform loads, specifies the change in temperature. For linearly varying loads, specifies the temperature change at the top edge (+y) or right edge (+z) of the member. Not applicable for nonlinear gradients.
**Temperature Change at Bottom**
For linearly varying loads, specifies the temperature change at the bottom edge (-y) or left edge (-z) of the member. Not applicable for uniform loads or nonlinear gradients.

**Nonlinear Temperature Curve**
Specifies the nonlinear temperature curve to apply. Create a curve in the Input Data → Properties → Temperature Curve spreadsheet first.

**Nonlinear Thermal Gradient Curves**
Nonlinear thermal gradient curves are entered in a spreadsheet located at Input Data → Properties → Temperature Curve. Add a row to this spreadsheet and name the curve. Then right-click the spreadsheet and choose Edit Curve.

Add rows to this spreadsheet to specify location and temperature change values on the curve.

**Temperature Change**
Each temperature value is specified in load-temperature units (see Model Units in LARSA 4D User’s Manual). It is the temperature change at a location in the cross-section.

**Location Reference**
The location of each temperature coordinate is specified in one of four ways: + Edge indicates the location is measured from the extreme +y or +z fiber (depending on the Variation Direction specified in the load spreadsheet). - Edge indicates the location is measured from the extreme -y or -z fiber. In both cases positive values are toward the centroid. Rel. indicates that the location is specified relative to the member depth or width in the range from 0.0 to 1.0. Abs. indicates the location is specified in section length units. For instance, using the + Edge, Rel. option, the value 0.0 indicates the top or right edge of the section and 0.25 specifies a location 1/4th the way into the section from the top or right. Or, using the - Edge, Abs. option, the value 0.0 indicates the bottom or left edge of the section and 5.8 specifies a location 5.8 section units into the section measured from the left or bottom.

For More Information, please refer to the following documentation.
• Sections on page 23.
• Members on page 57.
• For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
Plate Loads

Uniformly distributed pressure loads, concentrated point forces, thermal loads, and strain loads can be applied to plates.

To apply a uniform pressure load, a point force load, or a strain load, use the Type, Direction, and Load fields. Keep the Uniform Temperature Change and Temperature Gradient fields as zero.

To apply a thermal load, use the Uniform Temperature Change and Temperature Gradient fields. The Load field should be kept as zero so that the Type and Direction fields are not used. Alternatively, a thermal load can be combined with another type of load by using all five fields.

Temperature loads on plates can be specified as a uniform change or as a linear temperature gradient through the thickness of the element. The temperature effects are constant over the lateral directions of the element. The coefficient of thermal expansion specified in the material data of the element must have a nonzero value, otherwise there will be no thermal effects. Uniform temperature loading causes in-plane (lateral) expansion. Temperature gradient loading causes out-of-plane bending only.

Multiple plate loads can be applied to a plate in a load case.

Plate Load Fields

Plate

The ID of the plate to which the load will be applied.

Type

The type of the load.

- **Uniform Force** - A uniform force applied on the surface of the element.
- **Point Force** - A force applied at a single point on the element.
- **Strain** - A strain added to the element.

Direction

Uniformly distributed loads can be specified in the local coordinate directions of the plate or in the Global Coordinate System directions, or as projected loads in the Global Coordinate System directions. Point loads can be specified in the local coordinate directions of the plate or in the Global Coordinate System directions. Strain loads can be applied in the local x or y directions only. Positive magnitude uniform and point force loads are in the direction of the positive axis selected and for strain loads are tensile.

- **Local X/Y/Z.** The load acts in the direction of the local axis of the plate.
- **Global X/Y/Z.** The load acts in the direction of the global axis.
- **Projection on Global X/Y/Z.** Applies to uniform loads. The direction of the load is in the direction of the global axis, and the magnitude of the load is measured per projected area. The projected area is in the plane perpendicular to the direction of loading.

Load

The magnitude of the pressure, force, or strain. For uniform loads, the load is constant throughout the plate. Not used for thermal loads.
Relative X Location (x/XL)
For point loads, the relative distance of the load with respect to the I-J or K-L sides of the element, as explained below.

Relative Y Location (x/XL)
For point loads, the relative distance of the load with respect to the J-K side of the element for quadrilateral plates and with respect to the local y-axis for triangular plates, as explained below.

Uniform Temperature Change
The uniform temperature change through the thickness of the element. Positive values are an increase in temperature by that many degrees.

Temperature Gradient
The temperature change per unit of length measured perpendicular to the plate, i.e. through the thickness of the element. Positive values mean the top (+z side) is subject to a temperature increase while the bottom (-z side) is subject to a temperature decrease. The middle of the plate (z=0) has no temperature change. For example, if a plate 3 inches thick is given a 1 degree/inch temperature gradient, it experiences a 1.5 degree temperature increase at its top surface, no temperature change through its midplane, and a -1.5 degree temperature change at its bottom surface.

Point Load Coordinates
The locations of point force loads on plates are specified in a special coordinate system which is neither the global nor plate local coordinates. The choice of coordinate system is intended to simplify the specification of the location so that the x- and y-coordinates are both in the range [0, 1].

For quadrilateral plates, the load coordinates of the four corners of the plate from I to L are (0,0), (1,0), (1,1), (0,1). For rectangular plates, the load coordinates are simply scaled plate local coordinates. For irregularly shaped plates, the load coordinates are essentially warped to fit in the quadrilateral shape. See the following figure:

Load coordinate specification for quadrilateral plates

(I-, J-, K-, and L-joints refer to the first, second, third, and fourth joints, respectively, that define the plate geometry in the plates spreadsheet (page 71).)

The load location specification for triangular plates is on a scaled local coordinate system, as with perfectly rectangular plates. The load coordinates of the three corners of the plate from I to K are (0,0), (1,0), and (q,1), for some q. For triangular plates, unlike quadrilateral plates, not every coordinate with components in the range [0, 1] will actually fall within the plate area. The coordinate grid for load specification on triangular plates is shown below:
Load coordinate specification for triangular plates

For More Information, please refer to the following documentation.

- Plates on page 71.
- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*. 
Moving Loads

Moving loads are added into moving load cases for the Moving Load Analysis (page 203). These loads specify how to generate the result cases for a vehicle moving on a user-defined path (lane) on a structure.

The analysis requires a database file containing the load patterns [see “Connecting Databases” in LARSA 4D User’s Manual], geometry definition of lanes (page 99), and load cases in which the user can specify the load pattern, the loaded lane and related data for a traveling load pattern.

A single load case with multiple moving loads simulates the simultaneous marching of the loads. Multiple load cases can be used to simulate multiple independent vehicles on the same lane or multiple lanes.

Moving load cases can be combined with static load cases, response spectra cases, and with other moving load cases. Combinations can be made within the analysis using Load Combinations (page 109) or after the analysis using Linear Result Combinations [in LARSA 4D User’s Manual].

Lane Loaded
The lane (page 99) on which the load pattern will be marching.

Load Pattern
The load pattern to march on the lane. All load patterns from connected moving load pattern databases are available to choose from. See Connecting Databases [in LARSA 4D User’s Manual].

Applied Load Direction
The direction of the load application. The direction can be in the local coordinate systems of the members the lane passes through, or in global directions. Positive magnitude loads are in the direction of the positive axis selected.

Load Factor
A factor to scale the load pattern. The default is 1.0.

Direction of Travel
The load pattern can march forward on the lane or backward. In Static Moving Load Analysis, both means first forward then backward.

Position Increment
The distance between each successive position of the load pattern as it marches along the lane. Smaller increments will generate more result cases but may be more informative and accurate. Static Moving Load Analysis only.

Vehicle Speed
The speed of the vehicle as it moves along the lane. Dynamic Rolling Stock Analysis only.

Start Position
The starting position of the load, measured in length units along the lane from the start of the lane. 0.0 is the start of the lane.

For More Information, please refer to the following documentation.

• For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
• Lanes on page 99.
• Standard Vehicles on page 225.
• Moving Load Analysis on page 203.
• Connecting Databases in LARSA 4D User’s Manual.
• Influence Loads on page 127.
Influence Loads

Influence loads are added into moving load cases for the Influence Line & Surface Analysis (page 205). These loads specify how to generate the result cases for a unit load moving on a user-defined path or surface.

The analysis requires the geometry definition of lanes (page 99) and load cases in which the loaded lane and related data for a traveling load pattern are specified.

Generally speaking, a separate moving load case is required for each lane or surface to be considered. In each load case, a single influence load is applied to that lane.

In some circumstance, it may be appropriate to apply two or more simultaneous influence loads in a single moving load case. To simulate the distribution of vehicle axle loads onto multiple girders in an influence line analysis, you may place multiple influence loads within the same moving load case, each influence load applied to a lane on a different girder, so long as the factors sum to 1.0 and the lanes are of the same length. In an influence line or influence surface analysis, you may combine dead load (in global -Z), centrifugal (local +y), or braking (local +x) forces in one case by including multiple influence loads within the same moving load case, each in a different direction.

Lane Loaded

The lane (page 99) on which the unit load will be marching. If the lane is defined with reference to members, it is a linear path for an influence line. If the lane is defined with reference to plates, it is a surface for an influence surface analysis and width should be specified for the lane.

Applied Load Direction

The direction of the load application. The direction can be in the local coordinate systems of the members or plates the lane passes through or in global directions. Local directions are recommended. When the lane includes offsets from the member centroid (such as for live load centrifugal and braking forces conditions), local directions must be used. Moments due to the offset will not be created when the applied load direction is in a global direction. Positive magnitude loads are in the direction of the positive axis selected.

Load Factor

A factor to scale the unit load. The default is 1.0.

Forward Increment

The distance between each successive position of the load pattern as it marches along the lane. In an influence surface load case with simultaneous loads, the forward increment of the first influence load will be applied to all influence loads in the load case.

Transverse Increment

Used for influence surfaces only. The transverse distance between lateral strips used to approximate the surface. The transverse increment must divide the lane width evenly; if it does not, LARSA will round-up the increment. A transverse increment of zero instructs LARSA to choose a sensible increment based on the forward increment. In an influence surface load case with simultaneous loads, the transverse increment of the first influence load will be applied to all influence loads in the load case.

Smaller increments will generate more result cases but may be more informative and accurate.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in LARSA 4D User’s Manual.
• Lanes on page 99.
• Standard Vehicles on page 225.
• Moving Load Analysis on page 203.
• Influence Line & Surface Analysis on page 205.
Time History Loads

Time history loads include Applied Excitation Functions and Initial Conditions. They are applied in time history load cases for the Linear Time History Analysis (page 193) and Nonlinear Time History Analysis (page 195).

Excitation Functions

There are three types of excitation functions:

1. Uniform base excitations, which must be force or acceleration functions, are applied to all free joints in the model. Fixed joints represent the ground.

2. Joint-localized excitations are applied in a single direction at a joint. Displacement excitation functions are applied to joints fixed in the direction of the excitation function, while force and acceleration excitation functions are applied to joints free to move in the direction of the excitation function. Multiple excitation functions can be simultaneously applied to achieve an excitation in more than one direction at the same joint, or at more than one joint.

3. A static load case containing joint, member, and plate loads can be chosen in conjunction with a “factor” excitation function. The “factor” function is unitless and is applied as a scale factor to the loads in the static load case in each time step. Joint loads, member force, moment, and self-weight loads, and plate force and self-weight loads can be used. An “inconsistent load vector” method is used, meaning element loads are transferred to joints and element-internal forces (such as Member Sectional Forces (page 271) and Plate Forces on Center (page 287)) will not account for these loads.

When an acceleration excitation curve is chosen, whether for a uniform base acceleration or as a joint-localized excitation, an equivalent force is applied at applicable joints computed as the acceleration multiplied by the mass at the joint. The mass is the sum of the mass due to elements connected to the joint (computed in the same way as self-weight (page 111)), mass specified on a mass element (page 85) at the joint (in the applied direction of the excitation function), and equivalent mass specified in load cases (page 107) with the Is Dynamic Mass option turned on.

Before entering the load data for a time history excitation, a database file containing the time history excitation curve must be connected to the project. See Connecting Databases [in LARSA 4D User's Manual].

The time history excitation loads spreadsheet has the following fields:

**Time History Curve**
- The time history excitation curve to be applied, selected from all of the time history curves in databases connected to the project. A displacement, force, acceleration, or factor function can be chosen.

**Joint ID**
- The ID of the joint to which a joint-localized excitation will be applied. For a uniform base excitation, enter zero. This field is not applicable for static load case excitations.

**Applied Direction**
- The direction of the applied displacement, force, or acceleration: translation x, y, or z or rotation x, y, or z. The excitation function is applied in joint displacement coordinate system directions, which is by default the Global Coordinate System. This field is used with uniform base excitations and joint-localized excitations but not factored static load case excitations.

**Static Load Case**
When a “factor” excitation function is used as the time history curve, any Static Load Case (page 107) can be chosen to apply factored loads in each time step.

Initial Conditions

Initial conditions are for initial nonzero displacements, velocities, and accelerations. When no initial conditions are specified, the structure is assumed at rest initially.

The following data is used for initial conditions:

**Joint**
- The ID of the joint for which this initial condition applies.

**Displacement**
- The initial displacement of the joint.

**Velocity**
- The initial velocity of the joint.

**Acceleration**
- The initial acceleration of the joint.

**Direction**
- The direction of the specified displacement, velocity, and acceleration. May be translational or rotational in x, y, or z. The directions are in the joint's displacement coordinate system.

Additional Commands

Some additional related commands are available. These commands are used to modify input data or to access additional attributes of the data referenced above. The commands can generally be accessed in one of several ways.

In the Time History Excitations spreadsheet, the commands are available in the Time History Excitations menu, or by right-clicking the spreadsheet. Some commands can be applied to more than one row in the spreadsheet at once by selecting multiple rows before activating the command.

The additional commands are as follows:

**Add Excitation for Selected Joints**

This command adds excitation records to the active load case for every selected joint.

For More Information, please refer to the following documentation.

- For help on using spreadsheets, see Using the Model Spreadsheets in *LARSA 4D User’s Manual*.
- Linear Time History Analysis on page 193.
- Nonlinear Time History Analysis on page 195.
- Connecting Databases in *LARSA 4D User’s Manual*.
Construction Activities

In the Staged Construction Analysis, changes to the structure are modeled as construction activities. Data input for construction activities is described in this section.

Construction activities are used in the Staged Construction Analysis (page 229) and are added in the Construction Stages Explorer [in LARSA 4D User's Manual], or with the Construction Stage Editor [see “Construction Stage Editor” in LARSA 4D User's Manual] in the Input Data menu.

For More Information, please refer to the following documentation.

• Staged Construction Analysis on page 229.
• Construction Stages Explorer in LARSA 4D User’s Manual.

Construct and Deconstruct Activities 133
  Deconstruction 133
  Self Weight and Mass 133
  Segmental Construction Methods 134

Load Activities 137
  Creating Load Activities 137
  Using Load Activities 137
  Self-Weight Load Cases 137

Support and Hoist Activities 139
  Activity Fields 139

DOF Constraint Change Activities 141
  Activity Fields 141

Tendon Stressing and Slackening Activities 143
  Activity Fields 143

Displacement Initializations 145
  What It Does 146
  Activity Fields 146

Cast Concrete Activities 147

Composite Section Construction Activities 149

Analysis Scenarios 153
Construct and Deconstruct Activities

The construct and deconstruct staged construction activities add and remove structural elements from the model.

Structural elements are at any point constructed or unconstructed. Constructed elements are used in the analysis of the structure, while unconstructed elements are ignored. Elements start off unconstructed until they are explicitly constructed in a step.

To construct or deconstruct a part of the model, form a structure group [see “Structure Groups Explorer” in LARSA 4D User’s Manual] and then drag that group from the Structure Groups Explorer into the Construction Stages Explorer [in LARSA 4D User’s Manual]. The elements in that group will be constructed. To deconstruct those elements, right-click the stage, choose Properties, and mark the Deconstruct option.

Elements may be constructed in locations relative to the deformed state of the structure. Segmental construction methods (described below), Support and Hoist Activities (page 139), and Displacement Initializations (page 145) are used to specify such behavior. Normally, elements are constructed in the position that they are initially defined in.

Deconstruction

Unless otherwise marked, structure groups included in a stage are constructed in that stage.

To deconstruct elements, right-click the step in which the elements are to be deconstructed, choose Properties, and then choose the Deconstruct option. All structure groups included in the step are deconstructed.

When elements are deconstructed, the internal forces in the elements are removed, resulting in unbalanced forces at the joints that connected the deconstructed elements to the model. These unbalanced forces are resolved during the construction step. Any automatic self-weight applied to deconstructed elements is also removed. See Load Class Tracking (page 247).

Self Weight and Mass

Self-weight is not automatically included in Staged Construction Analysis. To include self-weight for newly constructed elements, add a load activity (page 137) for a self-weight (page 111) load case in the same step the construction activity is placed in. Self-weight will be applied to all and only the elements constructed in that stage. Self-weight will not be applied again on elements already constructed. However, if an element is included a second time in a group included in a stage, and self-weight is applied in both stages, self-weight loads may be applied twice to that element.

If a self-weight load case is placed in a construction step without a construction activity, self-weight loading will be applied (possibly again) to all elements in the model constructed up to that point.

When a dynamic analysis is performed as an analysis scenario (page 153) in Staged Construction Analysis, element mass is computed automatically for elements constructed normally. Elements constructed with the stiffness-only option will not have mass. (If an element is constructed normally, that is with stiffness and weight, but a self-weight load case is not applied, mass will still be included.)
Segmental Construction Methods

In Staged Construction Analysis, when joints are activated they normally enter at the location that was initially given to them. Joints sometimes need to become active in a location relative to the deformed structure of the model, rather than in an exact position known ahead of time. The segmental construction method option of construction steps and joint displacement initialization activities specify how to place a joint relative to the deformed location of other joints.

The segmental construction method option of construction steps specifies that the locations of all joints becoming active in the step will be updated according to a segmental construction rule. The segmental construction method can be chosen by right-clicking a step, choosing Properties, and choosing among the options described below. Segmental construction methods have been designed to work with line element models of segmentally constructed structures and are not yet intended to be used when plate elements are used to model the segmental parts. Displacement Initializations (page 145) should be used in plate models.

The example below shows why a special segmental construction method might be necessary. It is a simple segmental bridge built in stages.

In the first stage, only the left segment is activated.

After the application of loads, the segment deforms. The center node is displaced, but the right node is left unchanged because inactive joints are ignored in the analysis.

As a result, when the next segment is constructed, the segments form an angle, even though there was no angle when the segments were initially modeled. This may not be the desired model.

Two other behaviors might have been desired:

**Hinged Cast**

The next segment should be constructed by shifting the end node down to match the translational displacement of the center node. i.e. The segment was drawn horizontal, and it should remain horizontal.

**Matched Cast**

The next segment should be constructed by shifting the end node down to match the angular displacement of the center node. i.e. The segment was drawn parallel to the last, and it should remain parallel.

These are shown in the figure below.
Hinged Cast versus Matched Cast

Displacement Initializations (page 145) are also used to construct elements in locations relative to the deformed position of other elements.

For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- Geometry on page 53.
- Displacement Initializations on page 145.
Load Activities

Load activities in the Staged Construction Analysis apply and remove loading from the model.

A static load case (page 107) is applied to the structure in a load activity. Loads are applied cumulatively; that is, a load applied in one stage will automatically continue to be applied in all future stages.

Creating Load Activities

To apply loads in a construction step, form a load case [see “Load Cases Explorer” in LARSA 4D User’s Manual] and then drag that group from the Load Cases Explorer into the Construction Stages Explorer [in LARSA 4D User’s Manual].

A loading factor can be applied to load cases included in a stage. In the Construction Stages Explorer [in LARSA 4D User’s Manual], click the load case as it appears in a construction step. At the bottom of the explorer, enter a factor in the Loading Factor field, and then click the checkmark to confirm the change.

Using Load Activities

To "turn off" a load, there are two methods. The Temporary Loads option of steps can be turned on in step properties, by right-clicking a step, choosing Properties, and then checking the Auto-remove loads prior to next step option. All load cases included in the step will be removed in the following construction step. You can also apply the load again with a -1 load factor.

Create a load case with a Self Weight (page 111) to include self-weight in a Staged Construction Analysis. However, see the notes on self-weight in Construct and Deconstruct Activities (page 133).

The pre-/post-tension and jacking elongation member load (page 117) types are of particular interest in the Staged Construction Analysis.

Self-Weight Load Cases

Self-weight load cases are treated specially in Staged Construction Analysis. See the note in Construct and Deconstruct Activities (page 133).

For More Information, please refer to the following documentation.

• Staged Construction Analysis on page 229.
• Loads on page 105.
• Construction Stages Explorer in LARSA 4D User’s Manual.
• Load Cases Explorer in LARSA 4D User’s Manual.
Support and Hoist Activities

Support activities are used to change support conditions in a model during construction. Hoist activities are used to hoist a joint in a direction by the same but opposite amount it deformed.

A support activity changes the support conditions of a joint in the model. A hoist activity applies a displacement to a joint, in one of its six directions of displacement, with a magnitude equal and opposite to the total displacement of the joint since initial conditions in that direction, and then restrains the joint in that direction.

When a support or hoist activity is applied to a joint, the new support conditions for all six degrees of freedom at the joint must be specified. In each of the six directions, which as with joint restraints (page 55) are in global directions unless a joint displacement coordinate system is set on the joint, the joint must be specified as free, fixed, or hoisted.

Do not leave a direction as free to mean that the joint's support conditions have not changed. This will remove a support in that direction if one was previously specified.

When hoist is chosen in a given direction, the joint is forced to deform back to its initial coordinate in the direction and then is supported in that direction.

When a support is removed, by changing a direction to free if the joint was already supported in that direction, the reactions at that joint are applied back into the model. See Load Class Tracking (page 247).

Activity Fields

Joint
The ID of the joint whose support conditions are changing.

X/Y/Z-Translation and X/Y/Z-Rotation
The new support condition of the joint in each direction: free, fixed, or hoist. These are described above.

For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- Joints on page 55.
DOF Constraint Change Activities

DOF Constraint activities in the Staged Construction Analysis apply and remove DOF Constraints between degrees of freedom of two joints.

Like the DOF Constraints (page 87) specified for other analysis types and for initial conditions in the Staged Construction Analysis (page 229), DOF Constraint activities are used to manage equal-displacement constrained behavior between two degrees of freedom in the model. A DOF constraint activity adds or removes such constraints during a step of construction.

When a DOF constraint activity is applied to a joint, the conditions of each of that joint's six degrees of freedom must be specified. A condition for a direction can be left as (none) when the joint is free to move in that direction, or the ID number of a so-called "master" joint can be specified to ensure that displacements in that direction are equal between the master and the dependent joint.

Do not leave a condition field (master joint ID) as (none) to mean that the constraints do not change in the step. Leaving a field as (none) means that the joint becomes free in that direction in the step.

If a DOF constraint is removed through an activity, the new unbalanced forces from the elements on the joint are added back into the structure. See Load Class Tracking (page 247).

Activity Fields

**Dependent Joint**

The ID of a joint to constrain or unconstrain.

**Master Joint for Translation X/Y/Z and Rotation X/Y/Z**

If a Joint ID is specified, then the displacement in this direction of the dependent joint will be equal to the displacement in the same direction as this master joint, replacing any previously set master joint in this direction. If no joint ID is specified, then the displacement in this direction of the dependent joint will be free (and any previously set master joint in this direction will be unset).

For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- DOF Constraints on page 87.
Tendon Stressing and Slackening Activities

Tendon stress/slacken activities apply or remove the effect of pre- and post-tensioning tendons (page 89) in the model.

The effect of Tendons (page 89) defined in the model is not applied until they are stressed in a Staged Construction Analysis (page 229) activity. A tendon stressing activity indicates that the equivalent loading representing the tendon is to be added into the model.

If a Time-Dependent Staged Construction Analysis is run and the appropriate time-dependent effects are selected, these effects on tendons will be automatically included in the results. See Time Effects on Materials (page 243).

It is also possible to slacken a tendon by choosing Slacken as the activity type.

Activity Fields

**Tendon**
- The tendon being stressed or slackened.

**Activity**
- Stress or Slacken

For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- Tendons on page 89.
- Construction Stages Explorer in *LARSA 4D User’s Manual*. 
Displacement Initializations

Displacement initialization construction activities specify that a joint's location should be updated based on the displacements of one or two other joints or a fixed offset.

When joints are constructed at different times, geometrical relationships between joints may be altered due to displacements in the model. For instance, although three joints may be colinear in initial conditions specified in the geometry spreadsheet, displacement at two of the joints before the third is constructed may alter that geometric relationship. When the third joint is constructed, the members connecting the joints may form an angle rather than a straight line. This is depicted in the figure below.

Displacement initializations are used to update the initial conditions of a joint before it is constructed based on the displacement of one or two other joints. Displacement initializations can also be used to specify an exact geometry update so that the joint coordinates in the joints spreadsheet (page 55) can be overridden in Staged Construction Analysis.

Displacement initializations cannot be used in the first construction step.

See also the segmental construction methods (page 133) for other options to update joint coordinates based on displacement during Staged Construction Analysis.
What It Does

One dependent and up to two master joints are specified. The dependent joint is the non-constructed joint whose location is to be updated. The master joint(s) are already constructed and probably have had displacement by the time of the previous construction step.

