

**LARSA 2000 Reference**

**for**

**LARSA 2000**

**Finite Element Analysis and Design Software**



Larsa, Inc.  
Melville, New York, USA

Revised August 2004

# Table of Contents

<b>Introduction</b>	<b>6</b>
<b>Model Data Reference</b>	<b>7</b>
Elements Overview	9
The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements	10
The Plate Element: Plates and Shells	17
The Spring Element: A Multipurpose Element	19
Isolators and Bearings: The Seismic Elements	22
The Tendon Pseudoelement	25
Properties	28
Materials	29
Sections	32
Spring Property Definitions	36
Isolator/Bearing Property Definitions	40
Time-Dependent Material Property Definitions	44
User Coordinate Systems	49
Geometry	53
Joints	54
Members	57
Spans	62
Plates	63
Springs	66
Isolators and Bearings	69
Mass Elements	70
Slave/Master Constraints	72
Tendons	74

## LARSA 2000 Reference

---

Lanes	76
Tendon and Lane Paths	77
Loads	82
Static Load Cases	84
Load Combinations	86
Self Weight	89
Joint Loads	91
Support Displacements	92
Member Loads	93
Member Thermal Loads	96
Plate Loads	98
Moving Loads	100
Time History Loads	102
<b>Analysis Reference</b>	<b>105</b>
Static Analysis	107
Linear Static Analysis	108
P-Delta Analysis	111
Nonlinear Static Analysis	114
Nonlinear Buckling Analysis	120
Nonlinear Pushover Analysis	122
Nonlinear Static Analysis Options	123
Restart Nonlinear and Nonlinear Buckling Analysis Options	125
Nonlinear Pushover Analysis Options	127
Eigenvalue Analysis	129
Eigenvalue and Stressed Eigenvalue Analysis	130
Response Spectra Analysis	132
Eigenvalue Analysis Options	138
Stressed Eigenvalue Analysis Options	139
Time History Analysis	140
Linear Time History Analysis	141
Nonlinear Time History Analysis	143
Time History Analysis Options	145
Nonlinear Time-History Analysis Options	147
Live Load Analysis	149
Moving Load Analysis	150

## LARSA 2000 Reference

---

Influence-Based Analysis	152
Influence Method Options	159
Influence Lines	162
Influence Surfaces	165
Standard Vehicles	167
Moving Load Analysis Options	171
Staged Construction Analysis	172
<b>Analysis Results Reference</b>	<b>174</b>
Joint Results	176
Joint Displacements	177
Joint Reactions	179
Joint Velocities & Accelerations	181
Member Results	183
Member End Forces	184
Member Sectional Forces	187
Member Stresses	190
Member Displacements	192
Member Plastic Deformation	194
Member Yield and Strains	196
Span Displacements and Forces	198
Analyzed Member Loads	200
Plate Results	202
Plate Forces on Center	203
Plate Forces at Joints	205
Plate Stresses	207
Spring Results	210
Spring Forces	211
Spring Yield	213
Connection Beam Yield	214
Tendon Results	215
Eigenvalue Results	217
Modal Frequencies	218
Mode Shapes	220
Modal Reactions and Modal Member/Plate Forces	222



# Introduction

LARSA 2000 is the premier general purpose structural analysis and design software. In use throughout the world, LARSA 2000 boasts advanced analytical features, from influence surface based analysis to nonlinear time history analysis, and an all-new user interface. LARSA 2000: 4th Dimension, the most advanced program in the LARSA 2000 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 2000 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 2000 Reference Manual, a part of the series of manuals for LARSA 2000. 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 2000: staged construction analysis, bridge analysis, and pushover analysis. In addition, the User's Guide explains how to use LARSA 2000'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.

<b>Elements Overview</b>	<b>9</b>
The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements	10
The Plate Element: Plates and Shells	17
The Spring Element: A Multipurpose Element	19
Isolators and Bearings: The Seismic Elements	22
The Tendon Pseudoelement	25
<b>Properties</b>	<b>28</b>
Materials	29
Sections	32
Spring Property Definitions	36
Isolator/Bearing Property Definitions	40
Time-Dependent Material Property Definitions	44
User Coordinate Systems	49
<b>Geometry</b>	<b>53</b>
Joints	54
Members	57
Spans	62
Plates	63
Springs	66
Isolators and Bearings	69
Mass Elements	70
Slave/Master Constraints	72
Tendons	74
Lanes	76

---

## LARSA 2000 Reference

---

Tendon and Lane Paths	77
<b>Loads</b>	<b>82</b>
Static Load Cases	84
Load Combinations	86
Self Weight	89
Joint Loads	91
Support Displacements	92
Member Loads	93
Member Thermal Loads	96
Plate Loads	98
Moving Loads	100
Time History Loads	102

---

## Elements Overview

This section provides an overview of the member, plate, spring, isolator/bearing, and tendon elements of LARSA 2000. Each of these element types is a broad grouping of related element models, most of which have linear, nonlinear, and inelastic (hysteretic) forms.

The member element, for instance, represents beams, cables, trusses, and other beam-like elements that connect exactly two joints. The plate element models both plates and shells and can be either triangular or quadrilateral.

The spring element can represent springs or any general stiffness element. Springs can be grounded, which means they connect a single joint to a fixed point in space, or they can be two-node, which means they connect two joints.

The formulation of the isolators, bearings, and tendons are also described in this section.

<b>The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements</b>	<b>10</b>
<b>The Plate Element: Plates and Shells</b>	<b>17</b>
<b>The Spring Element: A Multipurpose Element</b>	<b>19</b>
<b>Isolators and Bearings: The Seismic Elements</b>	<b>22</b>
<b>The Tendon Pseudoelement</b>	<b>25</b>

---

## **The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements**

Members are used to model beams, trusses, cables, and other beam-like elements, except springs and isolators. Members connect two joints [p54] and have six degrees of freedom at each joint: three translational and three rotational. Members must have a non-zero length.

Members come in several forms, depending on the element it is being used to model. The types of members are listed below.

### **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.

## **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  $I_x$ ,  $I_y$ ,  $I_z$  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.

## 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 [p36] 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*.

## Material Properties of Members

A member is assigned one material [p29], which is constant throughout the member.

## Sectional Properties of Members

A member is assigned one or two sections [p32], 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.

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 Members [p57]), 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.

### **Initial Prestressing**

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 [p74]. Member loads [p93] 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.

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 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).

### **End Offsets (Rigid Links)**

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.

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.

### **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  $F_y$  and  $F_z$  may be released. Semi-rigid releases can be applied only for  $M_y$  and  $M_z$ .

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.

## **Self-Weight Computation**

The self-weight of members used in static load cases with self-weight [p89] 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.

---

### **For More Information**

- Members [p57].

## **The Plate Element: Plates and Shells**

The plate is used to model shell behavior, plate bending (out-of-plane action), or plane stress (membrane actions, in-plane action). Shell behavior includes both actions. The plate is a planar element with constant thickness, either triangular or quadrilateral in shape, with isotropic material properties.

The plate element is a linear element with linear material properties [p29]. Geometric nonlinearity is only introduced through the updating of the deformed shape that occurs in the nonlinear analyses.

### **Membrane Planar Element**

Membrane behavior is based on a displacement field leading to a compatible isoparametric element formulation. The displacement field includes higher order displacement functions that allow the displacements to vary quadratically between the nodes. This increase in the degrees of freedom within the element yields a more flexible element.

### **Bending Planar Element**

Bending behavior employs one translational and two rotational degrees of freedom at each joint. A cubic polynomial is used to describe the displacement of the mid-plane.

### **Shell Element**

The shell element is the most general form of the plate element. It has both in-plane membrane and out-of-plane bending, as described for the previous two element types.

## **Self-Weight Computation**

Automatic self-weight computation [p89] uses the plate's material unit weight [p29] and the plate's thickness to determine the magnitude of the uniform distributed load placed in the given direction of self-weight.

---

### **For More Information**

- Plates [p63].