If no master joints are specified, then the dependent joint's coordinates are updated by adding in the values in the Displacement X/Y/Z columns.

If one master joint is specified, or if two master joints are specified but they are coincident in initial conditions, then the dependent joint's location is updated by adding to it the total displacement of the (first) master joint at the end of the previous construction step, plus any additional displacement entered into the Displacement X/Y/Z columns.

If two master joints are specified and they are not coincident in initial conditions, then the dependent joint's location is updated by adding to it a linear interpolation between the two master joint's total displacements at the end of the previous construction step, plus any additional displacement entered into the Displacement X/Y/Z columns. If the three joints are colinear in initial conditions (master 1--dependent--master 2), then the interpolation is simply linear along that line. For instance, if the dependent joint is mid-way between the two master joints, then it will be adjusted by an amount equal to the average of the two displacements. However, in the general case, the interpolation is linear using the ratio of the distance between the first master and the dependent joint to the distance between the two master joints.

The Direction column limits the update to only certain components of the dependent joint's location. If the Direction is all, the coordinate update is as describe above. If the Direction is y, for instance, then the x- and z-components of the computed dependent joint displacement are reset to zero.

Activity Fields

**Joint**

The ID of the joint whose location is to be updated (the dependent joint).

**Master Joint 1/2**

The zero, one, or two master joints whose displacements in the previous construction step are used to adjust the location of the dependent joint.

**Direction**

The translational degrees of freedom of the joint to update. Select all to update the joint normally. Or, for example, specify just x to update the joint's x location only, leaving its y and z coordinates unchanged.

**Displacement X/Y/Z**

Additional displacement values to apply to the dependent joint. Displacements in directions that are not specified in the Direction column are ignored.

For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- Construction Stages Explorer in *LARSA 4D User’s Manual*.
- Construct and Deconstruct Activities on page 133.
- Joints on page 55.
Cast Concrete Activities

The Cast Concrete staged construction activity is used to set the casting day of member elements and plate elements for use in Time-Dependent Staged Construction Analysis.

The activity sets the casting day of the element or shape to the day of construction set on the stage in which the activity occurs. The member and plate elements are chosen using a Structure Group.

For a member whose section is defined with a single shape or for a plate element, the casting day of the element can be set by using the Cast Concrete activity within stages of construction, similar to entering the Casting Day information into members or plates spreadsheet.

Section Composed Of A Single Shape

For a member whose section is defined with multiple shapes, i.e. a built-up section, in order to assign a different casting day to each individual shape, the Cast Concrete activity must be used instead of the members spreadsheet. In that case, the Part to Cast should be set to the name of the shape to set the casting day of. The Part to Cast column will only be available if all of the elements in the chosen Structure Group are members with the same section property assigned.

Built-up Section

The Cast Concrete activity is available starting with LARSA 4D version 8.0. In version 8.00.9010 and later, it also works for plate elements in addition to member elements.

For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- Members on page 57.
• Construction Stages Explorer in LARSA 4D User’s Manual.
Composite Section Construction Activities

Composite Construction activities are used to change the composite construction sequence state [see “Composite
Construction Sections” in LARSA Section Composer Manual] of member elements in order to add individual shapes
within built-up cross sections into the model.

For sections assembled part-by-part during Staged Construction Analysis, the construction sequence states defined in
the Section Composer are activated using construction activities. A Composite Construction Activity is used each time
the section changes. These activities are assigned within construction steps to structure groups containing members
with the same section.

Consider a member section with its composite construction sequence states defined in the Section Composer as in
the below figure:

Before using the composite construction activities within stages of construction, the member(s) whose sections will
change during the Staged Construction Analysis are put into a structure group, here named “Composite.”

A Composite Construction Activity is created by right-clicking the step in the Construction Stages Explorer at the point
where the activity is to occur and then by choosing Composite Activities. When the composite activity spreadsheet
for that step is shown, a new row is created for the structure group “Composite,” and the new construction state for
the section is chosen which will take effect beginning with this step.
Composite Activities

Instead of defining State C in Section Composer, “All Shapes Active” can be used instead in the composite activity spreadsheet, under the **New State** column.

All Shapes Active

The composite construction activity is available starting with LARSA 4D version 8.0.
For More Information, please refer to the following documentation.

- Staged Construction Analysis on page 229.
- Members on page 57.
- LARSA Section Composer in *LARSA Section Composer Manual*.
- Construction Stages Explorer in *LARSA 4D User’s Manual*. 

151
Analysis Scenarios

Analysis Scenarios are analyses performed during a Staged Construction Analysis (page 229), such as a live load analysis or pushover, based on the deformed state of the structure at any time.

Any time of analysis can be performed in a construction step. To change the analysis type, right-click the construction step and choose Properties. Then at the bottom right of the window, choose the analysis type.

You will also need to add a load case into the construction step as a load activity (page 137) that specifies the parameters for the scenario. For instance, for a live load (moving load or influence) scenario, include a load case with a moving load or influence set-up load.

For More Information, please refer to the following documentation.

• Staged Construction Analysis on page 229.
• Construction Stages Explorer in LARSA 4D User’s Manual.
• Load Cases Explorer in LARSA 4D User’s Manual.
Each of the analysis types available in LARSA 4D are described in the sections below.

<table>
<thead>
<tr>
<th>Static Analysis</th>
<th>159</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear Static Analysis</td>
<td>161</td>
</tr>
<tr>
<td>Structure Model for Linear Static Analysis</td>
<td>161</td>
</tr>
<tr>
<td>Analysis Results</td>
<td>161</td>
</tr>
<tr>
<td>Assumptions in a Linear Static Analysis</td>
<td>162</td>
</tr>
<tr>
<td>P-Delta Analysis</td>
<td>163</td>
</tr>
<tr>
<td>Using Load Combinations for the P-Delta</td>
<td>163</td>
</tr>
<tr>
<td>Caveats</td>
<td>163</td>
</tr>
<tr>
<td>Method</td>
<td>163</td>
</tr>
<tr>
<td>Convergence Criteria</td>
<td>163</td>
</tr>
<tr>
<td>Nonlinear Static Analysis</td>
<td>165</td>
</tr>
<tr>
<td>Why a Nonlinear Analysis</td>
<td>165</td>
</tr>
<tr>
<td>Applications of the Nonlinear Analysis</td>
<td>166</td>
</tr>
<tr>
<td>Equations of Equilibrium and Solution</td>
<td>166</td>
</tr>
<tr>
<td>Caveats</td>
<td>166</td>
</tr>
<tr>
<td>Incremental Loading</td>
<td>167</td>
</tr>
<tr>
<td>Convergence Criteria</td>
<td>167</td>
</tr>
<tr>
<td>Displacement Convergence and Unbalanced Forces</td>
<td>167</td>
</tr>
<tr>
<td>Load Case Data for Nonlinear Static Analysis</td>
<td>168</td>
</tr>
<tr>
<td>Nonlinear Buckling Analysis</td>
<td>169</td>
</tr>
<tr>
<td>Advantages of a Nonlinear Analysis</td>
<td>169</td>
</tr>
<tr>
<td>Using the Nonlinear Buckling Analysis</td>
<td>169</td>
</tr>
<tr>
<td>Nonlinear Pushover Analysis</td>
<td>171</td>
</tr>
<tr>
<td>Nonlinear Static Analysis Options</td>
<td>173</td>
</tr>
<tr>
<td>Incremental Nonlinear and Nonlinear Buckling Analysis Options</td>
<td>175</td>
</tr>
<tr>
<td>Nonlinear Pushover Analysis Options</td>
<td>177</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Eigenvalue Analysis</th>
<th>179</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eigenvalue and Stressed Eigenvalue Analysis</td>
<td>181</td>
</tr>
<tr>
<td>Unstressed (Standard) Eigenvalue Analysis</td>
<td>181</td>
</tr>
<tr>
<td>Stressed Eigenvalue Analysis</td>
<td>181</td>
</tr>
<tr>
<td>Response Spectra Analysis</td>
<td>183</td>
</tr>
<tr>
<td>Response Spectra Curve</td>
<td>183</td>
</tr>
<tr>
<td>Modal Damping</td>
<td>183</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Modal Combination</td>
<td>184</td>
</tr>
<tr>
<td>Spatial Combination</td>
<td>184</td>
</tr>
<tr>
<td>Ground Motion Directions</td>
<td>184</td>
</tr>
<tr>
<td>Response Spectra Load Cases</td>
<td>185</td>
</tr>
<tr>
<td>Caveat About Sign</td>
<td>186</td>
</tr>
<tr>
<td>Analysis Options for Response Spectra</td>
<td>186</td>
</tr>
<tr>
<td>Eigenvalue Analysis Options</td>
<td>187</td>
</tr>
<tr>
<td>Stressed Eigenvalue Analysis Options</td>
<td>189</td>
</tr>
<tr>
<td>Time History Analysis</td>
<td>191</td>
</tr>
<tr>
<td>Linear Time History Analysis</td>
<td>193</td>
</tr>
<tr>
<td>Nonlinear Time History Analysis</td>
<td>195</td>
</tr>
<tr>
<td>Overview</td>
<td>195</td>
</tr>
<tr>
<td>Newmark-Beta with Newton-Raphson</td>
<td>195</td>
</tr>
<tr>
<td>Sparse Solver Technology</td>
<td>195</td>
</tr>
<tr>
<td>Nonlinear Time History Data</td>
<td>195</td>
</tr>
<tr>
<td>Linear Time History Analysis Options</td>
<td>197</td>
</tr>
<tr>
<td>Nonlinear Time-History Analysis Options</td>
<td>199</td>
</tr>
<tr>
<td>Geometric Nonlinearity</td>
<td>199</td>
</tr>
<tr>
<td>Live Load Analysis</td>
<td>201</td>
</tr>
<tr>
<td>Moving Load Analysis</td>
<td>203</td>
</tr>
<tr>
<td>Preparing a Moving Load Analysis</td>
<td>203</td>
</tr>
<tr>
<td>Linear Static Moving Load Analysis</td>
<td>203</td>
</tr>
<tr>
<td>Linear Dynamic Rolling Stock Analysis</td>
<td>203</td>
</tr>
<tr>
<td>Influence Line &amp; Surface Analysis</td>
<td>205</td>
</tr>
<tr>
<td>Influence Analysis Overview and Options</td>
<td>207</td>
</tr>
<tr>
<td>The Assumption of Linearity</td>
<td>207</td>
</tr>
<tr>
<td>The Vehicle Loading Algorithm</td>
<td>207</td>
</tr>
<tr>
<td>Performing an Influence Analysis</td>
<td>208</td>
</tr>
<tr>
<td>General Loading Parameters</td>
<td>209</td>
</tr>
<tr>
<td>Vehicular Loading Options</td>
<td>212</td>
</tr>
<tr>
<td>Vehicular Options for the 7.01 Compatible Solution Method</td>
<td>216</td>
</tr>
<tr>
<td>Uniform/Patch Loading Options</td>
<td>216</td>
</tr>
<tr>
<td>Procedure: Getting Results</td>
<td>216</td>
</tr>
<tr>
<td>Suggestions for Influence Analysis</td>
<td>219</td>
</tr>
<tr>
<td>Suggestions for Speed</td>
<td>219</td>
</tr>
<tr>
<td>Influence Surfaces in LARSA 4D Version 7.01</td>
<td>221</td>
</tr>
<tr>
<td>Design Lane Options</td>
<td>221</td>
</tr>
<tr>
<td>Vehicle Type and Placement Options</td>
<td>222</td>
</tr>
<tr>
<td>AASHTO LFD Point Loading in LARSA 4D Version 7.01</td>
<td>223</td>
</tr>
<tr>
<td>Standard Vehicles</td>
<td>225</td>
</tr>
<tr>
<td>AASHTO and CALTRANS Vehicles</td>
<td>225</td>
</tr>
<tr>
<td>AASHTO Load Patterns for Influence Analysis</td>
<td>225</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>IRC Load Patterns for Influence Analysis</td>
<td>226</td>
</tr>
<tr>
<td>Staged Construction Analysis</td>
<td>229</td>
</tr>
<tr>
<td>Overview of Staged Construction Analysis</td>
<td>231</td>
</tr>
<tr>
<td>Staged Construction Activities</td>
<td>233</td>
</tr>
<tr>
<td>Setting Up the Model</td>
<td>235</td>
</tr>
<tr>
<td>Preparation</td>
<td>235</td>
</tr>
<tr>
<td>Creating Activities in the Explorer</td>
<td>236</td>
</tr>
<tr>
<td>Creating Activities in the Stage Editor</td>
<td>238</td>
</tr>
<tr>
<td>Time Effects on Materials</td>
<td>243</td>
</tr>
<tr>
<td>Material Time Effects</td>
<td>243</td>
</tr>
<tr>
<td>Design Codes</td>
<td>245</td>
</tr>
<tr>
<td>Load Class Tracking</td>
<td>247</td>
</tr>
<tr>
<td>Setting up the model</td>
<td>247</td>
</tr>
<tr>
<td>The Load Classes of Result Cases</td>
<td>247</td>
</tr>
<tr>
<td>Accessing Class-Based Results</td>
<td>248</td>
</tr>
<tr>
<td>Staged Construction Analysis Options</td>
<td>249</td>
</tr>
<tr>
<td>Options for Analysis Scenarios</td>
<td>250</td>
</tr>
<tr>
<td>Solver Options</td>
<td>253</td>
</tr>
</tbody>
</table>
Linear Static Analysis 161
  Structure Model for Linear Static Analysis 161
  Analysis Results 161
  Assumptions in a Linear Static Analysis 162

P-Delta Analysis 163
  Using Load Combinations for the P-Delta 163
  Caveats 163
  Method 163
  Convergence Criteria 163

Nonlinear Static Analysis 165
  Why a Nonlinear Analysis 165
  Applications of the Nonlinear Analysis 166
  Equations of Equilibrium and Solution 166
  Caveats 166
  Incremental Loading 167
  Convergence Criteria 167
  Displacement Convergence and Unbalanced Forces 167
  Load Case Data for Nonlinear Static Analysis 168

Nonlinear Buckling Analysis 169
  Advantages of a Nonlinear Analysis 169
  Using the Nonlinear Buckling Analysis 169

Nonlinear Pushover Analysis 171

Nonlinear Static Analysis Options 173

Incremental Nonlinear and Nonlinear Buckling Analysis Options 175

Nonlinear Pushover Analysis Options 177
Linear Static Analysis

The linear elastic static analysis is a first order analysis that excludes nonlinearity.

The following are the basic assumptions:

- Materials are assumed to be elastic and unyielding with their properties invariable.
- The equations of equilibrium are formulated on the geometry of the unloaded structure.
- The deformations are assumed to be so small as to be insignificant in their effect on the equilibrium.
- Strains are small and stresses are proportional to strains.

Linear static analysis can compute the response of the structure to various types of static loads in the form of self weight, joint loads, joint displacements, loads applied to beam elements, thermal loads applied to beam, truss, and cable elements, and surface and thermal loads applied on plate and shell elements.

Linear static analysis is performed on load cases (page 107) whose types are set to Static and factored combination of these load cases (load combinations (page 109)). The load conditions are solved simultaneously in a single solution.

Structure Model for Linear Static Analysis

In the linear static analysis, the structure must include only linear elements. Cable and compression only foundation spring elements are examples of nonlinear elements that cannot be used in the linear static analysis. If nonlinear elements are used in the structure model while performing linear static analysis, LARSA will replace these elements with their equivalent linear elements on-the-fly without modifying the model data. For example, a cable element is automatically replaced by a truss element in linear static analysis.

Here is a list of nonlinear elements that are replaced with their linear equivalent if used in a linear static analysis:

- Cable Element
- Tension-Only and Compression-Only Truss Elements
- Grounded Spring Element with different stiffness for tension and compression
- One- or Two-Node Spring Element with a nonlinear spring property definition
- Hysteretic Beam Element
- Connection Beam Element
- Hysteretic Spring Element
- Seismic Isolator and Bearing Elements

Analysis Results

Typical results from the linear static analysis are:

- Joint displacements
- Reaction forces at support joints
- Member end forces in global XYZ or in member local directions
Assumptions in a Linear Static Analysis

By definition, the linear elastic static analysis is a first order analysis that excludes both geometric and material nonlinearity. In practical terms the assumptions of this analysis means:

- The geometry of the elements remains unchanged during the loading.
- First order, infinitesimal, and linear strain approximations can be used.
- The material behavior of the elements is linear.

When the acceleration dependent inertia forces and velocity dependent damping forces are ignored, the equations of equilibrium for static analysis are reduced to the following:

\[
K_t U = R
\]

where

- \( U \) = Joint displacement vector
- \( R \) = Applied joint force vector

with the tangent stiffness matrix, in the following form:

\[
K_t = K_o + K_g + K_s
\]

where

- \( K_t \) = Tangent stiffness
- \( K_o \) = Small displacement stiffness
- \( K_g \) = Large displacement stiffness for geometry change
- \( K_s \) = Geometric stiffness (stress stiffening)

In the linear static analysis, the displacements are assumed small, the effect of stress stiffening is ignored, and material behavior is assumed linear elastic. Therefore, the stiffness due to the change of geometry (\( K_g \)) and stress stiffening (\( K_s \)) are not considered. Then tangent stiffness matrix (\( K_t \)) becomes the same as the stiffness matrix corresponding to linear small displacement behavior:

\[
K_t = K_o
\]

It should be noted that for the linear analysis, the terms in the stiffness matrix are constant values and do not depend on the unknown displacements. The equilibrium equations are linear simultaneous equations and the closed-form solution is obtained directly for the unknown joint displacements. The element end forces are computed by simply multiplying the element end deformations by the element stiffness matrix.

For More Information, please refer to the following documentation.

If the loads applied to the structure change the geometry and stiffness, the first order linear elastic static analysis may not be satisfactory. It is then recommended to use some form of nonlinear analysis where second-order effects are considered. P-Delta is the most basic type of second order or nonlinear analysis.

The P-Delta can be defined as the secondary effect of column axial loads and lateral deflections on the moments in members. These effects can be separated as:

- A destabilizing moment equal to a gravity load times the horizontal displacement it undergoes as a result of the lateral displacement of the supporting structure.
- The influence of axial force on the flexural stiffness of individual members.

The most common situation where a second-order effect must be considered is a multi-story building under lateral loading (wind and/or earthquake) that must rely on the stiffness of the beams and columns to resist sway. A member subjected to a large axial force combined with transverse loads or to transverse end translation or rotation represents a typical P-Delta or geometric nonlinear problem. These type of second-order effects are often referred to as the beam-column problem, or a P-Delta problem.

Using Load Combinations for the P-Delta

When performing a P-Delta analysis, only active load combinations will be analyzed. The load cases within the load combination must be static load cases. The load combinations are solved separately but within a single analysis run.

The load cases are applied incrementally in the same order as entered in the load combination data. For the load combination 1.2*(Dead Load) + 1.6*(Live Load), LARSA will first apply and solve for the load vector as 1.2 times Dead Load and after convergence is established, 1.6 times the live load is applied and solved for.

Caveats

P-Delta analysis should not be used as a substitute for a nonlinear static analysis for any type of cable structure such as suspension bridges, cable-stayed bridges, and other cable structures like stadium roofs or guyed towers.

Method

The method for P-Delta in LARSA is the same method employed in all nonlinear analysis options but with a limited number of iterations and without any nonlinear elements. The method used here is the iterative full Newton-Raphson method, updating the tangent stiffness matrix in every iteration for the P-Delta effects. The equilibrium at the joints are based on the deformed shape of the elements in the structure. Therefore, the P-Delta effects are incorporated directly into the stiffness matrix as part of the geometric stiffness including geometry change and stress-stiffening.

Convergence Criteria

The Displacement Tolerance, Force Tolerance, and Maximum Number of Iterations are the basic convergence criteria specified in any of the nonlinear static analysis based options. However, for the P-Delta analysis, these values are preset.
The Displacement Tolerance is the ratio of the incremental displacement computed in the last iteration to the total displacement for the same degree of freedom. This criteria is set to 0.01 (1%) for P-Delta.

The Maximum Number of Iterations is the maximum number of iterations that can be performed for each load step. The value is set to 5 for P-Delta. The iterative analysis continues until all degrees of freedom in the model satisfy Displacement Tolerance of 0.01 but up to the maximum 5 iterations.

If the solution for a load step within a load combination is not convergent when number of iterations performed is equal or exceeds the maximum specified, a message is displayed on the screen and captured in the run log reporting nonconvergence. If there are additional load steps for the same load combination, analysis will continue if the structure has not become unstable. The unbalanced forces are carried into the next load step and a convergent solution may be obtained in the next load step. This may imply that your structure is fine and you either have to use smaller factored load steps within the load combinations or use a nonlinear static analysis where the user can increase the maximum number of iterations or use less restrictive tolerances.

For More Information, please refer to the following documentation.

• Running an Analysis in LARSA 4D User’s Manual.
Nonlinear Static Analysis

In structures, nonlinearity can be classified as material nonlinearity or as geometric nonlinearity. Material nonlinearity is associated with changes in material properties: inelastic behavior. Geometric nonlinearity is associated with changes in configuration, such as large deflections of a slender elastic beam or structures with cables.

The nonlinear static analysis takes into account geometric and material nonlinearity by iteratively solving for the displacements of a structure, stopping when convergence criteria have been met.

Deformations can significantly alter the location or distribution of loads so that equilibrium equations must be written with respect to the deformed geometry, which is not known in advance. One seeks a displacement state in which the deformed structure is in equilibrium with the loads applied to it.

Geometric nonlinearity in LARSA includes both the change in geometry and the effects of stress stiffening such as that encountered when a cable or a thin member is stressed in tension or in compression.

Structures can include several types of material nonlinearities at the element level. The inelastic beam element (a hysteretic beam element based on spread plasticity using a yield surface), the connection beam element with inelastic rotational connection springs, the nonlinear elastic and nonlinear inelastic springs, gap and hook elements, and foundation springs with nonlinear elastic or inelastic material behavior all exhibit these properties.

The nonlinear static analysis takes into consideration the following:

- Large displacements. The stiffness matrices of the elements are based on deformed geometry.
- Stress stiffening due to the internal axial forces in beam, truss, and cable elements, but not plate elements.
- Effects of follower forces from joint loads due to displaced geometry.
- Gaps and hooks which open, close, and slide.
- Nonlinear elastic and inelastic material behavior of spring elements.
- Compression-only grounded springs with zero stiffness when open.
- Yield force capacity in foundation elements.
- Inelastic material behavior in beam elements using tri-axial yield surfaces.
- Ability to have plastic hinges along the member length without specifying locations.
- Different paths during post-yield.
- Plastic moment capacity in beams.

Why a Nonlinear Analysis

It is fortunate that the linear analysis based on linear approximations works quite well for many problems in structures. In a linear analysis, the material is assumed to be unyielding and its properties invariable, and the equations of equilibrium are formulated on the geometry of the unloaded structure.

Although most analyses and designs stay within the linear elastic range, many physical situations present nonlinearities, and more and more structures demand that nonlinearities be considered for more accurate analysis and design. Slender structures such as suspension systems, arches, tall buildings, and structures subject to local yielding and cracking are basic types among structures requiring nonlinear analysis.
The fundamental goal of using the nonlinear analysis is to improve the quality of design with a more reliable prediction of performance of a structure. Nonlinear analysis can help us reduce the uncertainty regarding actual behavior of a structure. Meanwhile, the effort for modeling the structure and handling the equations of analysis is significantly increased.

Applications of the Nonlinear Analysis

Suspension bridges, cable-stay bridges, and cable domes are structures with high geometric nonlinearity. These structures are complex in shape, and usually the final deformed shape of the structure is given. The analysis involves finding the initial shape of the structure along with the prestress forces to be applied to bring the structure to its final shape under given loading conditions. The analysis needs to consider the deformed geometry and the stress stiffening effects.

Similarly, for the computation of the eigenvalues and mode shapes for suspension bridges and similar structures, the effects of large displacements and stress stiffening need to be considered. Neglecting the nonlinear effects may result in incorrect values for natural frequencies of the structure.

Consider foundations and bridge piers. These structures are usually supported by soil or friction bearing piles which may not sustain tensile forces. The model for these structures may consist of a combination of beam and plate elements representing the bridge structure. The piles and soil can be represented using the nonlinear foundation elements. The stiffness of the foundation elements will depend on the displacements. The solution can be obtained in one single analysis run using the nonlinear analysis.

Many of the seismic design and analysis procedures for bridges require consideration of nonlinear behavior. These methods assume plastic hinges will form to utilize the full displacement capacity during an earthquake. Abutments, hinges, and restrainers in bridge structures are nonlinear elements. Abutments and hinges can transfer only load in compression. Restrainers transfer only load in tension. Nonlinear analysis is a more reasonable tool to use than a series of linear analyses for these types of analyses.

Plastic deformations of steel structures, cracking or creep of reinforced concrete structures, inelastic interaction of axial force and bending moments, connection deformations, and panel zone deformations are among the many sources of nonlinearity. Rarely is it possible to model all sources of nonlinearity for a practical structure. Normally, the goal is to choose a model that does provide adequate analytical simulation of the actual behavior.

Equations of Equilibrium and Solution

In a nonlinear analysis, the equilibrium equations are formulated with respect to the deformed geometry of the structure, which is not known in advance and will change with the applied loads. The stiffness matrix depends on the unknown displacements. The result is a set of nonlinear simultaneous equations. It is necessary to employ an iterative technique to obtain a solution in nonlinear analysis. An iterative scheme based on Newton-Raphson Method has been implemented in LARSA.