## **The Spring Element: A Multipurpose Element**

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, which 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.

For coupled springs, see Isolators and Bearings: The Seismic Elements [p22].

Nonlinear springs can be used in any nonlinear static or nonlinear dynamic analysis, including staged construction analysis.

The spring element is weightless. Joint loads [p91] for static analysis or mass elements [p70] for dynamic analysis can be used to model the weight of springs, if necessary.

### **Grounded Spring Element**

The element 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.

This element can act in translation (axial) or in rotation (torsional). The direction is along or about the global coordinate system, or the joint displacement directions [p54] assigned to the joint the spring is connected to if a joint displacement system has been assigned to the joint.

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 [p36] is applied to the spring.

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 [p36] can be assigned describing the force-deformation curve of the element.

### **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. It can act in axial or torsional directions along or about the straight line connecting the spring's ends, provided the length of the element is non-zero.

It can also act in one of translation x, y, or z, or rotation x, y, or z. In these cases, the length of the element may be zero. 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 [p54], or along or about a user coordinate system [p49] 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.

The linear two-node spring element has the same stiffness in both tension and compression. Only one spring constant can be specified (see Springs [p66]).

A spring curve [p36] can be assigned to nonlinear two-node springs, describing the force-deformation curve of the spring. The nonlinear elastic spring element loads and unloads along the same path.

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. [p22]

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.

---

### **For More Information**

- Spring Property Definitions [p36].
- Springs [p66].
- For coupled springs, see Isolators and Bearings: The Seismic Elements [p22].

## **Isolators and Bearings: The Seismic Elements**

The isolator and bearing elements are used to model isolation devices, energy dissipation devices, and bearings, and they exhibit a nonlinear force-deformation relationship with hysteretic behavior. These elements can be either uniaxial or biaxial.

The following types of isolator and bearing elements are supported.

### **Viscous Fluid Dampers**

This element 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 because the response of the element is velocity-dependent.

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 can take values in the range of 0.5 to 2.0.

This element can be either linear or nonlinear. For the linear element, the power of the velocity is assumed one. Only a single viscous constant is needed.

For the nonlinear element, the nonlinear damping force of these devices is proportional to the power of the relative velocity across the damper. The nonlinear element can be single-range (type 1) or two-range (type 2). These elements are used to model Coulomb damping, viscous damping, orifice damping, and velocity-raised-to-power-p damping. One or two sets of viscous constants, power ranges, and force offsets are needed.

### **Elastomeric Bearings and Steel Dampers**

The element models the behavior of low damping rubber bearings, high damping rubber dampings in the range of strain prior to stiffening, and lead-rubber bearings.

This element is made up of three springs. The plane of the bearing element (xy-plane) is

made of two inelastic springs with hysteretic behavior, thus hysteretic spring properties must be specified for both of these planar springs (x and y material values), including elastic stiffness, yield displacement, and post-to-pre-yield stiffness ratio. The springs can be either coupled or uncoupled.

The third spring acts perpendicularly to the bearing plane carrying the vertical load. The stiffness for this spring ( $K_{zz}$  compression and tension constants) is optional.

### **Stiffening Biaxial Hysteretic Element**

The element is appropriate for modeling the behavior of high damping rubber bearings. Typically, these bearings exhibit higher stiffness at large strains. This element can be considered a combination of the elastoplastic version of the biaxial hysteretic element and a stiffening bilinear spring.

### **Frictional Elements**

The frictional elements are flat sliding bearings, friction pendulums, and Coulomb model bearings.

For flat sliding bearings, the mobilized forces are proportional to the coefficient of sliding friction and the vertical load carried by the bearing. The conditions of separation and reattachment and biaxial interaction are accounted for by using an additional variable. The minimum and maximum values of the coefficient of friction and the transition constant are provided by the user. From these, the coefficient is calculated internally. When the minimum and maximum values are equal, the element becomes a Coulomb model bearing.