Caveats

Using nonlinear analysis is not difficult, but misuse of it without understanding the methodology may lead to disasters. It is always a good practice to explore and understand the implementation of the nonlinear analysis in the computer program that is used. Users should always keep in mind that a nonlinear analysis may not have a unique solution as with the linear analysis. Therefore, any solution obtained must be verified by the user. We recommend the user knows:

- Sources of nonlinearity that are significant.
• How to represent these nonlinearities in the model.
• The formulation of the element stiffness matrices: tangent, secant or initial.
• The material nonlinearity formulation of the elements.
• The solution technique used - Full or Modified Newton-Raphson.
• How convergence criteria are satisfied.
• The number of iterations required for convergence.
• The magnitudes of unbalanced forces on the structure.

Incremental Loading

It must be kept in mind that in computing displacements for a given set of loads within a range of nonlinear behavior, it is essential to apply the loads incrementally. In general, it may not be possible to start from the unloaded state and apply the total loading in a single step due to numerical instabilities in the solution of the simultaneous equations.

The user must be cautioned that the solutions for nonlinear analyses are not direct closed form solutions as in linear analysis. Any loading causing abnormal displacements may not result in convergence.

The computed displacements, reactions and element forces at the end of any load step are the accumulated effects from all the load steps considered.

Convergence Criteria

The convergence criteria are specified by the user as the tolerable error ratio for displacement, maximum unbalanced force and maximum number of iterations. The tolerable error ratio is the incremental displacement computed in the last iteration divided by the total displacement for the same degree of freedom. The default value is 0.001 (0.10%). The iterative analysis continues until all degrees of freedom in the model satisfy both criteria. However, the number of iterations performed for each load step is limited not to exceed the maximum number of iterations specified by the user.

If the solution is not convergent for a load step, a message is displayed on the screen. If there are additional load steps, analysis continues as long as the structure is not unstable. LARSA carries the unbalanced forces into the next load step and a convergent solution may be obtained in the next load step. This may imply that your structure model is fine and you either have to increase the maximum number of iterations or use a less restrictive error ratio.

Displacement Convergence and Unbalanced Forces

The displacement convergence and unbalanced force reports include important information in evaluating the results of the nonlinear analysis. These reports are used for accepting the convergence of the results for the solution. The displacement convergence report provides the maximum error in the displacements during each iteration. The unbalanced force report tells us the error about the internal force equilibrium of the structure.

The structure must be in equilibrium for a convergent solution. When any unbalanced force remains, the structure is not in equilibrium. The unbalanced forces at the end of each load step are recorded in the log file. The report includes the joints with significant unbalanced forces and summarizes the maximum and total unbalanced force vector for all six DOF in the structure.

It is imperative that the user comprehend the information provided in these reports. In the best case, both reports have all zeros.
Load Case Data for Nonlinear Static Analysis

In LARSA, the nonlinear static analysis can be performed for:

- Computing the response of the structure to several load cases.
- Computing the response of the structure to several independent factored load cases.
- Tracing the load-deflection response of the structure.

Most of the nonlinear analysis falls in the second category where the response of the structure is computed for several load combinations in the same run. The complete load-deflection response of the structure to a series of incremental loads is required in a pushover analysis (page 171) or in a nonlinear buckling analysis (page 169).

The load cases and applied loads are defined exactly in the same manner as for the linear static analysis.

For More Information, please refer to the following documentation.

- Nonlinear Static Analysis Options on page 173.
- Nonlinear Buckling Analysis on page 169.
- Nonlinear Pushover Analysis on page 171.
Nonlinear Buckling Analysis

The nonlinear buckling analysis provides an accurate indication of the loads that will cause a structure to fail. The analysis traces the deformation of the structure using incremental load applications until the structure becomes unstable. After the analysis, load-deformation curves throughout the structure show the nature of buckling leading up to collapse.

Buckling occurs when compressive membrane forces in elements are large enough to cause a reduction in bending stiffness. In beam-column elements, this is when compressive axial force reduces bending stiffness. In plates, buckling occurs when in-plane forces reduce out-of-plane stiffness.

This analysis effectively plots load versus displacements for all parts of the structure up to collapse. The slope of these plots indicates how different parts of the structure are buckling before collapse. Just before buckling, small changes in force will yield rapid deformations.

Collapse occurs when the stiffness matrix of the structure becomes singular and thus unsolvable. A singular matrix indicates the structure has no predictable deformation under any loading. Analysis stops at this point.

Advantages of a Nonlinear Analysis

The nonlinear buckling analysis provides a more accurate indication of buckling than other methods. A linear bifurcation buckling analysis, usually based on eigenvalue solutions, oversimplifies buckling behavior and often under- or over-estimates buckling conditions. A linear analysis cannot account for important nonlinearities in a structure that play a crucial role in buckling. For instance, post-buckling strength in inelastic elements could not be accounted for, and the effects of changes in geometry and load directions are also ignored. By applying small load increments in a nonlinear analysis, the nonlinear buckling analysis accurately computes the effect of buckling as the structure approaches collapse.

Using the Nonlinear Buckling Analysis

The nonlinear buckling analysis is usually performed by first applying loads known not to cause buckling, and then incrementally applying additional loads until further analysis cannot be performed because the structure has become unstable. Loads are defined in static load cases (page 107).

Appropriate increment sizes depend entirely on the structure and nature of loads. Practically, the increment on the first condition should be 1.0 because the first condition should not cause buckling. The next condition may start with a factor of 1.0, and if it causes buckling the factor should be reduced to find the point of collapse. Once the point of collapse is known, the final loading condition should be applied in small increments, such as 10 increments of 0.1 factors. The increments and number of load applications can then be refined as needed.

When performing the analysis, the analysis engine will treat all load combinations (page 109) in the project cumulatively. Each load combination will be analyzed based on the deformation that occurred in the previous combination. Factors within the load combinations determine how to incrementally load the structure.

For instance, to apply a load case in 10 increments, create a load combination that includes the static load case to apply. Assign the static load case a factor of 0.1 in the combination, and then repeat the combination 10 times.

Factors need not be equal, and load combinations can be repeated as many times as is necessary to obtain results for the desired loading conditions.
For More Information, please refer to the following documentation.

- Nonlinear Static Analysis on page 165.
- Incremental Nonlinear and Nonlinear Buckling Analysis Options on page 175.
Nonlinear Pushover Analysis

Pushover analysis is the application of a static load case with progressively increasing scale. The pushover analysis is generally used along with the response spectra analysis (page 183) for seismic analysis, and can be used for buckling analysis as well. It is a nonlinear analysis.

Models analyzed with the pushover analysis usually contain elements with inelastic material properties, such as inelastic members (page 57) and springs (page 79), because the behavior of the model beyond its elastic limits is generally the interest of the analysis. For example, plastic hinges can be modeled with inelastic hysteretic springs. Elastic beams with yielding end connections can be modeled with the connection beam element, a type of member. When spread of plasticity with a beam is of interest, the beam can be modeled as a hysteretic beam, which is also a type of member.

The initial load factor is given by the user. The stopping criteria can be specified as a maximum displacement of a joint in a particular direction.

The analysis engine automatically chooses load factor increments based on auto-stepping and arc-length options. The user can specify the minimum and maximum step size. The maximum step size will be used except when the solution does not converge or it converges too slowly, in which case smaller step sizes are used.

For More Information, please refer to the following documentation.

• Nonlinear Pushover Analysis Options on page 177.
• Nonlinear Static Analysis on page 165.
Nonlinear Static Analysis Options

Solution Convergence

Equilibrium iterations at a given load level can cease when the result is "close enough" according to one or more criteria. Two criteria used in LARSA are that the unbalanced force be a small fraction of the total applied force in the current load level and that the current displacement increment be a small fraction of the displacement increment. The convergence criteria are specified by the user as the displacement tolerance, force tolerance and maximum number of iterations. The iterative analysis continues until all degrees of freedom in the model satisfy displacement and force tolerance criteria within the maximum number of iterations.

If the solution is not convergent for a load step, a message is displayed on the screen. If there are additional load steps, analysis will continue if the structure is not unstable. LARSA carries the unbalanced forces into the next load step and a convergent solution may be obtained in the next load step. This may imply that your structure is fine and you either have to increase the maximum number of iterations or use a less restrictive error ratio.

Max # of Iterations

The number of iterations performed for each load step will not exceed the maximum number of iterations specified by the user. We recommend 10 to 20 as a maximum number of iterations.

Displacement Tolerance

The displacement tolerance is the error ratio of the incremental displacement computed in the last iteration divided by the total displacement for the same degree of freedom. The default value is 0.001 (0.10%).

Force Tolerance

The force tolerance is the maximum unbalanced force at any degree of freedom.

Analyze For

Nonlinear static analysis can be performed for load cases and for load combinations in the same analysis run.

Analyze Load Cases

If this option is checked, nonlinear static analysis is performed for all active load cases.

Increments

When analyzing load cases, each load case is divided into this many equal load steps.

Analyze Load Combinations

The nonlinear static analysis is performed for each load combination by accumulating the loads on the structure in the same sequence as specified in the load combination data.

For More Information, please refer to the following documentation.

• Nonlinear Static Analysis on page 165.
• Running an Analysis in LARSA 4D User’s Manual.
Incremental Nonlinear and Nonlinear Buckling Analysis Options

The Incremental Nonlinear Analysis is used for computing the load-deflection response of the structure as in the plastic push-over or nonlinear buckling analysis. It can also be used as a simplified staged construction analysis.

Solution convergence control from the nonlinear static analysis (page 173) applies to this analysis type as well.

Load Combinations

In this analysis option, all active load combination cases are treated as load increments. A nonlinear static analysis is performed by accumulating the loads on the structure in the same sequence as specified from all load combinations. LARSA starts with the load cases specified for the starting load combination and performs nonlinear static analysis for all load combination steps until it reaches the ending load combination.

For example, a push-over analysis will be performed on a structure to compute the nonlinear load-deflection response of the structure as the lateral loads are incremented by 10% after its dead load is applied. We want to have the results for the following load combinations:

- Dead
- Dead + 10% Lateral
- Dead + 20% Lateral
- Dead + 30% Lateral
- Dead + 40% Lateral

The load combination data for this problem will be specified as shown below:

<table>
<thead>
<tr>
<th>Description</th>
<th>Case</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dead</td>
<td>Dead</td>
<td>1.0</td>
</tr>
<tr>
<td>Dead + 0.10*Lateral</td>
<td>Lateral</td>
<td>0.10</td>
</tr>
<tr>
<td>Dead + 0.20*Lateral</td>
<td>Lateral</td>
<td>0.10</td>
</tr>
<tr>
<td>Dead + 0.30*Lateral</td>
<td>Lateral</td>
<td>0.10</td>
</tr>
<tr>
<td>Dead + 0.40*Lateral</td>
<td>Lateral</td>
<td>0.10</td>
</tr>
</tbody>
</table>

This load combination data will instruct the program to do the following sequence of load applications in Restart Nonlinear Static:

- Load combination 1 (Step 1) - apply 100% of 'Dead'.
- Load combination 2 (Step 2) - add 10% of 'Lateral'.
- Load combination 3 (Step 3) - add 10% more 'Lateral'.
- Load combination 4 (Step 4) - add 10% more 'Lateral'.
• Load combination 5 (Step 5) - add 10% more 'Lateral'.

Starting Load Combination
The nonlinear static analysis starts with loads applied from this load combination case. Normally you will select the very first load combination in the data as the starting load combination. However the analysis can also start from a combination where the previous load combination has been solved in a prior run. There are limitations on the type of elements that can be used in a restart analysis performed from a previously solved load combination.

Ending Load Combination
The analysis is performed for all load combinations including the one specified as the ending one.

For More Information, please refer to the following documentation.

• Nonlinear Static Analysis on page 165.
• Running an Analysis in LARSA 4D User’s Manual.
Nonlinear Pushover Analysis Options

Solution convergence control from the nonlinear static analysis (page 173) applies to this analysis type as well.

Pushover Stopping Criteria

Analysis will stop based upon the criteria specified here. Select the joint, the direction and the value for maximum displacement.

Auto-Increment Options

The pushover analysis is based on automatically incrementing the load vector. Specify here the multipliers for the load vector that will be used in computing the initial load vector and the incremental load vector limitation.

Initial
This is the factor multiplying the load vector for computing the load to be applied on the structure in the first load step.

Minimum
The minimum factor multiplying the load vector that is used in any load step. When the iterative analysis requires reduction in the magnitude of the load vector, the multiplier cannot be smaller than this value.

Maximum
The maximum factor multiplying the load vector that is used in any load step.

For More Information, please refer to the following documentation.

Eigenvalue Analysis

Eigenvalue and Stressed Eigenvalue Analysis  181
   Unstressed (Standard) Eigenvalue Analysis  181
   Stressed Eigenvalue Analysis  181

Response Spectra Analysis  183
   Response Spectra Curve  183
   Modal Damping  183
   Modal Combination  184
   Spatial Combination  184
   Ground Motion Directions  184
   Response Spectra Load Cases  185
   Caveat About Sign  186
   Analysis Options for Response Spectra  186

Eigenvalue Analysis Options  187

Stressed Eigenvalue Analysis Options  189
Eigenvalue and Stressed Eigenvalue Analysis

The eigenvalue analysis is performed to extract the undamped free-vibration mode shapes and frequencies of a structure. The eigenvalue analysis is important as a precursor to any dynamic analysis because knowledge of the structure's natural frequencies and modes can help to characterize its dynamic response. The results of an eigenvalue analysis also help determine the number of modes to be used in a response spectra analysis.

Each eigenvalue-eigenvector pair is called a natural vibration mode of the structure. The modes are identified by numbers from 1 to n in the order of increasing frequency. The eigenvalue is the square of the circular frequency for that mode. The cyclic frequency, \( f \), and period, \( T \), of the mode are multiplicative inverses.

The mass matrix automatically includes the mass of members if section area and weight density are provided, and for plates if weight density is provided. No other elements automatically include mass in dynamic analysis. Mass elements (page 85), which are ignored in static analysis, are used to add additional (usually nonstructural) mass for eigenvalue analysis. It is also possible to compute mass from static loads by putting static loads in a load case (page 107) marked as representing dynamic mass rather than static loading.

When an eigenvalue analysis is performed, the results from a previous eigenvalue analysis are automatically overwritten in the results database.

Unstressed (Standard) Eigenvalue Analysis

When an eigenvalue analysis of the undeformed structure is requested, the stiffness matrix is computed using the undeformed geometry of the structure. Because no loads are applied, stress stiffening is not included. Static load cases are not used in this type of eigenvalue analysis.

Stressed Eigenvalue Analysis

In the stressed eigenvalue analysis, an eigenvalue analysis performed based on a nonlinear static analysis. The stiffness matrix includes the effects of static loads (stress stiffening) and the deformed geometry of the structure.

The user selects the load case for which the program performs a nonlinear static analysis. Eigenvalues are extracted based on the stiffness of the structure after the applied load.

For More Information, please refer to the following documentation.

- Eigenvalue Analysis Options on page 187.
- Stressed Eigenvalue Analysis Options on page 189.
Response Spectra Analysis

Response spectra analysis (RSA) is a statistical type of analysis where the peak response of a multi-degree-freedom system can be calculated. Results of RSA are not exact but are accurate enough for structural design applications. RSA provides an estimate for the response of forces and deformations of a structure to shock loading conditions, such as in seismic analysis.

RSA is not a dynamic analysis but a procedure for dynamic analysis, excluding response history. Although this analysis involves only static analysis, RSA is still considered a dynamic analysis procedure because it uses the vibration properties such as natural frequencies, corresponding mode shapes, and modal damping ratios of the structure and the dynamic characteristics of the ground motion through its response spectrum.

RSA requires the results of an eigenvalue analysis along with a known spectrum to calculate the most likely peak responses for displacements, forces and stresses that occur in the structure. To perform the response spectra analysis, the natural frequencies and mode shapes are either calculated prior to RSA within the same analysis run or must have already been computed in a previous run.

The response spectra analysis can be performed using natural frequencies and mode shapes of the undeformed structure (Eigenvalue Analysis) or of the deformed structure subject to loads (Stressed Eigenvalue Analysis).

Response Spectra Curve

Response spectra curves are graphs of the maximum response (displacement, velocity, or acceleration) to a specified excitation for a single degree-of-freedom system. The abscissa of the curve is the natural frequency (or period) of the structure, and the ordinate is the maximum response.

The construction of these plots requires the solution of single degree-of-freedom systems for a sequence of values of the natural frequency and of the damping ratio. Each solution provides only one point of the response spectrum and the task is lengthy to obtain the maximum response value for all frequencies of interest. Once these curves are available for the seismic excitation, the RSA is reduced to calculation of the natural frequencies and mode shapes of the structure and the use of the response spectra curves.

In general, the user does not develop the response spectra curves but obtains and use them in the program as a database of RSA curves. See The Database Editor [in LARSA 4D User’s Manual] and Connecting Databases [in LARSA 4D User’s Manual].

It is important to note that the values for the abscissa and ordinate of response spectra curves must be positive or zero. The ordinates can be the actual values or values normalized with respect to a ground acceleration of 1.0 g. If the curve ordinate is normalized with respect to g, then you must enter the appropriate g value for the curve in the database editor.

Modal Damping

The damping in the structure affects the shape of the response spectra curve and it also affects the coupling between the modes for modal combination when using CQC. The damping in the structure is included in the shape of the response spectra curve.

Modal damping is given as either a critical damping ratio or as a modal damping curve. A damping ratio must be entered if the CQC modal combination rule is used.
Modal Combination

The modal combination rules are for combining the peak modal responses to determine the peak value of the total response. In general, the peaks for modal responses are at different instants and the peak for combined response is at a different instant with no information about when the peak modal values occur. Modal combination rules are approximations introduced in combining the peaks of all modal responses determined using the earthquake response spectra.

The absolute sum (ABSSUM) modal combination rule is based on the assumption that all modal peaks occur at the same instant, ignoring the sign of response. This leads to an upper bound of the peak value of the total response. This method is not popular in structural design because it is usually considered too conservative.

The square-root-of-sum-of-squares (SRSS) modal combination rule, developed by E. Rosenblueth (1951), provides good response estimates for structures with well-separated natural frequencies. In SRSS, the peak response in each mode is squared, the squared modal peaks are summed, and the square root of the sum is the estimate of the peak total response.

Starting in the late 1960s, there have been several formulations for the complete quadratic combination (CQC) modal combination rule that is applicable to a wider class of structures. CQC eliminates the limitations of SRSS. E. Rosenblueth and J. Elorduy (1969) were the earliest in CQC formulation. However, the most widely used CQC formulation now is by A. Der Kiureghian (1981) and his formulation is used in LARSA.

According to the CQC rule, the product of the modal peak responses of the ith mode and jth mode and the correlation coefficient relating these 2 modes are summed and the square root of the sum becomes the estimate of the peak total response. The correlation coefficient varies from 0 to 1 with the value as 1 when i and j refer to the same mode. When i is not equal to j, each of the modal peak values may be positive or negative. A cross term is negative when the modal static responses assume opposite signs. Therefore, the estimate obtained by the CQC rule may be larger or smaller than the estimate provided by the SRSS rule.

Spatial Combination

Usually ground excitation occurs simultaneously in three directions (X, Y and Z). The ground excitation for each direction can be specified using three independent load cases, and their effects can be combined to yield the resultant effect. All three excitations can also be specified in the same load case. If more than one ground excitation is specified in a load case, you must also select the spatial combination method for combining the results from each direction. The spatial combination method can be SRSS for Square Root of Sum of Squares or ABS for Absolute Sum. If the excitation is for a single direction, the spatial combination method is ignored.

Ground Motion Directions

Ground motion in up to three directions can be specified in a response spectra load case. The directions usually coincide with the global x/y/z directions. By default, spectrum curves 1 and 2 are applied in the global x- and y- directions, respectively. The angle parameter specifies a rotation about the global z-axis to change those directions. The third direction is always in the direction of the global z-axis.

Generally, only one spectrum curve is applied per load case. If more than one curve is specified, their results are combined using the spatial combination method specified.
Response Spectra Load Cases

The computations in response spectra analysis are organized into response cases similar to load cases in a static analysis. Various response cases of ground motion can be defined as separate load cases, and they can be analyzed in a single run.

Before entering the load data for a response spectra case, the database file containing the spectrum curves must be connected to the project. See Connecting Databases [in LARSA 4D User’s Manual].

Spectrum Curve in Direction 1
The response spectra curve describing the ground motion applied in the first direction.

Scale in Direction 1
A multiplier applied to the amplitudes on the curve applied in the first direction.

Spectrum Curve in Direction 2
The response spectra curve describing the ground motion applied in the second direction.

Scale in Direction 2
A multiplier applied to the amplitudes on the curve applied in the second direction.

Spectrum Curve in Global Z
The response spectra curve describing the ground motion applied in the global-Z direction.

Scale in Global Z
A multiplier applied to the amplitudes on the curve applied in Z.

Angle
The angle in the xy-plane from the global x-axis to the direction of the first spectrum curve, in degrees.

Modal Combination Method
The modal combination method determines how the modal values are to be combined into final results. The modal combination options are CQC for Complete Quadratic Combination and SRSS for Square Root of Sum of Squares.

Spatial Combination Method
If more than one ground excitation is specified in a load case, you must also select the Spatial Combination method for combining the results from each direction. The Spatial Combination Method can be SRSS for Square Root of Sum of Squares or ABS for Absolute Sum. If the excitation is for a single direction, the Spatial Combination Method is ignored.

Sign Convention
Usually the results from a response spectra analysis are displayed with no signs. However LARSA gives the user the option to select sign compatible results that are assigned using the sign convention from the mode with largest mass participation.

Modal Combination System
Select the coordinate system for the modal combination of joint results such as reaction forces and displacements. Transformations are not mathematically applicable on the results obtained using modal combinations. For example, reaction forces computed cannot be transformed to obtain the reaction forces in another set of directions. The displacements and reaction forces at joints assigned displacement coordinate systems can be displayed in reference to the global coordinate system when the Modal Combination System is selected as "Global" and the same results can be examined in reference to the displacement coordinate system of the joint when the selection is "Displacement Coordinate System."
**Modal Damping Curve**

If the damping is not constant throughout the frequency range of the structure, a curve where the critical damping ratio can be defined as a function of frequency can be assigned. This curve data is created and saved within the same database file as the response spectrum curves.

**Critical Damping Ratio**

Used in CQC based modal combination in computation of correlation coefficients. The value must be the damping ratio used to develop the response spectra curve used in the load case. The default is 0.05%.

**Caveat About Sign**

Response spectra analysis combines the maxima of all modes for all spatial directions. The response maxima of the modes are first combined for a given motion in a spatial direction. Then it is followed by a spatial combination if more than one excitation direction is specified. Due to the nature of the modal and spatial combination methods, sign information is lost in producing results.

The sign of the results is important for many reasons. Without the sign, the deformed shape of the structure can be displayed using only node-to-node displacements. The force diagrams cannot be displayed in a meaningful manner.

**Analysis Options for Response Spectra**

The Linear Static + Eigenvalue + Response Spectra analysis type performs multiple analysis types in the same run. With this option, LARSA will perform a linear static analysis on static load cases, an eigenvalue analysis for the number of mode shapes requested, and finally a response spectra analysis. It will also perform any combinations that have been set up in load combinations (page 109).

The standard Response Spectra Analysis will require an eigenvalue analysis that has been previously analyzed and saved.

For More Information, please refer to the following documentation.

- Running an Analysis in *LARSA 4D User’s Manual*.
- Connecting Databases in *LARSA 4D User’s Manual*.
Modal Analysis

The eigenvalue analysis is used to determine the natural frequencies and mode shapes of a structure. The equations of motion are solved for an undamped system without any applied forces.

The dynamic characteristics, which are the resonant frequencies (eigenvalues) and mode shapes (eigenvectors) of the undamped structure are determined.

**Number of Mode Shapes**

The number of mode shapes to be extracted. The requested number of mode shapes can not be greater than the number of mass degrees of freedom.

For More Information, please refer to the following documentation.

- Eigenvalue and Stressed Eigenvalue Analysis on page 181.
Stressed Eigenvalue Analysis Options

Solution convergence control from the nonlinear static analysis (page 173) and mode shape options from the standard eigenvalue analysis (page 187) apply to this analysis type as well.

Load Case

Stressed eigenvalue analysis requires finding the deformed position of a structure along with the forces in the members prior to calculating the mode shapes. These options determine which static loads will be applied on the structure.

**Load Case**
This option is not currently supported.

**Load Combination**
Select a load combination from which the static loads will be applied to the structure.

For More Information, please refer to the following documentation.

• Running an Analysis in *LARSA 4D User’s Manual*.
• Eigenvalue and Stressed Eigenvalue Analysis on page 181.
Time History Analysis

Linear Time History Analysis 193

Nonlinear Time History Analysis 195
  Overview 195
  Newmark-Beta with Newton-Raphson 195
  Sparse Solver Technology 195
  Nonlinear Time History Data 195

Linear Time History Analysis Options 197

Nonlinear Time-History Analysis Options 199
  Geometric Nonlinearity 199
Linear Time History Analysis

Time history analysis is for computing the response of the structure to time-dependent loads which are specified as force, displacement, or acceleration excitation curves at joints, uniform base acceleration, or factored static loads. Multiple excitations can be specified simultaneously.