The friction pendulum (FPS) bearing consists of a spherical sliding surface and an articulated slider which is faced with a high pressure capacity bearing material. The force-displacement relation of an FPS bearing includes the radius of curvature of the spherical sliding surface, the normal load, and the coefficient of sliding friction.

The direct effects of the variation in the normal load on the behavior of the FPS bearing are to instantaneously change the stiffness and friction force. Another indirect effect is to change the coefficient of friction which is pressure dependent. Modeling of the behavior of the FPS bearing to this detail is important in the accurate estimation of the forces in individual bearings.

## **References**

Parts of this documentation were originally written in the following sources, and the elements are based on the research of these authors.

P.C. Tsopelas, M.C. Constantinou, and A.M. Reinhorn. (1994). "3D-BASIS-ME: Computer Program for Nonlinear Dynamic Analysis of Seismically Isolated Single and Multiple Structures and Liquid Storage Tanks." Technical Report NCEER-94-0010, State University of New York at Buffalo.

---

## **For More Information**

- Isolator/Bearing Property Definitions [p40].
- Isolators and Bearings [p69].

## The Tendon Pseudoelement

Tendons are used for post-tensioning members, and they are modeled with both short-term and long-term losses. The tendon in LARSA is a pseudoelement. It is modeled along side the other geometric elements of a model.

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 a tendon idealized as a series of straight segments there is a gradual stress loss along each segment due to wobble friction and curvature. 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:

$$\ln(P_s/P_x) = KL + ma$$

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.

## 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.

---

### **For More Information**

- Tendons [p74].

# 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.

<b>Materials</b>	<b>29</b>
<b>Sections</b>	<b>32</b>
<b>Spring Property Definitions</b>	<b>36</b>
<b>Isolator/Bearing Property Definitions</b>	<b>40</b>
<b>Time-Dependent Material Property Definitions</b>	<b>44</b>
<b>User Coordinate Systems</b>	<b>49</b>

---

## Materials

Material property data is for defining the properties for various materials in the structural model. These material properties are assigned to members, plates, and tendons.

Material properties most often define linearly elastic behavior; however, when assigned to hysteretic beam elements [p10], these properties can define inelastic behavior.

### 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.

#### Name

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

#### 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 in the range of -1.0 to 0.5, non-inclusive.

#### 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.

#### 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.

## **Material Properties for Steel Design and Inelastic Analysis**

These fields are used for steel design and for hysteretic beam 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.

### **Post-Yield to Initial Slope Ratio**

This entry defines the slope of the stress-strain curve after the material yields.

## **Concrete and Steel Properties**

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

Used for tendons. This is the guaranteed ultimate yield stress.

### **Tendon Relaxation Type**

A time-dependent material property definition [p44] used for tendons in time-dependent staged construction analysis to determine the long-term losses due to relaxation as a function of time.

### **Concrete $f_{c28}$ or Steel $F_u$**

Compressive strength of concrete at day 28 or ultimate stress for steel. This entry is used in time-dependent staged construction analysis if the material is concrete.  $F_u$  of steel 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.

## **Orthotropic Material Properties for Plates**

These fields are reserved for future use.

### **Modulus of Elasticity E2**

This is the Elastic Modulus of the orthotropic material for the second direction.

### **Shear Modulus G2**

This is the Shear Modulus of the orthotropic material for the second direction.

---

## **For More Information**

- Time Effects on Materials [in *LARSA 2000/4th Dimension: Staged Construction Analysis*].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## Sections

The section data is for describing the geometric section 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 local coordinate system of the member.

## 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.

### Torsional Constant

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

### Moment of Inertias $I_{yy}$ and $I_{zz}$

These are the area moments of inertia  $I_{yy}$  and  $I_{zz}$  about the member y- and z-axes. These values are used for the bending stiffness of beam elements.

### Shear Areas

The shear areas  $A_y$  and  $A_z$  are for transverse shear in the xy- and xz-planes of with corresponding transverse shear stiffness as  $A_y * G$  and  $A_z * G$ .

The shear areas are computed for custom sections from specified dimensions and the shape. The computation for some of the available shapes are:

- Rectangular Section  $A_y = A_z = (5/6) * \text{Section Area}$
- I-Shape Section  $A_y = (5/3) * \text{Flange Thickness} * \text{Flange Width}$
- I-Shape Section  $A_z = \text{Web Thickness} * \text{Depth}$
- Solid Circular Section  $A_y = A_z = (9/10) * \text{Section Area}$
- Circular Tube Section  $A_y = A_z = \text{PI} * \text{Radius} * \text{Thickness}$
- Rectangular Tube Section  $A_y = 2 * \text{Depth} * \text{Wall Thickness}$
- Rectangular Tube Section  $A_z = 2 * \text{Width} * \text{Wall Thickness}$

## Properties for Time-Dependent Analysis

### Perimeter

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

### Creep Shrinkage Type

This is the time-dependent material properties definition [p44] assigned to the section. This is used to determine creep and shrinkage effects.

## Properties for Inelastic Analysis

### Plastic Section Modulus $Z_{yy}$ and $Z_{zz}$

Plastic section moduli are used for calculating the plastic moment capacity of beam elements in nonlinear inelastic analysis. Plastic section modulus  $Z_{yy}$  is for yield moment  $M_y$  where  $M_y(\text{yield}) = Z_{yy} * F_y$  (bending about y) and plastic section modulus  $Z_{zz}$  is for yield moment  $M_z$  where  $M_z(\text{yield}) = Z_{zz} * F_y$  (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 beam.

## Stress Recovery Points

The stress recovery points are local Y and Z coordinates for four points on the outer edges of the section.

The (y, z) coordinates for these points are specified in the member coordinate system using the proper sign convention with respect to the member local axis. See Members [p57].

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 locating tendons in beams when tendon locations are specified with reference to the top/bottom or left/right edges of the section. See Tendon and Lane Paths [p77].

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 or from the LARSA Section Composer.

## **Section Dimensions**

These are the dimensions used in Custom Sections dialog box 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.

---

### **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Tendon and Lane Paths [p77].

## 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 [p66] 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)

## 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 data are only for inelastic hysteretic springs:

### 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 describing the decrease of elastic stiffness with increasing plastic deformation. Smaller value implies higher stiffness degradation. In general  $\text{Alpha}=2000$  implies practically no degradation.

### Beta 1

Ductility-based strength degradation parameter.

### Beta 2

Energy-based strength degradation parameter.

### Gamma

Slip parameter, a fraction of the yield moment, specified as a number between 0 and 1.

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.

The stiffness  $\mathbf{k}$  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  $\text{FORCE} = \mathbf{K} \times \Delta\mathbf{L}$  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.

For curves assigned to two-node springs (i.e. not grounded springs), the sign convention is as follows:

- Extensions and tensile forces have positive signs.
- Shortening and compressive forces have negative signs.

For curves assigned to grounded springs, the sign convention is as follows:

- Positive displacements will produce tensile forces (positive). This results in negative reaction forces.

---

### **For More Information**

- The Spring Element: A Multipurpose Element [p19].
- Springs [p66].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## Isolator/Bearing Property Definitions

Isolator/Bearing property definitions set the behavior of one or more isolator/bearing elements used in the model. These property definitions are assigned to isolator/bearing elements [p69] to give the elements their behavioral properties.

Property definitions are used to define properties for viscous fluid dampers, elastomeric bearings and steel dampers, frictional elements, and stiffening biaxial hysteretic elements. Different data are required for each isolator/bearing type.

Property definition data common to all types of isolator/bearing definitions consists of the following:

### **Name**

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

### **Class**

The property definition can define the behavior of one of these classes of isolators and bearings:

- Elastomeric: Elastomeric Bearings and Steel Dampers
- Frictional: Frictional Elements
- Stiffening: Stiffening Biaxial Hysteretic Elements
- Viscous: Viscous Fluid Dampers

### **Sub-Type**

For the Frictional and Viscous classes, a sub-type of isolator/bearing must be chosen. For Frictional, the sub-types available are Coulomb, Flat Sliding, and Friction Pendulum. For Viscous, the sub-types available are Linear, Type I Nonlinear, and Type II Nonlinear.