The excitations in a time history analysis are organized into time history load cases (page 129) containing excitations and initial conditions at joints. Multiple excitations can be specified simultaneously within a load case. For example, different displacement-based time history excitations at different supports can be applied to simulate variable multi-support excitation such as in long-span bridge structures. Or, separate load cases can be used to perform multiple analyses. Each time history load case generates a group of result cases for all the integration time steps solved and saved.

Excitation curves are loaded from external time-history databases. See Connecting Databases [in LARSA 4D User’s Manual].

The initial displacement, velocity, and acceleration conditions of the structure can be specified in time history load cases. The default assumes the structure is initially at rest with zero displacements, velocities and accelerations.

A static load case containing joint, member, and plate loads can be chosen in conjunction with a “factor” excitation function. An “inconsistent load vector” method is used for these element loads, meaning element loads are transferred to joints and element-internal forces (such as Member Sectional Forces (page 271) and Plate Forces on Center (page 287)) will not account for these loads. When using factored static element loads, only results reported at joints should be used.

Dynamic Rolling Stock Analysis is a type of Linear Time History Analysis. See Moving Load Analysis (page 203).

For More Information, please refer to the following documentation.

• Time History Loads on page 129.
• Nonlinear Time History Analysis on page 195.
• Linear Time History Analysis Options on page 197.
Nonlinear Time History Analysis

Overview

The nonlinear time history analysis is an extension of the linear time history analysis in which elements with geometric nonlinearity (e.g., cables) and elements with material nonlinear behavior (e.g., nonlinear elastic and inelastic springs) can be included. The use of such an advanced analysis method also may be the most cost effective approach to estimate the response of a structure to dynamic loads when nonlinear behavior needs to be accounted for realistically.

The most common applications of nonlinear time history in structural engineering are dynamic response to seismic and blast loadings when the structure includes elements with geometric nonlinearity like cables, or protective devices like seismic isolators, or elements with material nonlinearity such as inelastic springs with energy dissipater characteristics.

Newmark-Beta with Newton-Raphson

The nonlinear time history analysis is carried out by using the Newmark-Beta time integration algorithm with the (full) Newton-Raphson method using iterations within each integration time step. The number of iterations performed within each time step is controlled by user specified convergence criteria.

Newark and Newton-Raphson methods are used because of their superior accuracy. The solution is carried out in an incremental form. The forces from the seismic isolation elements and the energy dissipation elements (viscous, frictional and hysteretic devices) are computed at every iteration during the solution process.

The solution algorithm has the option of using a constant time step or adaptive variable time step. In the variable time step option, the step size is reduced automatically to a fraction of its value when needed for better convergence.

Sparse Solver Technology

The nonlinear time history analysis requires significant computational effort and computer time. With the use of the sparse solver technology, the computational effort is drastically reduced when compared to more widely known and used skyline methods. The nonlinear time history analysis in LARSA 4D requires the Direct Sparse Solver.

Nonlinear Time History Data

The setup for a nonlinear time history analysis is identical to that of the Linear Time History Analysis (page 193), with extra analysis options specified.

The nonlinear time history analysis requires more solution control parameters than other analysis options. These parameters include options to choose integration method, constant or self-adaptive time steps, choice of iterative method, and setting the convergence criteria. See Nonlinear Time-History Analysis Options (page 199).

For More Information, please refer to the following documentation.

- Linear Time History Analysis on page 193.
- Nonlinear Time-History Analysis Options on page 199.
Linear Time History Analysis Options

Time Step Size

Specify the time step sizes used in Newmark Integration.

**Time Step Size**

The time step interval, in seconds.

Save/End Options

This entry determines the ending time step to be analyzed and at what time interval solutions should be recorded.

**Solution Ending Time**

The analysis starts at time zero and ends with the time specified here. The value specified here depends on how time is measured in the excitation records.

**Step Interval for Saving Results**

The step interval for saving results. The results for the computed displacements, velocities, accelerations, and element forces are saved at the interval specified here. If the interval is 1, the results are saved for all integration steps. Two, for instance, specifies to save every other integration step.

Rayleigh Damping Constants

Rayleigh damping, \( c \), is defined as \( a_0m + a_1k \), where \( a_0 \) is the mass-proportional Rayleigh damping constant and \( a_1 \) is the stiffness-proportional Rayleigh damping constant.

The coefficients \( a_0 \) and \( a_1 \) can be determined from specified modal damping ratios, \( r_i \) and \( r_j \), for the \( i \) and \( j \) modes, respectively. If both modes are assumed to have the same damping ratio \( r \), then

\[
a_0 = \frac{2 \, r \, w_i \, w_j}{(w_i + w_j)} \quad a_1 = \frac{2 \, r}{(w_i + w_j)} \quad \text{where} \; w_i \; \text{and} \; w_j \; \text{are the two natural frequencies in rad per sec.}
\]

Note: Rayleigh damping constants are not the modal damping ratios.

For More Information, please refer to the following documentation.

- Running an Analysis in *LARSA 4D User’s Manual*.
- Linear Time History Analysis on page 193.
Nonlinear Time-History Analysis Options

Geometric Nonlinearity

Include Geometric Nonlinearity
If this option is checked, geometric nonlinearity is included in the analysis.

For More Information, please refer to the following documentation.

• Running an Analysis in LARSA 4D User’s Manual.
• Nonlinear Time History Analysis on page 195.
Live Load Analysis

Moving Load Analysis 203
- Preparing a Moving Load Analysis 203
- Linear Static Moving Load Analysis 203
- Linear Dynamic Rolling Stock Analysis 203

Influence Line & Surface Analysis 205
- Influence Analysis Overview and Options 207
  - The Assumption of Linearity 207
  - The Vehicle Loading Algorithm 207
  - Performing an Influence Analysis 208
  - General Loading Parameters 209
  - Vehicular Loading Options 212
  - Vehicular Options for the 7.01 Compatible Solution Method 216
- Uniform/Patch Loading Options 216
- Procedure: Getting Results 216

Suggestions for Influence Analysis 219
- Suggestions for Speed 219

Influence Surfaces in LARSA 4D Version 7.01 221
- Design Lane Options 221
- Vehicle Type and Placement Options 222

AASHTO LFD Point Loading in LARSA 4D Version 7.01 223

Standard Vehicles 225
- AASHTO and CALTRANS Vehicles 225
- AASHTO Load Patterns for Influence Analysis 225
- IRC Load Patterns for Influence Analysis 226
Moving Load Analysis

The moving load analysis simulates the movement of one or more vehicles passing over a structure on a user-defined path. Linear Static Moving Load Analysis and Linear Dynamic Rolling Stock Analysis are available.

Preparing a Moving Load Analysis

To prepare a moving load analysis, begin by creating a lane (page 99), which defines the path of the vehicle load on the structure. The torsion effects on beam elements are computed and included if lane eccentricity is specified.

Then connect a vehicle load pattern database (see Connecting Databases [in LARSA 4D User’s Manual]). LARSA 4D includes several Standard Vehicles (page 225) that can be used. User-defined vehicle patterns can be created using the Database Editor [see “The Database Editor” in LARSA 4D User’s Manual].

Then, create a load case. The type of the load case in which the moving loads are entered should be set to Moving (Static) for a Linear Static Moving Load Analysis or Time History for a Linear Dynamic Rolling Stock Analysis.

Finally, add Moving Loads (page 125) to the load case. For each moving load, select a lane, a vehicle pattern, a direction of loading (typically Global -Z) and motion information including an initial position, a forward increment or speed (depending on the analysis type), and a direction (forward or backward). If multiple Moving Loads are given in the same load case, the vehicles are applied simultaneously. By loading multiple lanes simultaneously with loading factors, the effect of distributing the load of a vehicle across lanes can be simulated.

Linear Static Moving Load Analysis

This analysis automatically generates all of the load cases for vehicle positions along the user-defined path and performs a Linear Static Analysis (page 161) on each generated case. In each successive case, the vehicle load pattern is moved forward by the Position Increment given in the moving load (page 125).

The moving load analysis is a linear analysis and therefore the structure should not include nonlinear elements such as cable elements or compression only foundation spring type nonlinear elements. If nonlinear elements are used in the model during moving load analysis, the elements will be analyzed as an equivalent linear element.

The type of the load case in which the moving loads are entered should be set to Moving (Static).

The results of the generated moving load cases can be combined with any other load case using Load Combinations (page 109) or Linear Result Combinations [in LARSA 4D User’s Manual], and envelopes are simplified with Extreme Effect Groups [in LARSA 4D User’s Manual].

Linear static moving load analysis is also used in the first step of influence-based live load analysis. See Influence Line & Surface Analysis (page 205).

Linear Dynamic Rolling Stock Analysis

The Linear Dynamic Rolling Stock Analysis is a Linear Time History Analysis (page 193). In each time step, the vehicle is moved forward according to the Vehicle Speed given in the moving load (page 125).
Like the Linear Static Moving Load Analysis, the Linear Dynamic Rolling Stock Analysis is a linear analysis and therefore the structure should not include nonlinear elements (see above). However, it is a dynamic analysis in which the mass and motion of the structure is considered. The mass of the vehicle is not considered and should be small relative to the mass of the structure and uncoupled with the resonances of the structure.

An “inconsistent load vector” method is used, meaning element loads are transferred to joints and element-internal forces (such as Member Sectional Forces (page 271) and Plate Forces on Center (page 287)) will not account for these loads. Only results reported at joints should be used.

The type of the load case in which the moving loads are entered should be set to Time History.

The Linear Time History Analysis Options (page 197) are applicable to this analysis type.

For More Information, please refer to the following documentation.

- Lanes on page 99.
- Standard Vehicles on page 225.
- The Database Editor in LARSA 4D User’s Manual.
- Moving Loads on page 125.
- Linear Time History Analysis on page 193.
Influence Line & Surface Analysis

Influence line and surface analysis is an extension of the moving load analysis which uses the effects of a unit load placed at many locations to rapidly compute the envelope of effects of arbitrarily complex load patterns.

### Influence Analysis Overview and Options
- The Assumption of Linearity 207
- The Vehicle Loading Algorithm 207
- Performing an Influence Analysis 208
- General Loading Parameters 209
- Vehicular Loading Options 212
- Vehicular Options for the 7.01 Compatible Solution Method 216
- Uniform/Patch Loading Options 216
- Procedure: Getting Results 216

### Suggestions for Influence Analysis
- Suggestions for Speed 219

### Influence Surfaces in LARSA 4D Version 7.01
- Design Lane Options 221
- Vehicle Type and Placement Options 222

### AASHTO LFD Point Loading in LARSA 4D Version 7.01
- 223
Influence Analysis Overview and Options

Influence line and surface analysis is an extension of the moving load analysis which uses the effects of a unit load placed at many locations to rapidly compute the envelope of effects of arbitrarily complex load patterns.

An influence line or surface analysis is often needed to determine the design conditions that are required in bridge design codes. In these codes, it is not sufficient to envelope the results of a single vehicle placed at each station along a girder. There are a large number of variables, including vehicle location and length, number of vehicles, and lane locations, that need to be controlled simultaneously to determine the worst case scenarios. In many cases there are so many loading possibilities that it would not be cost effective to run a complete analysis for each.

Influence analysis is the combination of 1) an assumption of linearity, and 2) efficiently searching the loading possibilities for the extreme effects using an appropriate algorithm.

The Assumption of Linearity

The core of an influence line or surface analysis is the assumption of linearity, that is, that the result of two simultaneous loads is equal to the sum of the results of the two loads when each is placed alone. This assumption allows results to be computed for different design conditions without the need to re-run a full model analysis because new results can be determined simply by adding and multiplying previous results.

In practice, influence analysis is carried out by first computing the results for a load of magnitude 1.0 (a unit load) either at pre-set increments along a line or curve (an influence line) or on a grid of locations (an influence surface). For instance, an influence line analysis of a 100-foot span with a 1-foot increment would require 100 result cases; an influence surface analysis of a 100-foot by 24-foot lane surface, with a 1-foot longitudinal and half-foot transverse grid spacing would require 4,800 result cases. The results are called influence coefficients. The effect of a wheel or axle is computed by multiplying the results of the unit load (the coefficient) at the location of the wheel/axle by the magnitude of the wheel/axle (or, if the wheel/axle does not fall directly on the location of a unit load, then by interpolating the results of the two (for influence lines) or four (for influence surfaces) nearest unit loads). The effect of vehicle is the sum of the effects of its wheels or axles, etc.

Uniform lane loading is similarly computed from unit loads by adding together the effects of the unit loads within the region of lane loading, and multiplying by an appropriate factor according to the magnitude of the load (and for influence surfaces, the transverse width over which the load is placed).

Bear in mind that the quality of influence analysis results rests on the suitability of the assumption of linearity as well as on the choice of unit load spacing.

For use with AASHTO LRFD, please see Guide to AASHTO LRFD 2007 Section 3.6: Live Loads [in LARSA 4D User’s Manual].

The Vehicle Loading Algorithm

The second part of influence analysis is the vehicle loading algorithm. No loading algorithm could try each possible combination of vehicle locations one-by-one. Such a process would take an inordinate amount of computing time.

The current algorithm implemented by LARSA 4D (starting with version 7.5) will find the worst-case loading configurations of vehicles and lane load. Note that although the unit loads are computed at a finite number of locations
(i.e. at fixed spacing), through interpolation it is possible to compute the effects of loading at at any location on or off the grid of influence coefficients. Nevertheless, LARSA 4D's influence algorithm assumes that if two loads are spaced no closer than the coefficient spacing, then one must have the maximum effect (that is, the maximum cannot occur between them). Additionally, vehicles with variable axle spacing are in some cases assumed to have either their minimum or maximum spacing. Only point force loads are supported in vehicle patterns.

This so-called "standard solution method" replaces and improves on the algorithm used in LARSA 4D version 7.01 and earlier. The improvements include the ability for trucks to be placed at the very edges of design lanes regardless of the transverse coefficient spacing, improved accuracy by solving for the best lane and vehicle positions simultaneously, improved speed especially on multi-core/multi-processor computers, simultaneous optimization over multiple lane configurations (different truck types and limits on the number of trucks of any type), and the ability to combine the span-by-span requirement of AASHTO LFD with multiple presence factors. The previous algorithm was a heuristic which for a large number of vehicles approximated and did not necessarily solve for the most extreme effect.

The influence algorithm effectively simulates all of the vehicle configurations that arise from the different number and location of design (traffic) lanes and vehicles placed, the direction of each vehicle (forward or backward), and the length of variable-axle-position load patterns. A traffic lane is a single row of vehicles. Design lanes are always parallel to the centerline of the influence surface. An influence line is always exactly one design lane.

One advantage of the influence surface method over the influence line method is the ability to apply "two-dimensional" load patterns. The load patterns are so named because the axles are arranged over a surface, rather than in a line. It is always recommended to use "2D" vehicle load patterns when using influence surfaces. For AASHTO LFD and LRFD, see the standard database AASHTO Vehicle Patterns. Only 2D load patterns model the width of vehicles, which plays a role in the requirements for vehicle placement within lanes.

**Performing an Influence Analysis**

An influence analysis is performed in the following manner:

- A lane or surface (page 99) is defined specifying the path of vehicles on the structure (for influence lines) or the surface on plate elements on which vehicles will be placed (for influence surface). This defines the path or surface on which unit loads are placed to compute the coefficients that make up the influence line or influence surface. A lane can be placed on member or plate elements, while a surface must be defined on plate elements only and its width must be given.

- Note that for influence surfaces, what LARSA 4D calls a "lane" is the complete deck surface with possibly multiple rows of traffic, not an individual traffic or design lane. This rectangular "lane" surface may extend over the bounds of the structure in the case that the structure is skewed or otherwise non-rectangular.

- An influence load (page 127) is created on each lane. The influence load is simply a point load with a magnitude of 1 with forward (longitudinal) and for influence surfaces transverse grid spacing. The elements in the lane (or surface) determine how the loading is distributed onto the structure. For influence line analysis that requires the distribution of the axles across multiple girders, specify multiple lanes, one on each girder, within a single load case, and use factors that add to 1.0. (In this case, each influence coefficient will actually be generated by two loads whose magnitude sums to 1.0, with the loads marching along the girders simultaneously.)

- Run a Moving Load Analysis (page 203) to generate an influence line or surface, i.e. the results of unit loading called influence coefficients.

- Influence result cases described below are set up with the design-appropriate vehicle and lane loading parameters. These result cases will compute and show the worst-case results and corresponding loading configuration.
In an influence line analysis, each axle or wheel of the loaded vehicles will be lumped on the centerlines of the members in the lane path, unless either 1) the lane is specified with offsets that take it away from the member centerline, 2) if a transverse offset is specified in the influence result case options, or 3) the load pattern has transverse offsets specified on the wheels. In cases 2 and 3, a second set of influence coefficients, based on a one-unit torque, is used to compute the moment induced by the eccentricity. Method 1 should not be combined with methods 2 and 3. Additionally, load distributions across multiple girders can be accomplished either through rigid cross-beams connecting girders, or by loading multiple lanes simultaneously (see below).

With the influence surface method, load distribution is accomplished automatically because of the nature of the models used for influence surfaces. Plate decks by their nature will spread load from the point of contact with the wheel throughout the deck, and to any connected elements.

**General Loading Parameters**

After the Moving Load Analysis is complete, create an influence result case by opening the [Results] menu and choosing **Influence Line/Surface Case**. This is where the lane and load pattern, and other options, are chosen. The first screen of options is shown below:

![Influence Analysis](image)

General Perameters for an Influence Result Case

The following options are presented.

**Solution Method**

The Standard Solution Method should be used for all new projects. Also available is the algorithm used in LARSA 4D prior to version 7.5, called the 7.01 Compatible Method, with either the one-vehicle-per-lane or two-vehicles-per-lane options.

**Influence Coefficients**

Choose the lane from the list. The list shows all influence coefficient results already generated by the last Moving Load Analysis.

**Design Lane Width**

209
For influence surfaces, the width of a design lane, i.e. a single row of traffic. Typically 12 feet. With the standard solution method, LARSA 4D will place as many lanes as will fit on the surface, according to the width of the surface specified in the input geometry, simultaneously maximizing the effect according to any multiple presence factors. (With the 7.01-compatible algorithm, LARSA 4D will load only as many lanes as are needed to place the maximum number of vehicles specified.)

**Transverse Offset**
This option for influence lines simplifies the creation of multiple lanes at different transverse locations. With this method, moments on the girder are computed by adding in the effect of special unit-torque influence coefficients, multiplied by the transverse offset and the axle/wheel magnitude. This method may be more useful than adjusting the lane geometry because it eliminates the need to create a separate lane definition (page 99) for each traffic lane. Only a single lane definition, specified on the centerline of the bridge girder, is necessary to compute the effects of vehicles on parallel lanes. The sign convention generally follows the right-hand-rule but is truly determined by the orientation of the member elements and the direction of the unit force loading, so it is always best to verify this by inspection.

**Load for Extreme Force Effects**
When this option is checked, vehicle wheels/axles and lane loading locations that do not contribute to extreme forces are not considered. For example, when enveloping for maximum forces, axles that cause negative forces are ignored. Normally this option should be on. In LARSA 4D 7.6, this option does not affect lane loading which is always applied in a patch loading manner. In LARSA 4D 7.1, this option must be turned on to apply lane loading as patch loading.

**Complete Patterns Only**
Often the worst position of a vehicle is when it is only partially on the lane or surface. When this option is checked, only vehicle positions where all wheels/axles fall on the lane or surface are considered. When no wheel/axle is at the zero-position of the vehicle load pattern, the zero-position must also be on the surface. When this option is not checked, the vehicle may extend past the beginning or end of the line or surface (longitudinally), but for an influence surface vehicles will never extend transversely outside of a design lane, which are always fully contained within the defined surface area.

**Overall Factor**
The overall factor is applied to the final influence results. Usually this is 1.0.

**Multiple Presence Factors**
For influence surfaces, this is a set of factors to apply to the final results depending on the number of lanes actually loaded. The number of lanes loaded will depend on how many lanes fit on the roadway surface and whether adding additional vehicles can produce a more extreme effect (since more lanes usually means a reduced factor). When editing multiple presence factors, enter a table whose first column is the number of lanes loaded and whose second column is the corresponding factor (see below). (With the 7.01-compatible solution algorithm, this option applies only to the one- or two-vehicles-per-lane option.)
The following advanced options are for modeling centrifugal and braking effects.

**Overturning Effect Factors**

For influence surfaces, this option allows the vehicle load pattern to be modified according to a common factoring scheme used with centrifugal force due to a curved direction of travel and superelevation. Load factors are specified for the wheels on the side of the vehicle closest to the positive transverse side of the influence surface (always check which way this is by inspection). On the assumption of two wheels of equal magnitude per axle, a corresponding factor of 2.0 minus the factor will be applied to the wheels on the other side of the vehicle (thus resulting in no overall change in applied force). To accommodate multiple spans with different degrees and directions of horizontal curvature and superelevation, the factor is entered as a curve. If the design lane has horizontal curvature toward the negative transverse side of the influence surface or typical superelevation in such a circumstance, the overturning effect will cause the vehicle load to shift toward the positive transverse side. In this case, specify a factor between 1.0 and 2.0. If the design lane is curving toward the positive transverse side of the influence, specify a factor between 0.0 and 1.0. The figure below shows how to enter a factor that is constant along the length of a 1000-foot bridge curving clockwise. (Note that these factors are for vertical loads. For centrifugal force effects, see Guide to AASHTO LRFD 2007 Section 3.6: Live Loads [in *LARSA 4D User’s Manual*].)
Overturning Effect Factors

Longitudinal Factors
This option allows an overall factor to be applied that varies longitudinally. It is primarily intended to be used when modeling centrifugal forces on a multi-curve bridge. For an example, see Guide to AASHTO LRFD 2007 Section 3.6: Live Loads [in LARSA 4D User’s Manual].

Coefficients are Signless
This option is used to model braking forces, which can occur in either longitudinal direction if vehicles are permitted to travel in either direction. When this option is on, each influence coefficient is treated as if it were its absolute value.

Vehicular Loading Options
The second set of options pertains to vehicular loading. With the standard solution method, vehicular loading may be omitted to apply only uniform lane loading. The options are shown in the following figure and are described below. The figure shows standard options for AASHTO LRFD.
Combined Cases

The following explanation applies to the standard solution method only.

In the vehicular loading options tab, the user specifies the vehicle type(s) to apply to the structure, and associated options. In the simplest case, a single vehicle type is applied to the structure. However, it is also possible to apply different vehicle types in each lane. This is explained later.

**Vehicle Load Pattern**

The load pattern to apply to the structure, from the load patterns in any connected databases (see Connecting Databases [in LARSA 4D User’s Manual]). The vehicle load pattern does not include lane loading.

**Factor**

For each load pattern, a factor may be applied to the magnitudes of the loads. The factor is applied to the vehicle axles only, and not any additional uniform lane loading specified.

To place two vehicles within a design lane, click the Add Vehicle button and then set the load pattern and factor of the second vehicle. (By default it will copy the settings of the first vehicle.)

The following options apply to all lane types (explained below):

**Min. Back-to-Front Spacing**

When multiple vehicles are placed in the same lane configuration, this is the smallest space allowed between the rear-most axle of one vehicle and the start of another (station zero in the vehicle definition).

**Design Lane Margin**

For influence surfaces, the margin on each side of design lanes in which vehicle loading is excluded. Thus for a 12 foot lane, a 1 foot margin means a possible loading area of 10 feet. This is depicted in the following figure:
Design Lanes and Vehicle Margin

One Load Pattern Per Span

The AASHTO LFD design code requires that in certain cases loads be applied only once per span. This option enforces that constraint. Be sure to add multiple load patterns to the lane configuration and to place *span break markers* in the lane path definition (page 99). Place span break markers between geometry points that fall on different spans in the structure. Span break markers should be placed in the row immediately following a geometry point positioned at the end of a span. For instance, if a span ends between members 3 and 4, a geometry point should be created either on member 3 but with an *x-offset of end* or, as is usually done, on member 4 but with the default *x-offset of start*. A span break marker then follows this row. Thus note that a span break marker usually follows a geometry point on a member that is actually on the next span. Span break markers must be added to the lane definition before the moving load analysis is performed.
Span Break Markers in the Lane Path Definition

The standard solution method supports the notion of a lane type. A lane type is a loading scenario. In AASHTO LRFD, for instance, there are three loading scenarios which can be entered as three lane types: 1) one design truck, 2) two design trucks with a factor, and 3) a design tandem. To create additional lane types after the first, click the Add Lane Type button and then set the Vehicle Load Pattern and Factor options for each vehicle entered within the new lane type.

The LARSA 4D influence vehicle loading algorithm will choose the lane type for each design lane that causes the most extreme effect. Each design lane may be assigned a different lane type.

Each lane type has two additional options:

**Max Lanes**
Some design codes require a special "permit" vehicle which may be placed on up to a single lane. This can be entered as an additional lane type, with Max Lanes for the lane type set to 1. Normally, this option is left as No Limit. Any positive integer may be used.

**UDL Factor**
Some design codes require the lane types to have different factors for the lane load. This field allows an additional factor to be applied to the lane load for design lanes assigned to this lane type. The factor is applied in addition to other factors relevant to the lane load.

**Lane Transverse Constraints Min/Max**
In some cases, vehicular and UDL loading must be constrained to a particular transverse location on the surface. In the Min field, provide the minimum distance from the negative-side edge of the surface (typically the right side) to the near edge of any design lanes of this lane type, in the length unit indicated. In the Max
field, provide the maximum distance from the negative-side edge of the surface (typically the right side) to the far-side edge of any design lanes of this lane type. If either Min or Max is zero or left blank, the corresponding constraint will not apply.

Vehicular Options for the 7.01 Compatible Solution Method

When using the 7.01 compatible solution method, the options for vehicular loading are entered differently. See Influence Surfaces in LARSA 4D Version 7.01 (page 221).

Uniform/Patch Loading Options

This set of options is for uniform/patch/lane loading.

Lane Load Method
The "simple" method indicates to place a uniform load throughout the lane surface, except in locations that do not contribute to the extreme effect (i.e. patch loading). This is the only method currently supported for the standard solution method.

Load Magnitude
This value is the magnitude of the uniformly distributed force. The magnitude is entered in force per linear length, per design lane, i.e. kip/ft per design lane.

Exclude Under Vehicles
Exclude the uniform lane load from the positions where vehicles are placed.

Procedure: Getting Results

Overview
The influence analysis result case can provide results in the same manner as any other load case: through the spreadsheet, graphically, or using any of the tools on the Results menu. The results presented reflect the positive and negative worst-case possibility for the application of the load pattern on the selected lane.

Because influence-based results are calculated on-the-fly (that is, when the actual data is requested), each data request involved calculating the extreme values. Influence-based results will operate slower than standard results, but they do not require additional analysis engine time or hard drive space. See Suggestions for Influence Analysis (page 219).

Envelopes
Because influence-based results search for a minimum and maximum, LARSA 4D must know which column from a result spreadsheet to envelope on. If LARSA 4D is enveloping on the member's z-moment, the member's corresponding y-moment will not be the minimum/maximum y-moment but instead will be the y-moment produced from the same axle positions that produced the greatest z-moment. In this way, an entire row of a spreadsheet comes from the same axle positions. You must use envelopes to view influence-based results so that LARSA 4D knows which value to envelope on.

Not all graphical results use envelopes automatically. The other results, such as member forces and stresses, automatically envelope on the result column being displayed. If axial force is being shown graphically, then the influence results will envelope on axial force.
**Special Graphical Results Tools**

A special influence coefficients graphical view [see “Influence Coefficients Graphical View” in LARSA 4D User’s Manual] is available, showing the influence coefficients as well as computed truck axle positions.

**Special Spreadsheet Results Features**

In results spreadsheets, the Result Case column of influence-based results indicates the specific loading configuration that yielded the extreme result of each row.

The cell value lists the coordinate of each placed vehicle in the form of “s4./t35.25”, for example, which means the head of the vehicle is 4 coordinate units from the start of the surface (“s” stands for “station”) and at a transverse location of 35.25 coordinate units (where zero is at the centerline of the surface and sign follows the right hand rule). The head of the vehicle is the (0,0) coordinate in the load pattern definition, which is typically at the location of the front wheels on the centerline of the vehicle.

If the configuration text is followed by “UDL”, it indicates that uniform lane loading was applied.

If the configuration text is followed by “* 1.2 (mp)”, a multiple presence factor (such as 1.2) was applied.

If the configuration text is followed by “* .9”, it indicates an overall factor was applied.

For More Information, please refer to the following documentation.

- Lanes on page 99.
- Influence Loads on page 127.
- Moving Load Analysis on page 203.
Suggestions for Influence Analysis

This section provides some suggestions for running influence analysis.

Suggestions for Speed

Because of the large number of computations involved in influence analysis, speed can be a concern.

**Using Larger Increments**

Try increasing the forward and transverse increments in the influence coefficients load case. With fewer coefficients generated, there are fewer results to scan through to optimize vehicle and UDL positions.

**Cache Results (Tools → Options → General Options)**

As influence based results are requested, the computed results are saved into a cache file. This way, the next time same influence result is requested, it can be retrieved from the cache quickly. Optimization occurs as user requests data.

**Pre-cache Results (Results → Pre-cache Results)**

You may want to use this tool before the leaving office so that it works during the night. The next day you can access all influence results just like static runs.

**I/O Optimization (Results → I/O Optimization)**

The first time LARSA tries to access a result file, the complete file will be loaded into system memory (assuming enough memory exist on the system). This way repeated disk access is avoided. The files are loaded into the memory as they are requested. This feature only loads original analysis engine generated result files, not the files generated by the above methods.
Influence Surfaces in LARSA 4D Version 7.01

This section describes options of an earlier version of LARSA 4D.

Design Lane Options

Since version 7.01, LARSA 4D can automatically determine the placement of design lanes on a roadway surface. Before this version, lane definitions were considered design lanes, and loading was applied accordingly. Since version 7.01, lane definitions can represent roadway surfaces, and the locations of design lanes will be found automatically.

In the 7.01 solution method there are three options for loading placement for influence surfaces which are chosen in the influence loading setup window:

**Place Vehicles Anywhere**

The lane definition is taken as a single design lane, with vehicles allowed to be placed anywhere on the lane. UDL lane loading is applied across the entire width of the lane. (This was the method always in effect before LARSA 4D version 7.01.) With this option, you may need to create multiple lane definitions and multiple influence cases to cover the different scenarios of live loads.

![Diagram of Place Vehicles Anywhere Option]

Place Vehicles Anywhere Option: The "lane" definition represents a single design lane.

**One Vehicle Per Lane**

The lane definition is considered a roadway surface, and the user must additionally specify the *Design Lane Width* and *Design Lane Margin*. With this option, trucks are loaded subject to a lane placement constraint. Only as many trucks listed in the *Max #* field will be placed, so the user must be sure to specify a number at least as large as the potential number of lanes that may fit on the roadway surface.

The lane placement constraint is that each vehicle must fall entirely within its own design lane, whose width is given as *Design Lane Width*, and cannot fall in a margin on each side of the lane (specified as the *Design Lane Margin*), meaning the width on either side of the lane that loading is excluded from), and of course lanes cannot overlap.

Any UDL lane load specified is applied only on the portion of the roadway surface that design lanes were placed at, excluding design lane margins. The UDL magnitude is applied per design lane, which means that unlike with the previous option, the magnitude of the UDL per liner foot across the entire roadway surface will depend on the number of design lanes automatically placed.
One Vehicle Per Lane: The "lane" definition represents a roadway surface. Design lanes are placed automatically on the surface, with one vehicle per design lane.

Two Vehicles Per Lane

This option works similarly as with One Vehicle Per Lane, except that at most two vehicles are placed in each lane. As above, only as many trucks listed in the Max # field will be placed, so the user must be sure to specify a number at least twice the potential number of lanes that may fit on the roadway surface. The placement of the second truck on any lane is constrained to be at the same transverse offset as the first truck.

Vehicle Type and Placement Options

Enter the total set of vehicles to be applied to the structure. Note that while in the standard solution method each type of vehicle is entered once, with the 7.01 compatible solution method a vehicle must be entered for as many times as it could be applied to the structure. On a three-lane span, the vehicle must be entered either three times or once but with the Max # field set to 3.

The UDL factor is not applicable. The design lane vehicle margin option is applicable with the one- or two-vehicles-per-design-lane option. The following additional option is available:

Min. Side-to-Side Spacing

For influence surfaces only, the smallest space allowed between the sides of two nearby vehicles, as determined by the (transverse) distance from the centerline of the leftmost load of one vehicle to the centerline of the rightmost load of another.

The one vehicle per span option has a different meaning with this solution method. See AASHTO LFD Point Loading in LARSA 4D Version 7.01 (page 223).
AASHTO LFD Point Loading in LARSA 4D
Version 7.01

This section describes options for a previous version of LARSA 4D.

The AASHTO LFD Point Loading option adds extra processing to the influence case for use with the AASHTO LFD provision that when applying point loads to determine negative moments, a second point load of equal magnitude is to be placed on a different span from the first point load.

To use this option, *span break markers* must be added to the lane path definition (page 99). Place span break markers between geometry points that fall on different spans in the structure. Span break markers should be placed in the row immediately following a geometry point positioned at the end of a span. For instance, if a span ends between members 3 and 4, a geometry point should be created either on member 3 but with an *x-offset of end* or, as is usually done, on member 4 but with the default *x-offset of start*. A span break marker then follows this row. Thus note that a span break marker usually follows a geometry point on a member that is actually on the next span. Span break markers must be added to the lane definition before the moving load analysis is performed.

![Lane Path Definition](image)

Span Break Markers in the Lane Path Definition

Then set up an influence case with two single-point-load load patterns and check the *AASHTO LFD Point Loading* option, as shown in the figure below.
Results produced by this case will reflect the application the first point load always, and in addition the second point load when negative moments in Member End Forces or Member Sectional Forces are being viewed.

Here, 'negative moments' is taken to mean the moments at the top of horizontal members, with the global-z axis being 'up'. For any given member, negative moments are applicable to either the Moment Y or Moment Z column, and either the envelope minimums or the envelope maximums. Which column and extreme is used depends on the orientation of the member and the sign convention of the result type, according to the following table:

<table>
<thead>
<tr>
<th>Member Orientation</th>
<th>Orientation Angle</th>
<th>Negative Moments Are...</th>
<th>Sign Used for End Forces</th>
<th>Sign Used for Sectional Forces</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local-Y is Up</td>
<td>90 (or close to 90)</td>
<td>Moment Z</td>
<td>Maximums at Start,</td>
<td>Minimums</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Minimums at End</td>
<td></td>
</tr>
<tr>
<td>Local-Y is Down</td>
<td>270</td>
<td>Moment Z</td>
<td>Minimums at Start,</td>
<td>Minimums</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Maximums at End</td>
<td></td>
</tr>
<tr>
<td>Local-Z is Up</td>
<td>0</td>
<td>Moment Y</td>
<td>Minimums at Start,</td>
<td>Maximums</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Maximums at End</td>
<td></td>
</tr>
<tr>
<td>Local-Z is Down</td>
<td>180</td>
<td>Moment Y</td>
<td>Maximums at Start,</td>
<td>Maximums</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Minimums at End</td>
<td></td>
</tr>
</tbody>
</table>
Various standard vehicle types are supplied with LARSA 4D. These vehicles include AASHTO (American Association of State Highway and Transportation Officials) trucks, AASHTO alternate military loads, CALTRANS P type permit loads and Indian Roads Congress vehicles.

AASHTO and CALTRANS Vehicles

The AASTHO H loadings consist of a two-axle truck, the HS loadings consist of a tractor with semi-trailers, and CALTRANS P loadings are special vehicular loads. These loads are found in the file movedata.dml.

The patterns are one dimensional: Axles are point loads applied to the center-line of lanes.

- HS20-44 is an AASHTO HS truck consisting of a tractor truck with a semi-trailer. The total weight is 72 kips.
- HS15-44 is an AASHTO HS truck with 75 percent loading of HS20.
- H20-44 is an AASHTO H truck consisting of a two-axle truck. The total weight is 40 kips.
- H15-44 is the Standard H truck with 75 percent loading of H20.
- H20-S16-T16 consists of five axles with total weight of 104 kips.
- HN-72 is a highway normal truck with total weight of 54 kips.
- HO-72 is a highway overweight truck with total weight of 108 kips.
- LEGAL-TYPE3 is an AASHTO Legal Load Type 3 with three axles and the total weight is 50 kips.
- LEGAL-TYPE3S2 is an AASHTO Legal Load Type 3S2 with five axles and the total weight is 72 kips. Input Data Description 9-137
- LEGAL-TYPE3-3 is an AASHTO Legal Load Type 3-3 with six axles and the total weight is 80 kips.
- P5, P7, P9, P11 and P13 are CALTRANS-P loads (permit design live loads). The distance between each axle is 18 feet. The first axle is 26 kips and the remaining axles are 48 kips. The vehicles have 3, 4, 5, 6, and 7 axles, respectively.

AASHTO Load Patterns for Influence Analysis

These load patterns are provided primarily for use in influence-based analysis following the AASHTO LRFD design code for live load analysis. Some are defined with variable axle spacing, which only applies in that type of analysis. These patterns are found in the files movedata.dml or AASHTO Vehicle Patterns.dml.

Some of these patterns are two dimensional. 2D load patterns can only be used with influence-based analysis.

- HL-93/HS20-44 Design Truck has three axles of 8, 32, and 32 kips. The spacing between the front two axles is 14 feet, and in influence-based analysis the spacing between the rear axles varies between 14 and 30 feet. This load pattern does not specify a width for the vehicle, so should not be used in influence surface analysis without noting the implications for vehicle placement and the side-to-side spacing option.
- HL-93 Design Tandem consists of two 25 kip loads spaced 4 feet apart. This load pattern does not specify a width for the vehicle, so should not be used in influence surface analysis without noting the implications for vehicle placement and the side-to-side spacing option.
• HL-93/HS20-44 2D Design Truck has six wheels, modeled as point loads offset 3 feet each way from the vehicle center. It has the same spacing, including variable axle spacing, as the one-dimensional HL-93 pattern above, except that the axles are split into pairs of wheels.

• AASHTO Point 18 kip moment/26 kip shear/13.5 kip moment/19.5 kip shear are point load patterns with the given type and magnitude. They are provided for using the influence-based analysis with span loading in the AASHTO code.

There are two vehicle load patterns that model tire contact area with distributed loads. Tire contact area is specified as uniformly distributed loads over a rectangular surface at the appropriate transverse coordinates.

• HL-93 Design Truck T/C is similar to the HL-93 Design Truck, except that tire contact area is more precisely modeled by specifying two tires per axle at the appropriate transverse positions, and by including transverse width and tire length. This pattern does not include variable axle spacing, however, and tire contact area rarely plays a role, so the HL-93/HS20-44 2D Design Truck Pattern is recommended to be used in place of this pattern.

• HL-93 Design Tandem T/C or HL-93 Design Tandem 2D is similar to the HL-93 Design Tandem, except it has four loads, two for the front axle and two for the back, each 12.5 kips distributed over a 10x20 inch area. The tires are laterally offset from the center-line by 3 feet.

Other load patterns are available but are no longer recommended for use:

• HL-93 Design Two Trucks consists of two HL-93 design trucks at variable spacing, but each truck having fixed axle spacing at 14’. The use of this load pattern is deprecated in favor of using the number of vehicles to load and minimum vehicle spacing options of the influence-based analysis.

IRC Load Patterns for Influence Analysis

These load patterns are provided primarily for use in influence-based analysis following the Indian Roads Congress design code for live load analysis. These patterns are found in the file IRC Load Patterns.dml.

These patterns are all two dimensional: Rather than specifying axles, the actual wheels of the load patterns are specified (generally two per axle) at the appropriate transverse position. In patterns marked "Distributed," tire contact area is specified as uniformly distributed loads over a rectangular surface. Those not marked "Distributed" have wheels modeled as point loads. Distributed patterns are provided for the influence surface analysis.

When used in the standard moving load analysis, transverse positions of wheels are ignored. All wheels are lumped on the center-line of the lane.

• IRC Class AA Tracked is the Class AA tracked tank pattern, comprising two 350 kN distributed loads 3.6 meters long and .85 meters wide, offset by 1 meter in each direction from the lane center-line.

• IRC Class AA Wheeled is the Class AA wheeled tank pattern, comprising two axles with four wheels per axle of 37.5 and 62.5 kN.

• IRC Class 70R Tracked is the Class 70R tracked tank similar to the Class AA tracked tank but with a length of 4.57 meters.

• IRC Class 70R Load Train is the 7-axle train with load intensities of 40, 60, and 85 kN per wheel (two wheels per axle).

• IRC Class 70R Bogie is the bogie pattern with two axles comprising four wheels of 100 kN each.

• IRC Class A Train is the 8-axle train with load intensities of 13.5, 57, and 34 kN per wheel (two wheels per axle).
• IRC Class B Train is the 8-axle train with load intensities of 8, 34, and 20.5 kN per wheel (two wheels per axle).

For More Information, please refer to the following documentation.
• Moving Load Analysis on page 203.
Staged Construction Analysis

Staged Construction Analysis analyzes a series of construction steps, which represent the steps in the evolution of a structure's construction, demolition, or rehabilitation. The Staged Construction Analysis is a form of nonlinear static analysis that retains the state of the structure from step to step, automatically applying construction activities specified by the user. Staged Construction Analysis makes scenario testing an automated process.

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overview of Staged Construction Analysis</td>
<td>231</td>
</tr>
<tr>
<td>Staged Construction Activities</td>
<td>233</td>
</tr>
<tr>
<td>Setting Up the Model</td>
<td>235</td>
</tr>
<tr>
<td>Preparation</td>
<td>235</td>
</tr>
<tr>
<td>Creating Activities in the Explorer</td>
<td>236</td>
</tr>
<tr>
<td>Creating Activities in the Stage Editor</td>
<td>238</td>
</tr>
<tr>
<td>Time Effects on Materials</td>
<td>243</td>
</tr>
<tr>
<td>Material Time Effects</td>
<td>243</td>
</tr>
<tr>
<td>Design Codes</td>
<td>245</td>
</tr>
<tr>
<td>Load Class Tracking</td>
<td>247</td>
</tr>
<tr>
<td>Setting up the model</td>
<td>247</td>
</tr>
<tr>
<td>The Load Classes of Result Cases</td>
<td>247</td>
</tr>
<tr>
<td>Accessing Class-Based Results</td>
<td>248</td>
</tr>
<tr>
<td>Staged Construction Analysis Options</td>
<td>249</td>
</tr>
<tr>
<td>Options for Analysis Scenarios</td>
<td>250</td>
</tr>
</tbody>
</table>
Overview of Staged Construction Analysis

Changes to a structure over time are defined in a series of construction steps. Construction steps can include construction activities, such as constructing parts of the structure, applying loads, and modifying support conditions. Time-effects on material behavior, including the effect of super-imposed loads, can be automatically accounted for.

Because the Staged Construction Analysis is a nonlinear analysis, geometric nonlinearity is included, a necessary feature for any structure with nonlinear elements, such as cables.

Staged Construction Analysis comes in two types. In the Standard type, a Nonlinear Static Analysis (page 165) is performed for each construction step, retaining the deformed properties of the structure from step to step. Time-effects on materials are ignored. In the Time-Dependent type, the same nonlinear analysis is performed, but time-effects on materials are included.

Elements are activated and deactivated through their inclusion in construction activities. When a Staged Construction Analysis begins, LARSA assumes no elements have been activated. Only elements explicitly constructed in construction activities contribute to the stiffness of the structure. If the structure is to start off constructed, the first construction activity should assemble the structure into its constructed state.

A structure can change over time any number of ways in the real world. LARSA supports the following construction activities:

- **Construction**: Parts to the structure are added (assembled). Joint locations can be automatically adjusted based on the deformed shape of the structure according to construction methods, a necessary tool for segmental projects such as balanced cantilever bridges.
- **Deconstruction**: Parts of the structure are removed (disassembled).
- **Weight and Stiffness**: Self-weight and stiffness can be activated in different steps.
- **Supports and Constraints**: Joints are restrained or released, and constraints can be added/removed.
- **Loads**: Loads are applied or removed.
- **Tendons**: Stressed or slackened.
- **Materials**: Creep and shrinkage for concrete, and relaxation for steel.

Staged construction often comes hand-in-hand with pre- and post-tensioning elements. Such tensioning is generally accomplished with tendons, which are discussed in the tendons section in the Model Data Reference (page 89).
Staged Construction Activities

LARSA is able to model continuous changes to a structure over time involving a useful set of construction activities.

Construction activities are specified in two ways, using the Construction Stages Explorer or the Construction Stages Editor. See Setting Up the Model (page 235).

LARSA arranges construction activities into stages and steps.

- **Stages**
  A stage represents one day of construction, which can consist of one or more steps. Stages are labeled with a day number, a temperature and a humidity value, which apply to all steps within the stage. The day of the stage is used for computing time-effects on materials.

- **Steps**
  A step represents one set of construction activities to accomplish a goal. Steps occur within stages. All construction activities occur within steps.

Both stages and steps can be given descriptive names.

The construction activities that can be placed within steps are given below. They are described in more detail in Construction Activities (page 131).

**Construction/Deconstruction**

Structural elements are at any point constructed or unconstructed. Constructed elements are used in the analysis of the structure, while unconstructed elements are ignored. Elements start off unconstructed until they are explicitly constructed in a step.

To construct or deconstruct (page 133) a part of the model, form a structure group [see “Structure Groups Explorer” in LARSA 4D User’s Manual] and then drag that group from the Structure Groups Explorer into the Construction Stages Explorer [in LARSA 4D User’s Manual]. The elements in that group will be constructed. To deconstruct those elements, right-click the stage, choose Properties, and mark the Deconstruct option.

Elements may be constructed in locations relative to the deformed state of the structure. The segmental construction methods (page 133) and joint displacement initialization (below) are used to specify such behavior. Normally, elements are constructed in the position that they are initially defined in.

See Construct and Deconstruct Activities (page 133).

**Support Change and Hoist Activity**

A support's properties have changed, or a joint is becoming a support. Support activities (page 139) specify the new restraints of a joint (page 55) in all six degrees of freedom.

See Support and Hoist Activities (page 139).

**DOF Constraint Activity**

A joint (page 55)'s DOF Constraint (page 87) is changed. DOF Constraint activities specify the new DOF Constraints of a joint. Old DOF Constraint connections in all degrees of freedom are discarded for the joint whenever a new activity for that joint is encountered.
See DOF Constraint Change Activities (page 141).

**Load Application**

A static load case (page 107) is applied to the structure. Loads are applied cumulatively; that is, a load applied in one stage will automatically continue to be applied in all future stages. To "turn off" a load, apply the load again with a negative load factor.

**Tendon Application**

A tendon (page 89) is stressed or slackened.

See Tendon Stressing and Slackening Activities (page 143).

**Displacement Initialization**

Alters the initial location of a joint. Normally, when joints are activated they enter at the location that was initially given to them. Joints sometimes need to become active in a location relative to the deformed structure of the model, rather than in an exact position known ahead of time. Displacement Initializations specify how to place a joint relative to the deformed location of other joints.

See Displacement Initializations (page 145).
Setting Up the Model

Construction activities are specified in two ways, using the Construction Stages Explorer or the Construction Stages Editor. This section explains how to use each method.

Structural changes occur in a serial or linear fashion. An example of staged construction might be . . .

1. Construct Left Pier
2. Construct Right Pier
3. Construct Center Temporary Pier (modeled as fixing center joint in space)
4. Construct Span from Left to Temporary Pier
5. Construct Span from Right to Temporary Pier
6. Stress Tendons
7. Deconstruct Temporary Pier (modeled as releasing center joint)

Each of those steps is referred to as a *construction activity*.

Preparation

Before defining construction activities in LARSA,

1. All elements in the structure that will ever be assembled must be modeled.

   Structures should be modeled as if every member, plate, and spring used at any point during construction are all present at the same time. The time at which elements are constructed will be specified later.

   As a result, all joints that will ever be a part of the model must be in the project. Joints are automatically included in the model when elements attached to it become constructed and are automatically removed from the model when all attached elements are deconstructed.

   Tendons (page 89) and other structural objects must also be entered before proceeding on to modeling construction activities.

2. Restraints must be set to initial conditions

   Joints will be initially restrained as their restraints are specified in the project, taking into account standard restraints (page 55) and DOF constraints (page 87).

3. Groups of elements that will be constructed or deconstructed together should be put into structure groups [see “Structure Groups Explorer” in *LARSA 4D User’s Manual*].

   Although structure groups need not be defined ahead of time, it will save time later.

The steps outlined from this point on assume that the above tasks have been completed.
Creating Activities in the Explorer

The Construction Stages Explorer presents the construction stages and steps in the same style as the example above. A screenshot is on the right.

For more information on the meaning of each type of construction activity, see Staged Construction Activities (page 233).

Using the Explorer:
• To open the Explorer if it is not already open, right click the title bar area of the open Explorer (where it reads "Construction Stages" in the figure) and choose Construction Stages.

• To **add a construction stage**, click the Add Stage button, which looks like a folder. A new stage will be added to the bottom of the list of stages. Each stage is labeled with the day on which its steps' activity occurs, which is "day 0" by default.

• To **add a construction step** to the project, select the stage in which to add the new step, and then click the Add Step button, which looks like a small frame.

• To **delete** a stage or step, select the stage or step and then click the Delete button, which looks like an X.

• To **rename** a stage or a step, select the stage or step, and then hit **F2**. Type the new name, and then hit **Enter**.

• If after adding various stages they have become out of **chronological order**, click the Sort button. The stages will be sorted by Day.

Preparing for Time Effects in the Explorer:

• To include **time-effects on materials** in construction analysis, each construction stage must have a time assigned, determined by a stage's Day. To edit a stage's day, select the stage. Then, make the change in the text box at the bottom of the explorer. Hit the checkmark to finalize the change.

• To modify the **humidity or temperature** of a stage, which is used to compute time-effects on materials in construction analysis, open the Load Groups & Stages spreadsheet window, which can be found in the **Input Data** menu. Select the Construction Stages tab. This spreadsheet will show a list of the stages added to the project, but not the construction steps.

Adding Construction Activities in the Explorer:

• If part of the structure is to be **constructed or deconstructed** in a step, click and drag a Structure Group [see “Structure Groups Explorer” in LARSA 4D User’s Manual] of those elements from the Structure Groups Explorer onto the step in the Construction Stages Explorer during which they will be constructed.