For an explanation of the different classes and sub-types of isolators and bearings, see Isolators and Bearings: The Seismic Elements [p22].

The remaining fields depend on the class of isolator chosen.

## Viscous Fluid Dampers

The Viscous Constant, Velocity Power, and Force Offset fields are entered for nonlinear Type I and II dampers. They are not applicable for dampers with the sub-type linear. For Type I Nonlinear dampers, the fields are entered for Range 1 only. For Type II dampers, they are entered for both Range 1 and Range 2.

### Viscous Constant (Range 1/2)

The damping coefficient.

### Velocity Power (Range 1/2)

The power that the velocity is raised to. The usual range is 0.5 to 1.2. If the value is 1.0, then the element is a linear viscous element.

### Force Offset (Range 1/2)

The initial force value at rest. This value is determined from the operation data of the damper.

The remaining two fields are entered for all viscous dampers.

### Max Damper Force

The maximum force that can be applied by the damper.

### Limit Velocity

[This information has not yet been entered into the documentation. We apologize for the delay.]

## Elastomeric Bearings and Steel Dampers

This element is made up of three springs, two in the base plane of the bearing, the X and Y springs, and one acting perpendicularly, the Z spring. Stiffness for the Z spring is optional.

### K Elastic (X/Y)

The spring stiffness in the elastic range for each of the X and Y springs.

### Yield Displacement (X/Y)

The yield displacement for the X and Y springs.

**Kpost/pre (X/Y)**

The post-to-pre yielding stiffness ratio for the X and Y springs.

**Kzz Compression/Tension**

The spring constants for the Z spring.

## **Stiffening Biaxial Hysteretic Element**

This element can be considered a combination of the elastoplastic version of the biaxial hysteretic element and a stiffening bilinear spring (the Z spring). Stiffness for the Z spring is optional.

The stiffness of this element is determined by three ranges of displacement, from 0 to D1 (initiation of stiffening), D1 to D2 (transition range), and greater than D2 (the final range). In the initiation of stiffening range, stiffness is constant at K1. In the transition range, stiffness linearly increases from K1 to K2. In the final range, stiffness is constant at K2.

**Kzz Compression/Tension**

The spring constants for the Z spring.

**Tangent K1**

The tangent stiffness during the initiation of stiffening.

**Tangent K2**

The tangent stiffness during the final range.

**D1**

The displacement at which the initiation of stiffening ends and the transition range begins.

**D2**

The displacement at which the transition range ends and the final range begins.

**Yield Displacement**

The yield displacement.

**Characteristic Strength Q**

The characteristic strength.

## Frictional Elements

The frictional elements are flat sliding bearings, friction pendulums, and Coulomb model bearings.

### Stiffness

The spring constant for the Z direction.

### Radius

Radius of curvature of the concave surface of the bearing. For friction pendulum bearings only.

The following fields are entered for each direction X and Y.

### Friction Coefficient at Maximum Velocity (X/Y)

The maximum coefficient of sliding friction, measured at almost zero pressure.

### Friction Coefficient at Minimum Velocity (X/Y)

The minimum coefficient of sliding friction, which is independent of pressure.

### Transition Constant (X/Y)

Controls the transition of coefficient of sliding friction from the maximum to minimum values.

### Yield Displacement (X/Y)

The yield displacement.

---

## For More Information

- Isolators and Bearings: The Seismic Elements [p22].
- Isolators and Bearings [p69].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## **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 sections [p32] when they are used to calculate creep and shrinkage for members [p57].

### **For Steel Relaxation**

Time-dependent material property definitions are assigned to materials [p29] when they are used to calculate relaxation for tendons [p74].