To see the two Explorers at once, double-click the title bar area of the open Explorer (where it reads "Construction Stages" in the figure). The right click the title area of the Explorer on the bottom and choose Structure Groups. Double click the title area of the top Explorer again to close the lower Explorer.

Structure groups in a step must be either all constructed or all deconstructed. Use multiple steps if both construction and deconstruction will occur.

If that structure group is to be **deconstructed**, right-click the construction step (not the group). Click **Properties** and choose the Deconstruct option. The step will become a deconstruction step, indicated by a red X through the step's icon.

Structure groups can be constructed in **weight-only mode** in which the self-weight of the group is applied to the structure but the group adds no stiffness to the structure. This is useful when a concrete member is poured but not immediately hardened. To specify that a structure group be weight-only, right click the group and select **Weight Only**. To have the group contribute stiffness in a later stage, add the group to the Construction Stages Explorer again, but without the weight-only option.

• **Static load cases** are be applied during a step by dragging a load case from the Load Cases Explorer [see “Load Cases Explorer” in LARSA 4D User’s Manual] onto the step in the Construction Stages Explorer during which the load case should be applied. To see the two Explorers at once, see the previous point.
• **Other activities** are accessed
• Using the Construction Stage Editor, which is explained below, or
• Through spreadsheets [see “Using the Model Spreadsheets” in *LARSA 4D User’s Manual*]. To add these activities to a step, right-click the step and then select [Support, DOF Constraint, Tendon Activity].

Creating Activities in the Stage Editor

The construction stage editor is an alternative way of building/editing the stage construction data in LARSA 4D. Unlike the Construction Stages Explorer, the Construction Stage Editor uses a spreadsheet based approach to organize the data.

The spreadsheet does not only contain the data but also some commands with a self explanatory label to their functionality. These commands can be easily distinguished by their red, italic, underlined format. Double clicking on these commands will activate them.

Each spreadsheet contains the commands to add, remove, and rename stages and steps.

The dialog box is composed of six tab menus. Each menu represents a type of spreadsheet. Below you find the explanation on each tab menu and how it is being used.

For more information on the meaning of each type of construction activity, see Staged Construction Activities (page 233).

On the **Stage/Steps** spreadsheet, you can:
- Add, remove, and rename stages and steps,
- Specify whether each step is for construction or deconstruction, and
- Define the segmental construction method (page 133) of each step. Unless your model needs to update joint locations to account for deformation, the Standard method should be chosen.
On the **Load Cases/Structure Groups** spreadsheet, you can:

- Add and remove load cases to steps,
- Add and remove geometry groups to steps, and
- Specify whether geometry groups are weight-only. (For an explanation of weight-only groups, see the previous section.)

On the **Support Activity** spreadsheet, you can add or remove support activity for each step by specifying the joint number and six degrees of freedom.
On the **DOF Constraint Activity** spreadsheet, you can add and remove DOF constraint activity for each step by specifying the joint number and the master joint at six degrees of freedom.

On the **Tendon Activity** spreadsheet, you can

- Add and remove tendon activity for each step, and
- Specify whether each tendon is stressed or slackened.
On the **Displacement Initialization Activity** spreadsheet, you can add and remove displacement initialization activities for each step by specifying the joint number that the initialization process will take place, the joints at both ends, and the adjustment direction.
Time Effects on Materials

Time effects on materials account for concrete creep and shrinkage and the time-effect on elastic modulus, steel relaxation, and other material changes.

Conventional analysis does not account for materials whose properties change over time. Concrete is subject to creep and shrinkage. Steel is subject to relaxation. These changes and elastic modulus variation are accounted for in LARSA 4D's Time-Dependent Stage Construction Analysis.

Time-dependent deformation of concrete resulting from creep and shrinkage is of crucial importance in the design of prestressed concrete structures because these changes result in a partial loss of prestress force with significant changes in deflections.

Time-Dependent Stage Construction Analysis also requires time, temperature, and humidity conditions at each stage.

To set other options, including the design code, open the Analysis and choose Time-Dependent Analysis Options.

Material Time Effects

The following time-dependent material effects can be computed by LARSA 4D. Each effect can be turned on or off in the Time-Dependent Analysis Options.

Concrete Creep

Creep is the property of many materials by which they continue deforming over considerable lengths of time at constant stress or load. Creep strain for concrete depends not only time, but on the mix proportions, humidity, curing conditions and the age of concrete when it is first loaded. The rate of strain increase is rapid at first, but decreases with time until a constant value is approached asymptotically. In LARSA 4D, creep is available for member and plate elements and includes creep due to axial force and the two bending moments.
Creep is applied to all member and plate elements with materials that have a *concrete cement hardening type* set to anything except *Not Concrete*. (See Materials (page 19).)

The volume-to-surface-area ratio is computed for member elements with the assumption that the entire perimeter of all shapes in the element cross-section are exposed to air but the end faces of the element are not exposed to air (e.g. a column-like layout of beam elements). It is computed for plate elements with the assumption that the top and bottom surfaces of the plate element are exposed to air but the side faces along the thickness of the plate are not exposed to air (e.g. a slab-like plate mesh).

Creep in member elements due to incremental tensile strain is ignored unless *Include creep of tensile axial force for beams* is turned on. This option should be enabled when member elements have composite action.

Creep is not applied if *Include Creep* is turned off.

**Concrete Shrinkage**

Normal concrete mixes contain more water than is required for hydration of the cement where the free water evaporates in time. Drying of concrete is accompanied by a reduction in volume. The rate of drying depends on the humidity, ambient temperature, and the size and shape of the concrete element and the change in volume occurs at a higher rate initially.

Shrinkage is applied to all member and plate elements with materials that have a *concrete cement hardening type* set to anything except *Not Concrete*. (See Materials (page 19).)

See the explanation for creep for how the volume-to-surface-area ratio is computed for member and plate elements.

It is assumed that the elements undergoing shrinkage represent pre-cast segments. Consequently, shrinkage that occurs before the day of construction is not applied to the model.

Shrinkage is not applied if *Include Shrinkage* is turned off.

**Time Effect on Modulus of Elasticity**

The strength of concrete varies significantly from initial days to day 28, although from day 28 on the variation is very slow. This variation of strength is caused by changes to the elastic modulus. LARSA 4D can vary the elastic modulus of concrete materials according to several codes or a user-defined curve.

If a user-defined curve is not entered, LARSA 4D uses the formula:

\[
\left( \frac{t}{(4.0+0.857t)} \right)^{0.5}
\]

where \( t \) is the age of the element in days since its casting day. This curve has a value of 1.0 on day 28.

This effect is not applied if *Include Time Effect on Modulus* is turned off.

**Steel Relaxation**

When prestressing steel is stressed to the levels that are customary during initial tensioning and at service loads, it exhibits a property known as relaxation. Relaxation is defined as the loss of stress in a stressed material held at a constant length. This same phenomenon is known as creep when defined in terms of change in length of a material under constant stress. In prestressed concrete members, creep and shrinkage of the concrete as well as fluctuations in superimposed loads cause change in tendon length. Relaxation is not a short-lived phenomenon, it continues almost indefinitely, although at a diminishing rate.
Relaxation must accurately be accounted for in the design because it produces significant loss of prestress force. The amount of relaxation varies depending on the type and grade of steel, but most significant parameters are time and intensity of the initial stress.

Prestressing tendons are held stressed at essentially constant length during the lifetime of a member, although there is some reduction in length due to concrete creep and shrinkage and superimposed loads. There will be a gradual reduction of stress in the steel under these conditions resulting from relaxation, even though the length is held nearly constant.

The amount of relaxation depends on the intensity of steel stress as well as time and, for stress-relieved steel, the ratio of stress to initial stress.

Relaxation curves (explained above) must be entered into the project and assigned to any material that will undergo relaxation, and those materials must be assigned to the tendons.

Relaxation is not applied if Include Steel Relaxation is turned off.

**The Effect of Creep, Shrinkage, and Superimposed Loads on Post-Tensioning**

Concrete creep and shrinkage and other loads applied after a tendon is stressed (superimposed loads) generate strain, which alters the forces in the post-tensioning.

This effect is not applied if Losses due to Creep/Shrinkage/SI Loads is turned off.

**Design Codes**

The following design codes are available for time-dependent material effects:

**CEB-FIP 78**

This code uses a user-defined material model based on the CEB-FIP 78 guidelines. Use this option to supply custom curves for special environmental conditions. This code requires the following time-dependent material curves (page 49) to be entered by the user:

- Concrete Shrinkage Curve
- Concrete Delayed Plastic Strain Curve
- Concrete Delayed Elastic Strain Curve
- Stress/GUTS vs. Relaxation Curve
- Time vs. Relaxation Curve
- Time versus Elastic Modulus Curve.

**CEB-FIP 90**

This code requires the following time-dependent material curves (page 49) to be entered by the user:

- Stress/GUTS vs. Relaxation Curve
- Time vs. Relaxation Curve

This code has built-in equations for the computation of creep and shrinkage coefficients and therefore does not require the following curves to be entered by the user:

- Concrete Shrinkage Curve
- Concrete Delayed Plastic Strain Curve
• Concrete Delayed Elastic Strain Curve
• Time versus Elastic Modulus Curve

Additionally, TS is set to 3.

**CEB-FIP 2010**
TS is set to 3.

**EN1992-1-1:2004(E) (Eurocode)**
TS is set to 14.

**AASHTO LRFD v6**

Note that the ts constant, the age at which shrinkage begins, is set to zero for this code.
Load Class Tracking

Code compliance requires creating factored load combinations, with different factors applied to different classes of loading, such as dead load, live load, creep, and shrinkage. The Staged Construction Analysis of LARSA 4D reports cumulative effects after each construction step, but it is also possible to extract the partial effects of each load class. Starting with 7.01.60, the partial class-based effects of support, DOF constraint, and element deconstruction activities are now available, and these are crucial for creating code-based load combinations in a Staged Construction Analysis.

We discuss below setting up a model for tracking class-based loading in Staged Construction Analysis and how this method applies to support changes and element deconstruction.

**Setting up the model**

In a Staged Construction Analysis model that will use class-based load tracking, it is necessary to ensure that each load case is assigned a load class. Load classes can be chosen from a pre-set list of 45 classes including "Dead Load", "Live Load", "Creep", "Snow Load", and "Prestress". The choice of class does not impact the analysis and is used only for reporting partial effects during post-processing. To choose a load class for a load case, use the Load Cases & Combinations Spreadsheet or the Load Cases Explorer [see “Load Cases Explorer” in LARSA 4D User’s Manual]. In the Load Cases Explorer, right-click on a load case, choose Properties, and select a load class. Don't forget to set the "Dead Load" load class to load cases for self-weight.

When setting up construction stages and steps, do not group activities together into a single step if they are assigned different load classes. For example, dead load and live load cases should be placed into separate construction steps. Element construction and support addition activities do not add forces to the model and can be grouped with any other activity. Tendon stressing activities are analyzed separately from other activities and are automatically assigned the Prestress Load class in results.

Three other types of activities redistribute existing forces in the structure: support removal, DOF constraint removal, and element deconstruction. Each of these activities must be handled properly if load class tracking is needed. Because a reaction force may be the result of loading from several load classes, when the support and reaction are removed the effects on the structure must be attributed correctly to the load classes that created the reaction force. For example, if Dead Load is responsible for a reaction at a support, then when the support is removed the resulting deformation of the structure must be attributed to Dead Load. If loading from several load classes has contributed to the reaction, LARSA 4D uses an incremental balanced removal procedure that generates a separate result case during the analysis for each load class’s component of the reaction, allowing the effect of each load class to be included in factored combinations later. DOF constraint removal and element deconstruction are handled similarly.

Before running a Staged Construction Analysis, activate load class tracking in the analysis window by changing to the Staged Construction Options tab and selecting "Enable Class-Based Load Tracking." If this option is not turned on, redistributed forces from the construction activities mentioned above will be assigned to the "none" load class or will be grouped with loading from a load case applied within the construction step.

**The Load Classes of Result Cases**

In analysis results, each result case is assigned a single load class. You can check which class a result case is marked as by clicking the case in the Analysis Results Explorer [see “Analysis Results Explorer” in LARSA 4D User’s Manual] and then looking at the information box below the explorer.
For result cases that correspond directly to a construction step in the input data, the load class will be the same as the load class of any load cases applied in the construction step. If a self-weight load case marked as the Dead Load class is applied in a construction step, the results for that step will be marked as Dead Load as well. If the construction step contained only element construction and support additions but no load cases, the results will have the "none" load class for that step.

LARSA 4D will also generate special result cases. "PT Activity" result cases are generated for construction steps in which tendons are stressed: these load cases are marked as "Prestress Load" automatically. In the Time-Dependent Staged Construction Analysis, special Creep, Shrinkage, PT Relaxation, and Other PT Losses cases are created. These cases are assigned to separate load classes for these effects.

When a construction step contains a support or DOF constraint removal or element deconstruction, LARSA 4D automatically redistributes forces separately by load class, as described above. When this occurs, separate result cases will be generated for each load class that contributed to the element internal forces. By creating separate result cases, the partial effects of each load class in the support, DOF constraint or element removal can be inspected.

### Accessing Class-Based Results

LARSA 4D has several results tools which make use of load classes.

In spreadsheets [see “Results Spreadsheets” in LARSA 4D User’s Manual] and graphical results [see “Viewing Results Graphically” in LARSA 4D User’s Manual], the user has the option to view 1) full cumulative results, meaning the total effect on the structure of all loading and activities up to the construction step selected, 2) incremental results, meaning the changes since the immediately preceding result case, and 3) load-class extracted results, which reports the partial effect due to a single load class up to the construction step selected, by adding together the incremental effects of all construction steps for that load class up to and including the selected construction step. See Incremental and Cumulative Results [in LARSA 4D User’s Manual].

Factored combinations of class-based results can be created with Linear Result Combinations [see “Linear Result Combinations” in LARSA 4D User’s Manual], found in the Results menu. Using Linear Result Combinations, one can combine both the full cumulative results reported by a result case or the load-class extracted results. Full cumulative results are normally used to combine separate result cases of a basic static analysis. In Staged Construction Analysis, one can use load-class extracted results to make a factored combination of the partial effects at the last construction step. Here, one factors the extracted effects of different load classes but for the same result case. For instance, one can combine the dead load at the last step of construction with the prestress losses at the last step of construction, or at any other step of construction.

The Auto Load Combinations [see “Automatic Code-Based Load Combinations” in LARSA 4D User’s Manual] tool in the results menu can be used to create linear result combinations automatically for code-specified combinations by selecting from the available results cases for each of the load classes referred to by the code.
Staged Construction Analysis Options

If you will be including time-effects on materials, please read the section Time Effects on Materials (page 243) to see how to set those options.

For the difference between the Standard and Time Dependent analysis types, see Overview of Staged Construction Analysis (page 231).

Because the Staged Construction Analysis is a derivative of the nonlinear static analysis, the analysis shares some of the same options, which are explained below. In addition, the Standard analysis may perform plastic pushover analyses at certain stages. Plastic pushover's auto-increment options for those steps are explained below.

Construction Stages

Choose which construction stages are to be analyzed. The analysis can only pick up from any stage up to where it last left off, or from the start.

Starting Construction Stage
The stage that the analysis will begin at.

Ending Construction Stage
The stage that the analysis will stop at.

Staged Construction Options

Choose whether geometric nonlinear will be included in the analysis.

Included Geometric Nonlinearity
Whether geometric nonlinearity will be included in the analysis. For models involving tendons, this option should be unchecked as the equivalent loads used to represent tendons do not currently follow the deformation of the model. For models involving cables, this option must be selected.

Enable Class-Based Load Tracking
Class-based load tracking is enabled so that support change and deconstruction activities apply the proportion of the reactions and element internal forces back onto the structure grouped under the load class in which the forces were created. If this option is checked, result cases will be generated as needed for each load class involved in each construction step. If this option is not checked, class-based load tracking is not enabled and support change and deconstruction activities cause reactions and element internal forces to be applied back into the model under the load class set for the deconstruction step.

Analyze Load Cases Incrementally
Each load case applied within a construction step is analyzed independently so that the structure can be loaded incrementally by load case which may improve solution convergence. Using this option may slow down the analysis.
Solution Convergence

Equilibrium iterations at a given load level can cease when the result is "close enough" according to one or more criteria. Two criteria used in LARSA are that the unbalanced force be a small fraction of the total applied force in the current load level and that the current displacement increment be a small fraction of the displacement increment. The convergence criteria are specified by the user as the displacement tolerance, force tolerance and maximum number of iterations. The iterative analysis continues until all degrees of freedom in the model satisfy displacement and force tolerance criteria within the maximum number of iterations.

If the solution is not convergent for a load step, a message is displayed on the screen. If there are additional load steps, analysis will continue if the structure is not unstable. LARSA carries the unbalanced forces into the next load step and a convergent solution may be obtained in the next load step. This may imply that your structure is fine and you either have to increase the maximum number of iterations or use a less restrictive error ratio.

Max # of Iterations
The number of iterations performed for each load step will not exceed the maximum number of iterations specified by the user. We recommend 10 to 20 as a maximum number of iterations.

Displacement Tolerance
The displacement tolerance is the error ratio of the incremental displacement computed in the last iteration divided by the total displacement for the same degree of freedom. The default value is 0.001 (0.10%).

Force Tolerance
The force tolerance is the maximum unbalanced force at any degree of freedom.

Options for Analysis Scenarios

The options below are available for pushover analysis scenarios, i.e. pushover analysis steps within Staged Construction Analysis.

Auto-Increment Options

The pushover analysis is based on automatically incrementing the load vector. Specify here the multipliers for the load vector that will be used in computing the initial load vector and the incremental load vector limitation.

Initial
This is the factor multiplying the load vector for computing the load to be applied on the structure in the first load step.

Minimum
The minimum factor multiplying the load vector that is used in any load step. When the iterative analysis requires reduction in the magnitude of the load vector, the multiplier cannot be smaller than this value.

Maximum
The maximum factor multiplying the load vector that is used in any load step.

Auto-Increment Options

The pushover analysis is based on automatically incrementing the load vector. Specify here the multipliers for the load vector that will be used in computing the initial load vector and the incremental load vector limitation.
**Initial**
This is the factor multiplying the load vector for computing the load to be applied on the structure in the first load step.

**Minimum**
The minimum factor multiplying the load vector that is used in any load step. When the iterative analysis requires reduction in the magnitude of the load vector, the multiplier cannot be smaller than this value.

**Maximum**
The maximum factor multiplying the load vector that is used in any load step.

For More Information, please refer to the following documentation.
- Running an Analysis in *LARSA 4D User's Manual*. 
The core of the LARSA 4D analysis is the solution of the stiffness equations for displacements given forces at joints. The stiffness equations form the stiffness matrix $K$ such that $Kd = p$, where $d$ is a displacement vector and $p$ is a force vector. Several numerical methods are provided to solve this system for $d$. The options are listed in Analysis → Solver Options.

The choice of solver is a trade-off between speed and error checking. Fast numerical methods often come at the cost of not detecting all forms of numerical instability or reporting less detailed error messages. The stiffness matrix is numerically unstable when there are free degrees of freedom, also called rigid body motions. In other words, the structure is unstable. The solvers able to report which degrees of freedom are free are also the slowest.

Models may not have rigid body motions, meaning unconstrained motions. For example, two-dimensional models must have constraints in all three directions (or use Universal Restraints [in LARSA 4D User’s Manual]) even when all loading is limited to two dimensions. A model with unconstrained motion results in a stiffness matrix $K$ that is "singular." Some solvers detect some cases of a singular stiffness matrix and will report the problem and halt.

It is computationally expensive to know for sure whether a matrix is singular. Fast solvers will not attempt detect all types of instability. It is up to the user to ensure the model is entered correctly. These solvers may continue to report results even in these cases, but an inspection of the deformed model by the user typically will reveal there is a problem.

The LAPACK Banded Storage Solver is the recommended solver for small to medium sized models. This solver is optimized for Intel processors and uses a banded storage format to reduce memory consumption. It uses LU factorization with partial pivoting (similar to Gaussian elimination) and then forward/backward substitution. If the stiffness matrix is singular, the LAPACK Banded Storage Solver will report at least one degree of freedom (a joint number and direction) involved in rigid body motion.

The Intel PARDISO Direct Sparse Solver is recommended for large models. It is the fastest and most memory-efficient solver. As a sparse solver, it only stores the non-zero terms of the stiffness matrix in memory. The PARDISO solver is also multi-threaded, meaning it can take advantage of multi-core processors. This solver does not check the stiffness matrix for instability and may continue to produce results even for unstable models.

The Skyline Solver is LARSA 4D's oldest solver. It uses banded storage and a solution procedure similar to the LAPACK Banded Storage Solver. The Skyline solver currently reports the most informative error messages in the case of model instability. It is also the slowest solver.

The Vectorized Direct Sparse Solver was LARSA 4D's first sparse solver, with performance considerably faster than the Skyline Solver. As with the Intel PARDISO Solver, this solver uses sparse matrix storage, which means it is memory efficient for large models. On single-core processors the two sparse solvers have similar speeds. On multi-core processors, the Intel PARDISO Solver may be faster. This solver does not check the stiffness matrix for instability and may continue to produce results even for unstable models.

The Dense Matrix Storage Solver is used for debugging unstable models only. It stores the complete matrix in memory as a dense 2D array and uses a diagonal pivoting factorization method to solve the system of equations. With the Run Instability Check option turned on, Singular Value Decomposition is used to compute the largest and smallest eigenvalues of the stiffness matrix. An eigenvalue of zero, or extremely close to zero, indicates instability. Degrees of freedom (joint number and direction) in rigid body motions are reported.

As a convenience, all solvers remove degrees of freedom with no stiffness on the diagonal of the stiffness matrix. This usually occurs in joints that are not connected to active elements or intermediate joints on spans of cable elements.
where there is no rotational stiffness. If these degrees of freedom were not removed, the stiffness matrix would be singular and unsolvable.

The default solver selection is Automatic Selection, which uses the LAPACK Banded Storage Solver for small models and the PARDISO Direct Sparse Solver for large models.
This section is a reference for each type of result produced by the analysis engine.

For More Information, please refer to the following documentation.

- Getting Results in *LARSA 4D User’s Manual*.

### Joint Results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Joint Displacements</td>
<td>259</td>
</tr>
<tr>
<td>Definitions</td>
<td>259</td>
</tr>
<tr>
<td>Joint Reactions</td>
<td>261</td>
</tr>
<tr>
<td>Definitions</td>
<td>261</td>
</tr>
<tr>
<td>Joint Velocities &amp; Accelerations</td>
<td>263</td>
</tr>
<tr>
<td>Definitions</td>
<td>263</td>
</tr>
<tr>
<td>Camber Adjustment</td>
<td>265</td>
</tr>
<tr>
<td>Definitions</td>
<td>265</td>
</tr>
</tbody>
</table>

### Member Results

<table>
<thead>
<tr>
<th>Result Type</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member End Forces</td>
<td>269</td>
</tr>
<tr>
<td>Definitions - General</td>
<td>269</td>
</tr>
<tr>
<td>Definitions - When Reported in Global Directions</td>
<td>269</td>
</tr>
<tr>
<td>Definitions - When Reported in Local Directions</td>
<td>269</td>
</tr>
<tr>
<td>Member Sectional Forces</td>
<td>271</td>
</tr>
<tr>
<td>Definitions</td>
<td>271</td>
</tr>
<tr>
<td>Member Stresses</td>
<td>273</td>
</tr>
<tr>
<td>Computation of Normal Stresses</td>
<td>273</td>
</tr>
<tr>
<td>Definitions</td>
<td>273</td>
</tr>
<tr>
<td>Member Displacements</td>
<td>275</td>
</tr>
<tr>
<td>Definitions</td>
<td>275</td>
</tr>
<tr>
<td>Member Plastic Deformation</td>
<td>277</td>
</tr>
<tr>
<td>Definitions</td>
<td>277</td>
</tr>
<tr>
<td>Member Yield and Strains</td>
<td>279</td>
</tr>
<tr>
<td>Definitions</td>
<td>279</td>
</tr>
<tr>
<td>Span Displacements and Forces</td>
<td>281</td>
</tr>
<tr>
<td>Span Displacements</td>
<td>281</td>
</tr>
<tr>
<td>Span Sectional Forces</td>
<td>281</td>
</tr>
</tbody>
</table>
Analyzed Member Loads
  Definitions 283

Plate Results 285
  Plate Forces on Center
    Definitions 287
  Plate Forces at Joints - External
    Definitions 289
  Plate Forces at Joints - Internal
    Definitions 291
  Plate Stresses on Center and at Joints
    Definitions 293

Spring Results 295
  Spring Forces
    Local Axial/Local Torsional Springs 297
    Translation/Rotation X/Y/Z Directions 297
    Spring Deformations 299
    Local Axial/Local Torsional Springs 299
    Translation/Rotation X/Y/Z Directions 299
  Spring Yield 301
  Connection Beam Yield 303

Tendon Results 305

Eigenvalue Results 307
  Modal Frequencies
    Definitions 309
  Mode Shapes
    Definitions 311
  Modal Reactions and Modal Member/Plate Forces 313
Joint Results

Joint Displacements
Definitions
259
259

Joint Reactions
Definitions
261
261

Joint Velocities & Accelerations
Definitions
263
263

Camber Adjustment
Definitions
265
265
Joint Displacements

Joint displacements are the translational and rotational displacements of the joints as a result of the loading on the structure.