## **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:

$$j(t, t_0) = ba(t_0) + jdbd(t-t_0) + (jf1jf2)[bf(t) - bf(t_0)]$$
$$ba(t_0) = 0.8(1 - fc(t_0)/fc_{ref})$$

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

See Time Effects on Materials [in *LARSA 2000/4th Dimension: Staged Construction Analysis*] 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:

$$h_0 = (\text{Notional Thickness Coefficient}) * (2 * \text{Cross-Section Area}) / (\text{Perimeter in Contact with the atmosphere})$$

### Flow Coefficients

#### Flow Coefficient 1

jf1. 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

jf2. 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, jf1 and jf2.

Note: jf1 = 2 and jf2 = 1.4 will represent an average concrete setting and aging in average conditions with a notional thickness of 0.40m.

### Delayed Modulus of Elasticity

jd. 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

$es1$  . 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

$es2$  . 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-t0) is given by

$$e0 = es1 \cdot es2 \text{ (This is the basic shrinkage coefficient.)}$$
$$es(t, t0) = e0[bs(t) - bs(t0)]$$

The basic shrinkage coefficient corresponds to *EPS* in the BC software program.

$bs$  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  $bd$ , flow function  $bf$  and shrinkage function  $bs$  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.

## Material Curves

See Time Effects on Materials [in *LARSA 2000/4th Dimension: Staged Construction Analysis*] 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 bS 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 bF 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 bD 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. To set the day of construction, see Setting Up the Model [in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

### **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Time Effects on Materials [in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

## 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. UCSs may be rectangular, cylindrical, or spherical.

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 [p54]), 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 [p66] and isolators [p69] 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.

### 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 fields:

**Name**

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

### **Type**

The coordinate system type can be rectangular, cylindrical, or spherical. 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.

### **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.

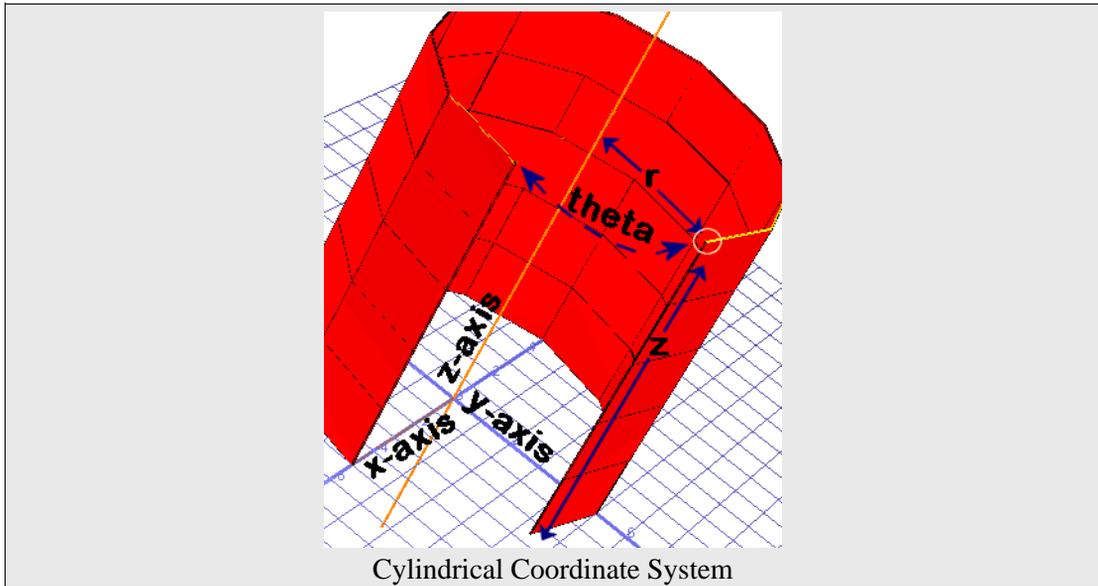
The x-axis of the UCS is the vector from the Origin to the Axis Point.

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



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.