Every joint has, by default, six degrees of freedom: translation x/y/z and rotation x/y/z. Unless a joint's degree of freedom has been restrained by a support the joint is free to displace in that direction. (To exclude degrees of freedom from an analysis, see Universal Restraints [in LARSA 4D User’s Manual].)

In Staged Construction or incremental nonlinear analysis, displacements are accumulated from the start of the analysis and reflect the total displacement of the joint from its undeformed position.

Definitions

Joint
The ID of the joint that this row of data refers to.

Result Case
The name of the result case that this row of data refers to.

Translation (X/Y/Z)
The translation of the joint in x, y, and z.

specified w.r.t. global coordinates, or, if Report in Local UCS is chosen, the displacement coordinate system of the joint (see Joints (page 55))

positive values are in the positive direction of the corresponding axis

units: displacement-category units (see Results Units)

Rotation (X/Y/Z)
The rotation of the joint about the x-, y-, and z-axes.

specified w.r.t. global coordinates, or, if Report in Local UCS is chosen, the displacement coordinate system of the joint (see Joints (page 55))

positive values are about the corresponding axis using the right-hand rule

units: displacement-category units (see Results Units)

For More Information, please refer to the following documentation.

• Joints on page 55.
Joint Reactions

Joint reactions are the restraining forces caused by supports: joint restraints and grounded springs.

Reactions are only reported for joints that are fully or partially supported in at least one degree of freedom.

Definitions

**Joint**

The ID of the joint that this row of data refers to. Reactions only apply for joints restrained in one or more degrees of freedom.

**Result Case**

The name of the result case that this row of data refers to.

**Force (X/Y/Z)**

The force reactions at the joint in x, y, and z.

*specified w.r.t. global coordinates, or, if Report in Local UCS is chosen, the displacement coordinate system of the joint (see Joints (page 55))

*positive values* are in the positive direction of the corresponding axis

*units*: force-category units (see Results Units)

**Moment (X/Y/Z)**

The moment reactions at the joint about the x-, y-, and z-axes.

*specified w.r.t. global coordinates, or, if Report in Local UCS is chosen, the displacement coordinate system of the joint (see Joints (page 55))

*positive values* are about the corresponding axis using the right-hand rule

*units*: force-category units (see Results Units)

For More Information, please refer to the following documentation.

- Joints on page 55.
Joint Velocities & Accelerations

Joint velocities and accelerations are the velocities and accelerations at the joints.

Definitions

**Joint**
The ID of the joint that this row of data refers to.

**Result Case**
The name of the result case that this row of data refers to.

**Translation (X/Y/Z)**
The translational velocity or acceleration of the joint in x, y, and z.

*specified w.r.t. global coordinates, or, if Report in Local UCS is chosen, the displacement coordinate system of the joint (see Joints [page 55])*

*positive values* are in the positive direction of the corresponding axis

*units*: displacement-category units (see Results Units)

**Rotation (X/Y/Z)**
The rotational velocity or acceleration of the joint about the x-, y-, and z-axes.

*specified w.r.t. global coordinates, or, if Report in Local UCS is chosen, the displacement coordinate system of the joint (see Joints [page 55])*

*positive values* are about the corresponding axis using the right-hand rule

*units*: displacement-category units (see Results Units)

For More Information, please refer to the following documentation.

- Joints on page 55.
Camber Adjustment

Camber is the value by which the position of a point on a structure at construction time must be adjusted, so that at a given subsequent time, the displacements of the structure would result in the cambered point to be at a location determined by the installer.

On order to use this result type, you must run a staged construction analysis and use matched cast method during construction.

Definitions

Joint
The ID of the joint that this row of data refers to.

Result Case
The name of the result case that this row of data refers to.

Construction Step
The step at which the joint is constructed.

Camber Adjustment (Tx/Ty/Tz)
The required adjustment to the joint to bring the structure to zero displacement at the selected result step.

units: displacement-category units (see Results Units)

Unadjusted Installation Position (Tx/Ty/Tz)
The installation position (from the initial location) without the camber adjustment. This position is determined by the engine during the construction when matched cast method is used.

units: displacement-category units (see Results Units)
# Member Results

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Member End Forces</td>
<td>269</td>
</tr>
<tr>
<td>Definitions - General</td>
<td>269</td>
</tr>
<tr>
<td>Definitions - When Reported in Global Directions</td>
<td>269</td>
</tr>
<tr>
<td>Definitions - When Reported in Local Directions</td>
<td>269</td>
</tr>
<tr>
<td>Member Sectional Forces</td>
<td>271</td>
</tr>
<tr>
<td>Definitions</td>
<td>271</td>
</tr>
<tr>
<td>Member Stresses</td>
<td>273</td>
</tr>
<tr>
<td>Computation of Normal Stresses</td>
<td>273</td>
</tr>
<tr>
<td>Definitions</td>
<td>273</td>
</tr>
<tr>
<td>Member Displacements</td>
<td>275</td>
</tr>
<tr>
<td>Definitions</td>
<td>275</td>
</tr>
<tr>
<td>Member Plastic Deformation</td>
<td>277</td>
</tr>
<tr>
<td>Definitions</td>
<td>277</td>
</tr>
<tr>
<td>Member Yield and Strains</td>
<td>279</td>
</tr>
<tr>
<td>Definitions</td>
<td>279</td>
</tr>
<tr>
<td>Span Displacements and Forces</td>
<td>281</td>
</tr>
<tr>
<td>Span Displacements</td>
<td>281</td>
</tr>
<tr>
<td>Span Sectional Forces</td>
<td>281</td>
</tr>
<tr>
<td>Analyzed Member Loads</td>
<td>283</td>
</tr>
<tr>
<td>Definitions</td>
<td>283</td>
</tr>
</tbody>
</table>
Member End Forces

Member end forces are the resultant forces and moments acting on the joints of members, representing the effect of the structure that caused deformation of the members. Member end forces can be reported in either global directions or in the members' local coordinate systems.

If member end forces are reported in the local systems, the results should not be summed across members unless the members are all parallel and have the same orientation angle.

Definitions - General

**Member**
The ID of the member that this row of data refers to.

**Joint**
The ID of the joint at one of the ends of the member that this row of data refers to.

**Result Case**
The name of the result case that this row of data refers to.

Definitions - When Reported in Global Directions

This section describes the meaning of the forces and moments when member end forces are reported in global directions.

**Force (X/Y/Z)**
The forces at the joint in x, y, and z.

- *specified w.r.t. global directions*
- *positive values* are in the positive direction of the corresponding axis
- *units*: force-category units (see Results Units)

**Moment (X/Y/Z)**
The moments at the joint about the x-, y-, and z-axes.

- *specified w.r.t. global directions*
- *positive values* are about the corresponding axis using the right-hand rule
- *units*: force-category units (see Results Units)

Definitions - When Reported in Local Directions

This section describes the meaning of the forces and moments when member end forces are reported in local directions.

**Force X**
The axial force at the joint.

- *specified w.r.t. the local coordinate system of the member (see Members (page 57))
positive values For the start joint, positive values are compressive. For the end joint, positive values are tensile.
units: force-category units (see Results Units)

**Force Y**
The shear force at the joint in the member's local xy-plane.

*specified w.r.t. the local coordinate system of the member (see Members [page 57])*

positive values are in the positive direction of the local y-axis

units: force-category units (see Results Units)

**Force Z**
The shear force at the joint in the member's local xz-plane.

*specified w.r.t. the local coordinate system of the member (see Members [page 57])*

positive values are in the positive direction of the local z-axis

units: force-category units (see Results Units)

**Moment X**
The axial torque (torsion) at the joint.

*specified w.r.t. the local coordinate system of the member (see Members [page 57])*

positive values are about the member's local x-axis using the right-hand rule

units: force-category units (see Results Units)

**Moment Y**
The bending moment in the member's xz-plane (about the member local y-axis) at the joint.

*specified w.r.t. the local coordinate system of the member (see Members [page 57])*

positive values are about the member's local y-axis using the right-hand rule

units: force-category units (see Results Units)

**Moment Z**
The bending moment in the member's xy-plane (about the member local z-axis) at the joint.

*specified w.r.t. the local coordinate system of the member (see Members [page 57])*

positive values are about the member's local z-axis using the right-hand rule

units: force-category units (see Results Units)

For More Information, please refer to the following documentation.

- Joints on page 55.
- Members on page 57.
Member Sectional Forces

Member sectional forces are the internal forces and moments along the length of the member, in member local directions.

Sectional forces are computed using numerical integration over the length of the element, based on the joint forces at the start joint and applied member loads. They are reported at stations along the member. The number of stations (which is also used for the number of integration points) is set in Graphical Results Options [in LARSA 4D User’s Manual]. Because numerical integration is an approximation, results toward the ends of members may have a small degree of error.

Sectional forces are reported at sections (faces) with a positive normal directed along the member's positive x-axis. Because sectional forces are reported in the local systems of the members, the results should not be summed across members unless the members are all parallel and have the same orientation angle.

Please note that x- and z-moments are reported with the right-hand rule, while y-moments are reported with the left-hand rule. This inconsistency has been kept for compatibility reasons over the years.

Definitions

Member
The ID of the member that this row of data refers to.

Station
The station along the member that this row of data refers to. Stations range from 0 (the start of the member) to the number of segments specified in Graphical Results Options [in LARSA 4D User’s Manual] (the end of the member). Stations are equally spaced.

Result Case
The name of the result case that this row of data refers to.

Force X
The axial force at the station.

specified w.r.t. the local coordinate system of the member (see Members (page 57))

positive values are always tensile (in the direction of the local x-axis)

units: force-category units (see Results Units)

Force Y
The shear force at the station in the member's local xy-plane.

specified w.r.t. the local coordinate system of the member (see Members (page 57))

positive values are in the positive direction of the local y-axis

units: force-category units (see Results Units)

Force Z
The shear force at the station in the member's local xz-plane.

specified w.r.t. the local coordinate system of the member (see Members (page 57))
positive values are in the positive direction of the local z-axis
units: force-category units (see Results Units)

Moment X
The axial torque (torsion) at the station.
specified w.r.t. the local coordinate system of the member (see Members (page 57))
positive values are about the member's local x-axis using the right-hand rule
units: force-category units (see Results Units)

Moment Y
The bending moment in the member's xz-plane (about the member local y-axis) at the station.
specified w.r.t. the local coordinate system of the member (see Members (page 57))
positive values cause compression at the positive-z side of the member (positive y-moments are about the member's local y-axis using the left-hand rule)
units: force-category units (see Results Units)

Moment Z
The bending moment in the member's xy-plane (about the member local z-axis) at the station.
specified w.r.t. the local coordinate system of the member (see Members (page 57))
positive values cause compression at the positive-y side of the member (positive z-moments are about the member's local z-axis using the right-hand rule)
units: force-category units (see Results Units)

For More Information, please refer to the following documentation.

• Joints on page 55.
• Members on page 57.
Member Stresses

Member stresses are the internal stresses along the length of the member, in member local directions.

Stresses are reported at stations along the member. The number of stations is set in Graphical Results Options [in LARSA 4D User’s Manual].

Stresses are reported at sections (faces) with a positive normal directed along the member’s positive x-axis. Because stresses are reported in the local systems of the members, the results should not be summed across members unless the members are all parallel and have the same orientation angle.

Computation of Normal Stresses

Normal stress is the sum of continuity stress and self-equilibriating stress. Continuity stress is computed through the formula $F_x/\text{Area} + z*F_y/I_{yy} + y*F_z/I_{zz}$. For non-built-up sections, Area, I_{yy}, and I_{zz} are determined from sectional properties (page 23) and $F_x$, $M_y$, and $M_z$ come from sectional force results (page 271).

For built-up sections defined in the Section Composer, Area, I_{yy}, and I_{zz} reflect the properties of the shape containing the stress point. $F_x$, $M_y$, and $M_z$ are the sectional forces in the shape containing the stress point. If the section is composed of multiple shapes, the sectional forces in the shapes may differ from the total member sectional forces, either due to force distribution or self-equilibriating stress. Forces are distributed to shapes according to incremental changes in strain. Stresses at mid-span locations are not currently available for members with cross-sections comprising more than one shape.

Starting in LARSA 4D version 7.9, nonlinear thermal gradients (page 119) may cause self-equilibriating stress. Stresses at mid-span locations are not available for members with loading that causes self-equilibriating stress.

Definitions

**Member**

The ID of the member that this row of data refers to.

**Station**

The station along the member that this row of data refers to. Stations range from 0 (the start of the member) to the number of segments specified in Graphical Results Options [in LARSA 4D User’s Manual] (the end of the member). Stations are equally spaced.

**Result Case**

The name of the result case that this row of data refers to.

**P/A at Centroid**

The axial stress at the station at the centroid of the member. This is also the component of any normal stress caused only by the axial force at the station. For built-up sections this value is reported as not-applicable because the component of stress due to axial force varies from location to location on the cross-section depending on the material at the point and the history of activity of the shape containing the point.

*positive values* are tensile

*specified w.r.t.* the local coordinate system of the member (see Members (page 57))
Normal Stress (at Point 1-4)
The normal stress at the four points on the plane of the station defined by the user-specified stress recovery points (page 23). Normal stress is the sum of extensional stresses caused by the axial force, extensional stresses caused by bending moments, and self-equilibrating stresses.

- **positive values** are always tensile
- **specified w.r.t.** the local x-axis of the member (see Members (page 57)) at the member's section's stress recovery points (see Sections (page 23))

Max ABS Total Stress
The maximum absolute value of the normal stresses.

- **positive values** are always tensile
- **specified w.r.t.** the local x-axis of the member (see Members (page 57)) at the member's section's stress recovery points (see Sections (page 23))

Shear Stress (Y/Z)
The shear stresses are Fy/Ay and Fz/Az. Shear stresses can only be reported if sectional shear areas have been specified (see Sections (page 23)). Shear stresses caused by torsion are not included. For built-up sections this value is reported as not-applicable because the component of stress due to axial force varies from location to location on the cross-section depending on the material at the point and the history of activity of the shape containing the point.

- **positive values** are in the direction of the corresponding positive member local y- or z-axis
- **specified w.r.t.** the local coordinate system of the member (see Members (page 57))

units: stress-category units (see Results Units)

---

For More Information, please refer to the following documentation.

- Members on page 57.
Member Displacements

Member displacements are the span deflections and deformations along the members.

Member displacements are computed using numerical integration over the length of the element, based on the joint forces at the start joint and applied member loads. They are reported at stations along the member. The number of stations (which is also used for the number of integration points) is set in Graphical Results Options [in LARSA 4D User’s Manual]. Because numerical integration is an approximation, results toward the ends of members may have a small degree of error.

Member displacements are with respect to the undeformed position of the member and so include the displacements of the member end joints. These are not displacements from the chord connecting the deformed location of the end joints.

Definitions

Member
The ID of the member that this row of data refers to.

Station
The station along the member that this row of data refers to. Stations range from 0 (the start of the member) to the number of segments specified in Graphical Results Options [in LARSA 4D User’s Manual] (the end of the member). Stations are equally spaced.

Result Case
The name of the result case that this row of data refers to.

Translation (X/Y/Z)
The translational displacement of the member at this station in x, y, and z from its undeformed position. specified w.r.t. the local coordinate system of the member (see Members (page 57)) units: displacement-category units (see Results Units)

Rotation (X/Y/Z)
The rotational displacement of the member at this station about the x-, y-, and z-axes. specified w.r.t. the local coordinate system of the member (see Members (page 57)) units: displacement-category units (see Results Units)

For More Information, please refer to the following documentation.

• Joints on page 55.
• Members on page 57.
Member Plastic Deformation

Member plastic deformations are the components of the deformation of a member that result from yielding, which can occur in members with hysteretic behavior. Unlike elastic deformation, plastic deformation cannot be recovered.

The hysteretic beam inelastic behavior includes both axial force-moment-moment interaction (3D coupled plasticity) and the spread of yielding along the length of the member due to hardening (distributed yield model).

Definitions

Member

The ID of the member that this row of data refers to.

Result Case

The name of the result case that this row of data refers to.

Axial Plastic Deformation

[This information has not yet been entered into the documentation. We apologize for the delay.]

Start Joint Plastic Rotation YY/ZZ

[This information has not yet been entered into the documentation. We apologize for the delay.]

*positive values

*specified w.r.t. the local coordinate system of the member (see Members (page 57))

*units: displacement-category units (see Results Units)

End Joint Plastic Rotation YY/ZZ

[This information has not yet been entered into the documentation. We apologize for the delay.]

*positive values

*specified w.r.t. the local coordinate system of the member (see Members (page 57))

*units: displacement-category units (see Results Units)

For More Information, please refer to the following documentation.

• Members on page 57.
• Materials on page 19.
Member Yield and Strains

Member yield and strains are reported for hysteretic beam elements. The hysteretic beam inelastic behavior includes both axial force-moment-moment interaction (3D coupled plasticity) and the spread of yielding along the length of the member due to hardening (distributed yield model).

Yielding of connection beams and hysteretic springs are reported in Connection Beam Yield (page 303) and Spring Yield (page 301), respectively.

Total strain is the summation of the axial strain and the curvature multiplied by the distance from the centroid to the fiber.

Definitions

Member
The ID of the member that this row of data refers to.

Result Case
The name of the result case that this row of data refers to.

Station
The station along the member that this row of data refers to. Stations range from 0 (the start of the member) to 1 (the end of the member). There are always 5 stations reported. Stations are at the Gauss integration points of the member.

Yield Ratio
An indication of the amount of yielding at this station on the member. Zero indicates there is no stress on the member. One indicates the member is at the point of yielding. Values greater than 1 indicate the extent of post-yielding.

Strain (Axial Only)
The total axial strain at this station on the member.

Curvature YY/ZZ
The curvature at this station on the member about the local y- or z-axis of the member. 
positive values are about the corresponding axis using the right-hand rule
specified w.r.t. the local coordinate system of the member (see Members (page 57))
units: displacement-category units (see Results Units)
Span Displacements and Forces

Span displacements and forces report the member deflections and forces for an entire span. Spans are special entities in LARSA that describe a series of connected members.

Span Displacements

Span displacements are reported in the same manner as Member Displacements (page 275) with the following exceptions:

- The locations of the stations on the members is distributed evenly throughout the span according to the number of stations set in Graphical Results Options [in LARSA 4D User’s Manual].
- The displacement of a station is the distance from its deformed position to the chord from the beginning of the span to the end of the span.
- As a result, the first and last points in a span always have zero displacement.

Span Sectional Forces

Span sectional forces are reported in the same manner as Member Sectional Forces (page 271) with the following exception:

- The locations of the stations on the members is distributed evenly throughout the span according to the number of stations set in Graphical Results Options [in LARSA 4D User’s Manual].

For More Information, please refer to the following documentation.

- Members on page 57.
- Spans on page 69.
- Member Displacements on page 275.
- Member Sectional Forces on page 271.
Analyzed Member Loads

LARSA reports back what member loads were included in the analysis for verification purposes. The reported loads include input loads (page 105) entered by the user, computed self-weight (page 111), moving load patterns (page 125) for each generated moving load case, and tendon (page 89) equivalent forces.

For an explanation of member loads, see Member Loads (page 117) and Member Thermal Loads (page 119).

Definitions

Member
The ID of the member that the load described by this row of data is applied to.

Result Case
The name of the result case that this row of data refers to.

Load Type
Whether the load is a point force, point moment, uniform force, uniform moment, trapezoidal force or moment, thermal load, or other type of member load.

Load Direction
The direction that the load is applied in.

Magnitude at Start
The magnitude of the load at its start. For point loads, this is just the magnitude of the load.

Magnitude at End
The magnitude of the load at its end, if applicable.

Start Position
The position of the start of the load, in relative units where 0 is the start of the member and 1 is the end of the member.

End Position
The position of the end of the load, if applicable, in relative units where 0 is the start of the member and 1 is the end of the member.

For More Information, please refer to the following documentation.

• Member Loads on page 117.
• Moving Load Analysis on page 203.
Plate Results

Plate Forces on Center 287
   Definitions 287

Plate Forces at Joints - External 289
   Definitions 289

Plate Forces at Joints - Internal 291
   Definitions 291

Plate Stresses on Center and at Joints 293
   Definitions 293
Plate Forces on Center

Plate forces on center are the internal forces and moments of plates calculated at the centroid of the element and reported in the element's local coordinate system.

Plate forces are reported per unit of in-plane length.

Definitions

Plate
The ID of the plate that this row of data refers to.

Result Case
The name of the result case that this row of data refers to.

Membrane Force Fxx
Membrane force in the element's local x-axis, caused by the membrane behavior of the element.

*positive values* are tensile

*specified w.r.t. the local coordinate system of the plate (see Plates (page 71))*

*units:* force-category units (see Results Units)

Membrane Force Fyy
Membrane force in the element's local y-axis, caused by the membrane behavior of the element.

*positive values* are tensile

*specified w.r.t. the local coordinate system of the plate (see Plates (page 71))*

*units:* force-category units (see Results Units)

In-Plane Shear Fxy
The in-plane shear force, caused by the membrane behavior of the element. Fxy, the in-plane shear force on the face normal to the x-axis and in the direction of y, is equal to Fyx, the in-plane shear force on the face normal to the y-axis and in the direction of x.

*positive values* are in the direction of the positive local x- and y-axes

*specified w.r.t. the local coordinate system of the plate (see Plates (page 71))*

*units:* force-category units (see Results Units)

Moment Mxx
Bending moment in the face normal to the local x-axis in the y direction, caused by the plate bending behavior of the element.

*positive values* cause tension on the top (positive local z) surface of the element

*specified w.r.t. the local coordinate system of the plate (see Plates (page 71))*

*units:* force-category units (see Results Units)

Moment Myy
Bending moment in the face normal to the local y-axis in the x direction, caused by the plate bending behavior of the element.  
*positive values* cause tension on the top (positive local z) surface of the element

*specified w.r.t.* the local coordinate system of the plate (see Plates (page 71))

*units*: force-category units (see Results Units)

**Torsion Mxy**

Twisting (torsional) moment on the x- and y-faces of the element, caused by the plate bending behavior of the element.

*positive values* cause tension on the top (positive local z) surface of the element

*specified w.r.t.* the local coordinate system of the plate (see Plates (page 71))

*units*: force-category units (see Results Units)

**Shear Qx**

Shear force corresponding to the moment Mxx, in the local z direction.

*units*: force-category units (see Results Units)

**Shear Qy**

Shear force corresponding to the moment Myy, in the local z direction.

*units*: force-category units (see Results Units)

For More Information, please refer to the following documentation.

- Plates on page 71.
- Plate Forces at Joints - External on page 289.
- Plate Forces at Joints - Internal on page 291.
Plate Forces at Joints - External

External plate forces at joints are the resultant forces and moments acting on the joints of plates, representing the effect of the structure that caused deformation of the plates.

External plate forces at joints are reported in global coordinates.

The forces represent the total force at the joints, as opposed to plate forces on center or internal plate forces at joints which are per unit of in-plane length.

Definitions

Plate
The ID of the plate that this row of data refers to.

Joint
The ID of the joint at the corner of the plate that this row of data refers to.

Result Case
The name of the result case that this row of data refers to.

Force (Fx/Fy/Fz)
The forces at the joint in x, y, and z.

*specified w.r.t. global directions*

*positive values* are in the positive direction of the corresponding axis

*units*: force-category units (see Results Units)

Moment (Mx/My/Mz)
The moments at the joint about the x-, y-, and z-axes.

*specified w.r.t. global directions*

*positive values* are about the corresponding axis using the right-hand rule

*units*: force-category units (see Results Units)

For More Information, please refer to the following documentation.

- Plates on page 71.
- Plate Forces at Joints - Internal on page 291.
- Plate Forces on Center on page 287.
Plate Forces at Joints - Internal

Internal plate forces at joints are the resultant forces and moments acting on the joints of plates, representing the effect of the structure that caused deformation of the plates.

Internal plate forces at joints are reported in plate local directions (page 71) per unit of in-plane length. See also Plate Forces at Joints - External (page 289) for resultant forces reported at joints, and also Plate Forces on Center (page 287).

Definitions

Plate
The ID of the plate that this row of data refers to.

Joint
The ID of the joint at the corner of the plate that this row of data refers to.

Result Case
The name of the result case that this row of data refers to.

Membrane Force (Fxx/Fyy)
Membrane force in the element's local x- and y-axes, caused by the membrane behavior of the element, in local directions, per unit of in-plane length.

specified w.r.t. local directions

positive values are in the positive direction of the corresponding axis

units: force-category units (see Results Units)

In-Plane Shear Fxy
The in-plane shear force, caused by the membrane behavior of the element, in local directions, per unit of in-plane length. Fxy, the in-plane shear force on the face normal to the x-axis and in the direction of y, is equal to Fyx, the in-plane shear force on the face normal to the y-axis and in the direction of x.

specified w.r.t. local directions

positive values are in the positive direction of the corresponding axis

units: force-category units (see Results Units)

Moment (Mx/My)
Bending moment in the face normal to the local x-axis in the y direction, caused by the plate bending behavior of the element, in local directions, per unit of in-plane length.

specified w.r.t. local directions

positive values are about the corresponding axis using the right-hand rule

units: force-category units (see Results Units)

Torsion (Mxy)
Twisting (torsional) moment on the x- and y-faces of the element, caused by the plate bending behavior of the element, in local directions, per unit of in-plane length.

specified w.r.t. local directions
positive values are about the corresponding axis using the right-hand rule
units: force-category units (see Results Units)

Shear (Qx/Qy)
Shear force Qx corresponding to the moment Mx, and likewise for Qy and My, in the local z direction, per unit
of in-plane length.
specified w.r.t. local directions
units: force-category units (see Results Units)

For More Information, please refer to the following documentation.

• Plates on page 71.
• Plate Forces at Joints - External on page 289.
• Plate Forces on Center on page 287.
Plate Stresses on Center and at Joints

Plate stresses are the internal stresses of plates. Stresses are reported either at the center of the plate, in the spreadsheet named Plate Stresses on Center, or at the joint locations, in the spreadsheet named Plate Stresses at Joints.

For each plate, three results are available: stresses at the top (local +z), middle, and bottom (local -z) of the plate. For Plate Stresses at Joints, these three stresses are reported at each of the plate's three (for triangular plates) or four (for quadrilateral plates) joints.

Definitions

**Plate**
The ID of the plate that this row of data refers to.

**Result Case**
The name of the result case that this row of data refers to.

**Joint**
For Plate Stresses at Joints, this column indicates the joint ID at which the stress is being reported at. For triangular plates, a blank row may be entered in the spreadsheet for the K-Joint field although it is not used.

**Location**
The location on the plate for where the stress is reported: Top (+z local), Middle (z=0), Bottom (-z local).

**Stress Sxx**
The stress on the face normal to the local x-axis and in the direction of the local x-axis.

*positive values* are always tensile

*specified w.r.t.* the local coordinate system of the plate (see Plates (page 71))

*units:* stress-category units (see Results Units)

**Stress Syy**
The stress on the face normal to the local y-axis and in the direction of the local y-axis.

*positive values* are always tensile

*specified w.r.t.* the local coordinate system of the plate (see Plates (page 71))

*units:* stress-category units (see Results Units)

**Stress Sxy**
The stress on the face normal to the local x- or y-axis and in the direction of the other. The stress on the x-face in the direction of y is equal to the stress on the y-face in the direction of x.

*positive values* are always tensile

*specified w.r.t.* the local coordinate system of the plate (see Plates (page 71))

*units:* stress-category units (see Results Units)

**Principal Max/Min**
The largest possible normal stress and the smallest possible normal stress, which are always in orthogonal directions and may not be aligned with the x- and y-axes of the plate. On planes on which maximum or
minimum normal stresses occur, there are no shear stresses. These planes are called the principal planes of stress. The direction of the principal planes of stress with respect to the local coordinate system of the plate is reported in the Angle field, described below.

The maximum shear stress can be computed by using the principal stresses as follows:

\[
T_{\text{max}} = \frac{(\text{Principal}_{\text{max}} - \text{Principal}_{\text{min}})}{2}
\]

\[
T_{\text{min}} = \frac{-(\text{Principal}_{\text{max}} - \text{Principal}_{\text{min}})}{2}
\]

*positive values* are always tensile

*units: stress-category units (see Results Units)*

**Von Misses Stress**

The average of the principal stresses. Von Misses Stress is usually used in yield and fracture criteria.

*positive values* are always tensile

*units: stress-category units (see Results Units)*

**Angle**

The direction of the principal max stress as the angular offset from the local x-axis, in degrees. If Angle is 0, the principal max stress is in the direction of the positive local x-axis.

*specified w.r.t. the local coordinate system of the plate (see Plates [page 71])*

*units: stress-category units (see Results Units)*

For More Information, please refer to the following documentation.

- Plates on page 71.
## Spring Results

### Spring Forces

<table>
<thead>
<tr>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Axial/Local Torsional Springs</td>
<td>297</td>
</tr>
<tr>
<td>Translation/Rotation X/Y/Z Directions</td>
<td>297</td>
</tr>
</tbody>
</table>

### Spring Deformations

<table>
<thead>
<tr>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Axial/Local Torsional Springs</td>
<td>299</td>
</tr>
<tr>
<td>Translation/Rotation X/Y/Z Directions</td>
<td>299</td>
</tr>
</tbody>
</table>

### Spring Yield

<table>
<thead>
<tr>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>301</td>
</tr>
</tbody>
</table>

### Connection Beam Yield

<table>
<thead>
<tr>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>303</td>
</tr>
</tbody>
</table>
Spring Forces

Spring forces report the forces and moments acting on grounded and two-node springs. The spring element has forces that are equal at both ends. Forces are reported differently for the different spring directions. The following two fields are common to all spring force results.

**Spring**
The ID of the spring that this row of data refers to.

**Result Case**
The name of the result case that this row of data refers to.

The meaning of the remaining fields vary depending on the spring's direction attribute.

### Local Axial/Local Torsional Springs

Spring forces for local axial and local torsional springs are reported in or about the axis from the start joint to the end joint. These directions are applicable to two-node springs only.

Either Force Fx or Moment Mx will have a nonzero value, depending on the spring direction.

**Force Fx**
The axial translational force in the spring.
- positive values are compressive
- units: force-category units (see Results Units)

**Moment Mx**
The axial moment in the spring.
- positive values are about the local axis in the direction of the right-hand rule
- units: force-category units (see Results Units)

### Translation/Rotation X/Y/Z Directions

Springs acting in these directions have forces reported in global directions, except when a spring's start joint (or for a grounded spring its only joint) has a displacement coordinate system assigned, in which case the displacement system directions of that joint are used (regardless of any displacement coordinate system assigned to the end joint).

Only springs with 6x6 stiffness matrices assigned can have stiffness in more than one direction. Other springs will only have one non-zero force component.

For these springs, there is no notion of tension or compression. The spring forces are reported as the forces at the start of the spring element, in the spring’s direction. For a grounded (one-node) spring, the sign of the spring force is same as that of the displacement of the joint in the spring’s direction. For a two-node spring, the sign of the spring force matches the sign of the displacement of the first node minus the displacement of the second node, in the spring’s direction.
**Force Fx/Fy/Fz**

The translational forces in the x-, y-, and z-directions.

*units*: force-category units (see Results Units)

**Moment Mx/My/Mz**

The moments about the x-, y-, and z-axes.

*units*: force-category units (see Results Units)

For More Information, please refer to the following documentation.

- Springs on page 79.
Spring Deformations

Spring deformations report the deformation (elongation) of grounded and two-node springs.

Deformations are reported differently for the different spring directions. The following two fields are common to all spring force results.

**Spring**
- The ID of the spring that this row of data refers to.

**Result Case**
- The name of the result case that this row of data refers to.

The meaning of the remaining fields vary depending on the spring's direction attribute.

**Local Axial/Local Torsional Springs**

Spring deformations for local axial and local torsional springs are reported in or about the axis from the start joint to the end joint. These directions are applicable to two-node springs only.

Either Translation X or Rotation X will have a nonzero value, depending on the spring direction.

**Translation X**
- The axial elongation of the spring.
  - *positive values* are elongation
  - *units*: length-category units (see Results Units)

**Rotation X**
- The axial rotation of the spring.
  - *positive values* are about the local axis in the direction of the right-hand rule
  - *units*: angle-category units (see Results Units)

**Translation/Rotation X/Y/Z Directions**

Springs acting in these directions have deformation reported in global directions, except when a spring's start joint (or for a grounded spring its only joint) has a displacement coordinate system assigned, in which case the displacement system directions of that joint are used (regardless of any displacement coordinate system assigned to the end joint).

Only springs with 6x6 stiffness matrices assigned can have deformation in more than one direction. Other springs will only have one non-zero deformation component. A spring's deformation may not match the relative displacements of the spring's joint or end joints because deformation is only counted in the direction(s) the spring acts in.

For these springs, there is no notion of elongation or shortening. Positive values correspond to a relative displacement of the end joint in the positive directive compared to the displacement of the start joint (That is, the end joint's displacement minus start joint's displacement), in the direction chosen.

**Translation X/Y/Z**
The translational deformation in the x-, y-, and z-directions.

*positive values* are in the direction of the positive axis

*units:* length-category units (see Results Units)

**Moment Mx/My/Mz**

The rotations about the x-, y-, and z-axes.

*positive values* are about the axis in the direction of the right-hand rule

*units:* length-category units (see Results Units)

For More Information, please refer to the following documentation.

- Springs on page 79.
Spring Yield

[This information has not yet been entered into the documentation. We apologize for the delay.]
Connection Beam Yield

[This information has not yet been entered into the documentation. We apologize for the delay.]
Tendon Results

Tendon results include tendon forces along the tendon, forces at member ends, and detailed forces at member ends. These results are available in the Staged Construction Analysis.

This section describes the four related tendon results.

**Tendon Forces Variation Along Length**
Reports the forces of a specific tendon along its path. Forces are reported at roughly 1-foot intervals.

**Tendon Forces Total at Member Ends**
For selected members, reports the total of all tendon forces passing through the ends of the member.

**Tendon Forces Detailed at Member Ends**
For selected members, reports the forces of each tendon passing through the member's ends.

**Tendon Forces Detailed at Member Ends for PT Path**
For a selected tendon, reports each member along the path of the tendon and the forces of each tendon passing through the members' ends.

**Tendon**
The name of the tendon that this row of data refers to.

**Member**
The ID of the member that this point on the tendon falls within.

**Location**
Specifies whether this row of data refers to a point at the start or end of the member.

**# of Tendons**
The total number of tendons passing through at the given location.

**Distance**
The distance of the station from the start of the specified tendon path.

**Stage**
The construction stage result case for which this result applies.

**Force in Tendon**
Total force in the tendon at the given distance.

**Force Fx/Fy/Fz**
The force in the tendon at the given distance in each direction in member local directions.

**X/Y/Z in Member Local**
The coordinates of the tendon at the given distance along the tendon path relative to the member local coordinate system.

**Primary Mx/My/Mz**
The primary moment caused by this tendon in each direction in member local directions.

**Secondary Mx/My/Mz**
The secondary moment caused by this tendon in each direction in member local directions.

For More Information, please refer to the following documentation.

- Tendons on page 89.
## Eigenvalue Results

<table>
<thead>
<tr>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modal Frequencies</td>
<td>309</td>
</tr>
<tr>
<td>Definitions</td>
<td>309</td>
</tr>
<tr>
<td>Mode Shapes</td>
<td>311</td>
</tr>
<tr>
<td>Definitions</td>
<td>311</td>
</tr>
<tr>
<td>Modal Reactions and Modal Member/Plate Forces</td>
<td>313</td>
</tr>
</tbody>
</table>
Modal Frequencies

Modal frequencies are the undamped, free vibration frequencies of the structure, computed by an eigenvalue analysis.

LARSA computes the modes starting with the lowest frequency modes of a structure. The number of modes computed is limited by the number of modes requested when the analysis is run (Eigenvalue Analysis Options (page 187)) and the number of mass degrees of freedom.

Mode frequencies are reported in order of increasing frequency.

The participating mass ratio indicates the importance of this mode in the computation of the dynamic response, such as in Response Spectra Analysis (page 183).

Mode Shapes (page 311) are available for each mode frequency.

Definitions

**Mode Shape**
The name of the mode shape.

**Frequency**
The frequency of the mode shape.

*units: cycles per second*

**Period**
The period of the mode shape.

*units: seconds*

**Participating Mass Percent (X/Y/Z)**
The modal participating mass in each global direction, reported as a percent of the total mass.

**Cumulative Participating Mass Percent (X/Y/Z)**
The cumulative sum of all participating mass ratios for all mode frequencies below and including this one, reported as a percent of the total mass.
Mode Shapes

Mode shapes (eigenvectors, characteristic vectors, or normal modes) represent the amplitudes of motion of the structure for each dynamic mode.

The amplitudes are normalized with respect to the mass matrix of the structure.

Mode shapes are reported for each modal frequency (page 309).

Definitions

Joint

The ID of the joint that this row of data refers to.

Result Case (Mode Shape)

The name of the mode shape that this row of data refers to.

Translation (X/Y/Z)

The translation of the joint in x, y, and z. These values are unitless.

*positive values* are in the positive directions of the global coordinate system

specified w.r.t. global coordinates

Rotation (X/Y/Z)

The rotation of the joint about the x-, y-, and z-axes. These values are unitless.

*positive values* are about the global axes using the right-hand rule

specified w.r.t. global coordinates

For More Information, please refer to the following documentation.

• Modal Frequencies on page 309.
Modal Reactions and Modal Member/Plate Forces

Modal internal forces and reactions are computed for each mode (page 309) using the mode shapes (page 311) as the displacements of the structure.

For more information on these modal results, see the corresponding results for a static/dynamic analysis:

- Joint Reactions (page 261)
- Member End Forces (page 269)
- Plate Forces on Center (page 287)
Index

AASHTO LFD Point Loading, 223
AASHTO LRFD, 243
acceleration, 263
acceleration excitation function, 129
active/inactive option, 107
analysis options, 253
analysis scenarios, 153
analytical members, 69
analyzed loads, 283
anchor set, 89
axial force
   Member End Forces, 269
   Member Sectional Forces, 271
   Spring Forces, 297
backbone, 29
banded storage solver, 253
beams, 57
bearings
   Isolator Property Definitions, 33
   Isolators, 83
beta angle, 57
bridge coordinate systems, 43
bridge paths, 43
buckling, 169
cables, 57
camber, 265
cantilever, 265
cast, 265
cast in place, 133
casting day
   Cast Concrete Activities, 147
   Members, 57
CEB-FIP 2010, 243
CEB-FIP 78, 243
CEB-FIP 90, 243
cement hardening, 19
chains of members, 69
characteristic vectors, 311
coefficient of thermal expansion, 19
collapse, 169
combining results, 109
composite sections, 149
compression
   Member End Forces, 269
   Member Sectional Forces, 271
compressive strength, 19
cancrete
   Cast Concrete Activities, 147
   Members, 57
concrete creep and shrinkage
   Time Effects on Materials, 243
   Time-Dependent Material Property Definitions, 49
constraints
  DOF Constraints, 87
    Joints, 55
construct, 149
construct activity, 233
constructing elements, 133
construction activities
  Analysis Scenarios, 153
    Cast Concrete Activities, 147
    Composite Section Construction Activities, 149
    Construct and Deconstruct Activities, 133
    Construction Activities, 131
    Displacement Initializations, 145
    DOF Constraint Change Activities, 141
    Load Activities, 137
    Support and Hoist Activities, 139
    Tendon Stressing and Slackening Activities, 143
construction steps, 231
coordinate systems
  Bridge Paths, 43
  User Coordinate Systems, 35
coordinates, 55
creep
  Sections, 23
    Time Effects on Materials, 243
    Time-Dependent Material Property Definitions, 49
cross-sectional properties, 23
curvature, 279
cylindrical coordinate systems, 35
deconstruct activity
  Load Class Tracking, 247
  Staged Construction Activities, 233
deconstructing elements, 133
deconstruction/demolition, 231
deflection, 265
deformation, 277
deformations, 299
dense matrix storage solver, 253
dependent joint, 87
design lanes, 221
direct sparse solver, 253
displacement, 299
displacement excitation function, 129
displacement initialization, 233
displacement initializations, 145
displacement loads, 115
displacements
  Joint Displacements, 259
  Member Displacements, 275
  Span Displacements and Forces, 281
distributed loads
  Member Loads, 117
  Plate Loads, 121
DOF Constraint activity
  Load Class Tracking, 247
  Staged Construction Activities, 233
DOF Constraints
  DOF Constraint Change Activities, 141
DOF Constraints, 87
ductility, 23
dynamic analysis
  Linear Time History Analysis, 193
  Nonlinear Time History Analysis, 195
  Response Spectra Analysis, 183
  Time History Loads, 129
dynamic mass, 129
eigenstress, 119
eigenvalue analysis
  Eigenvalue Analysis Options, 187
  Eigenvalue and Stressed Eigenvalue Analysis, 181
  Response Spectra Analysis, 183
  Stressed Eigenvalue Analysis Options, 189
eigenvectors, 311
elastic modulus
  Materials, 19
  Time Effects on Materials, 243
EN1992-1-1:2004(E), 243
end offsets and releases, 57
Eurocode, 243
excitations, 129
fck, 19
follower forces, 165
force excitation function, 129
forces
  Joint Reactions, 261
  Member End Forces, 269
  Member Sectional Forces, 271
  Plate Forces at Joints - External, 289
  Plate Forces at Joints - Internal, 291
  Plate Forces on Center, 287
  Spring Forces, 297
  Tendon Results, 305
frequencies, 309
generated loads, 283
groups of members, 69
guaranteed ultimate tensile strength, 19
hinged cast, 133
hoist, 139
hoist activity, 233
HSXX vehicles, 225
hysteretic members
  Member Yield and Strains, 279
  Members, 57
hysteretic springs
  Spring Property Definitions, 29
  Springs, 79
incremental loading, 169
inelastic material properties, 19
inelastic members, 279
influence lines and surfaces
  Influence Analysis Overview and Options, 207
  Influence Line & Surface Analysis, 205
  Influence Loads, 127
influence surfaces, 221
initial conditions, 129
in-plane shear, 287
instability, 253
Intel PARDISO Direct Sparse Solver, 253
IRC vehicles, 225
Is Dynamic Mass option, 107
isolators
  Isolator Property Definitions, 33
  Isolators, 83
isotropic material properties, 19
joint displacement
  Joint Displacements, 259
  Joint Velocities & Accelerations, 263
joint displacement loads, 115
joint loads
  Joint Loads, 113
  Support Displacements, 115
joint reactions, 261
joints, 55
lane (UDL) loading, 221
lane width (design lane), 221
lanes, 99
LAPACK Banded Storage Solver, 253
large displacements, 165
linear static analysis, 161
live load analysis
  Influence Line & Surface Analysis, 205
  Moving Load Analysis, 203
live loads
  Influence Loads, 127
  Moving Loads, 125
load activity, 233
load cases
  Load Activities, 137
  Static Load Cases, 107
  Tendon Stressing and Slackening Activities, 143
load classes, 247
load combinations, 109
loads, 105
loads in Staged Construction Analysis
  Load Activities, 137
  Tendon Stressing and Slackening Activities, 143
local coordinate system
  Members, 57
  Plates, 71
locking translation, 87
mass, 129
mass elements, 85
master joint, 87
matched, 265
matched cast, 133
materials, 19
maximum absolute stress, 273
member deflections
  Member Displacements, 275
  Span Displacements and Forces, 281
member deformation/displacements, 277
member forces
  Member End Forces, 269
  Member Sectional Forces, 271
Span Displacements and Forces, 281
member loads
   Analyzed Member Loads, 283
   Member Loads, 117
   Member Thermal Loads, 119
member sectional stresses, 273
member stresses, 273
member thermal loads, 119
member yield and strains, 279
members
   Members, 57
   Spans, 69
membrane force, 287
modal frequencies, 309
mode shapes
   Eigenvalue and Stressed Eigenvalue Analysis, 181
   Mode Shapes, 311
   Response Spectra Analysis, 183
modulus of elasticity, 19
moment of inertia, 23
moments
   Joint Reactions, 261
   Member End Forces, 269
   Member Sectional Forces, 271
   Plate Forces at Joints - External, 289
   Plate Forces at Joints - Internal, 291
   Plate Forces on Center, 287
   Spring Forces, 297
moving load analysis
   Influence Line & Surface Analysis, 205
   Lanes, 99
   Moving Load Analysis, 203
moving loads
   Influence Loads, 127
   Moving Loads, 125
natural frequencies
   Eigenvalue and Stressed Eigenvalue Analysis, 181
   Response Spectra Analysis, 183
Newmark-Beta, 195
Newton-Raphson, 195
nodes, 55
nonlinear buckling analysis, 169
nonlinear pushover analysis, 171
nonlinear restart analysis, 175
nonlinear springs
   Spring Property Definitions, 29
   Springs, 79
nonlinear static analysis
   Nonlinear Static Analysis, 165
   Nonlinear Static Analysis Options, 173
   Overview of Staged Construction Analysis, 231
   P-Delta Analysis, 163
nonlinear thermal gradient, 119
nonlinear time history analysis
   Nonlinear Time History Analysis, 195
   Nonlinear Time-History Analysis Options, 199
normal modes, 311
normal stress, 273
orientation angle, 57
orthotropic material properties, 19
P/A, 273
PARDISO solver, 253
partial loads, 117
participating mass, 309
pastic section modulus, 23
p-delta analysis, 163
perimeter, 23
physical members, 69
pin connection, 57
plastic deformation, 277
plate forces
  Plate Forces at Joints - External, 289
  Plate Forces at Joints - Internal, 291
  Plate Forces on Center, 287
plate loads, 121
plate stresses, 293
plates, 71
point loads
  Joint Loads, 113
  Member Loads, 117
  Plate Loads, 121
Poisson ratio, 19
post-tensioning
  Tendon Results, 305
  Tendons, 89
precamber, 265
prestress force, 57
pre-tensioned tendons, 89
primary moments, 305
principal stress, 293
pushover analysis
  Nonlinear Pushover Analysis, 171
  Nonlinear Pushover Analysis Options, 177
reactions, 261
rehabilitation, 231
relaxation
  Time Effects on Materials, 243
  Time-Dependent Material Property Definitions, 49
releases, 57
removing elements in Staged Construction Analysis, 133
residual strength, 23
response spectra analysis, 183
restraints
  DOF Constraint Change Activities, 141
  DOF Constraints, 87
  Joints, 55
rigid body motion, 253
rigid zones, 57
rolling stock analysis, 203
rotation
  Joint Displacements, 259
  Joint Velocities & Accelerations, 263
  Member Displacements, 275
  Mode Shapes, 311
  Span Displacements and Forces, 281
  Spring Deformations, 299
section properties and dimensions, 23
segmental construction methods
  Construct and Deconstruct Activities, 133
  Support and Hoist Activities, 139
seismic analysis, 183
self-equilibrating stresses, 119
self-weight, 111
self-weight in Staged Construction Analysis
  Construct and Deconstruct Activities, 133
  Load Activities, 137
  Tendon Stressing and Slackening Activities, 143
semi-rigid end connections, 57
shear area, 23
shear force
  Member End Forces, 269
  Member Sectional Forces, 271
  Spring Forces, 297
shear forces, 287
shear modulus, 19
shear stress, 273
shells, 71
shock loading, 183
shrinkage
  Sections, 23
  Time Effects on Materials, 243
  Time-Dependent Material Property Definitions, 49
Skyline Solver, 253
slave/masters, 87
solution convergence, 173
solvers, 253
span displacements, 281
span forces, 281
spans, 69
sparse solver, 253
spherical coordinate systems, 35
spring deformations, 299
spring forces, 297
spring properties, 29
springs
  Spring Property Definitions, 29
  Springs, 79
staged construction activities, 233
staged construction analysis, 229
Staged Construction Analysis
  Analysis Scenarios, 153
  Cast Concrete Activities, 147
  Composite Section Construction Activities, 149
  Construct and Deconstruct Activities, 133
  Construction Activities, 131
  Displacement Initializations, 145
  DOF Constraint Change Activities, 141
  Load Activities, 137
  Support and Hoist Activities, 139
  Tendon Stressing and Slackening Activities, 143
stages, 233
static analysis
  Linear Static Analysis, 161
  Nonlinear Static Analysis, 165
P-Delta Analysis, 163
static loads, 107
steel relaxation
  Time Effects on Materials, 243
  Time-Dependent Material Property Definitions, 49
steps, 233
stiffness degradation, 29
stiffness matrix, 253
strain, 279
strand area, 89
stress
  Member Stresses, 273
  Plate Stresses on Center and at Joints, 293
stress direction, 293
stress recovery points, 23
stress stiffening, 165
stress tendon activity, 233
stress trajectory, 293
stressed eigenvalue analysis, 181
structure groups, 133
support change activity
  Load Class Tracking, 247
  Staged Construction Activities, 233
support changes, 139
support displacements, 115
support settlements, 115
tendon GUTS, 19
tendon relaxation, 19
tendon results, 305
tendon stressing activity, 233
tendons
  Tendon Results, 305
  Tendons, 89
tension
  Member End Forces, 269
  Member Sectional Forces, 271
thermal expansion, 19
thermal loads
  Member Thermal Loads, 119
  Plate Loads, 121
time effect on elastic modulus, 243
time history analysis
  Linear Time History Analysis, 193
  Linear Time History Analysis Options, 197
  Nonlinear Time History Analysis, 195
  Nonlinear Time-History Analysis Options, 199
  Time History Loads, 129
time-dependent material properties
  Overview of Staged Construction Analysis, 231
  Time Effects on Materials, 243
  Time-Dependent Material Property Definitions, 49
torsion
  Plate Forces on Center, 287
  Spring Forces, 297
torsional constant, 23
translation
  Joint Displacements, 259
  Joint Velocities & Accelerations, 263
Member Displacements, 275
Mode Shapes, 311
Span Displacements and Forces, 281
Spring Deformations, 299
trapezoidal loads, 117
trusses, 57
ultimate stress, 19
uniform base excitation, 129
uniform loads, 117
unit weight, 19
unstable model, 253
user coordinate systems
  Bridge Paths, 43
  User Coordinate Systems, 35
Vectorized Direct Sparse Solver, 253
vehicles
  Lanes, 99
  Standard Vehicles, 225
vehicular loads
  Influence Loads, 127
  Moving Load Analysis, 203
  Moving Loads, 125
velocity, 263
Von Misses stress, 293
weight, 111
wobble, 89
yield ratio, 279
yield stress, 19
Young's modulus, 19