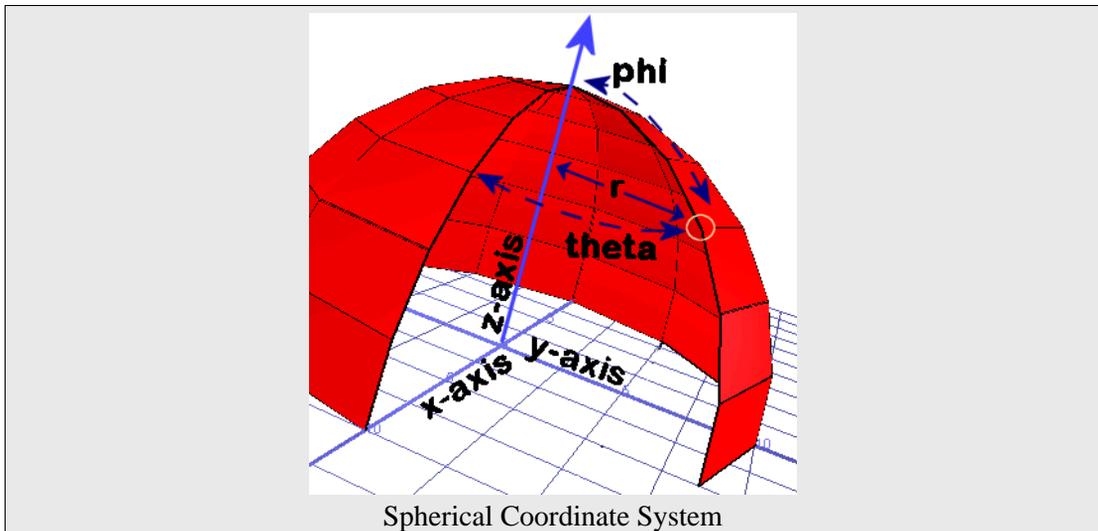
$\theta$  is the projected angle from the  $x$ -axis to a point, generally expressed in degrees.

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

## 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.



---

### For More Information

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

# 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.

<b>Joints</b>	<b>54</b>
<b>Members</b>	<b>57</b>
<b>Spans</b>	<b>62</b>
<b>Plates</b>	<b>63</b>
<b>Springs</b>	<b>66</b>
<b>Isolators and Bearings</b>	<b>69</b>
<b>Mass Elements</b>	<b>70</b>
<b>Slave/Master Constraints</b>	<b>72</b>
<b>Tendons</b>	<b>74</b>
<b>Lanes</b>	<b>76</b>
<b>Tendon and Lane Paths</b>	<b>77</b>

---

## 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 [p49] is the active coordinate system, then the coordinates are specified with respect to that coordinate system. (See Working with User Coordinate Systems [in *LARSA 2000 User's Guide*].)

### 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 2000 User's Guide*] 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 [p92] should be applied.

Joints can be constrained using Slave/Master Constraints [p72]. Slaved degrees of freedom are removed from the system of equations to be solved.

To model other support conditions, such as soil-structure interaction, see The Spring Element: A Multipurpose Element [p19].

## **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 [p49] can be assigned to a joint as the displacement coordinate system.

Supports, spring elements, constraints such as slave/masters, joint loads, and support displacements all act in the directions specified by the joint displacement coordinate system.

**For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- User Coordinate Systems [p49].

## 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. For general information on the member element, see also The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements [p10].

### 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. (See The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements [p10] for more information on the element types.) 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.

#### **Section**

The sectional properties of members are established by assigning a section type [p32] from the list of sections defined in the project. The section's name is used here.

For members with prismatic section properties (cross-sectional properties are constant throughout the beam), use only the Section at Start property and leave the Section at End property set to *none*.

For members with nonprismatic sectional properties, there are two options. Two different section properties can be set for the start and for the end of the member using Section at Start and Section at End. LARSA will linearly interpolate the sectional properties throughout the section.

Alternatively, a nonprismatic section defined in the Section Composer can be applied to the member. In this case, assign the section to Section at Start and leave Section at End set to *none*.

### Material

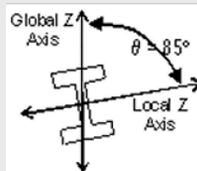
The material properties of members are established by assigning a material type [p29]. The material's name is used here.

### 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 [p74].

### Rigid Zones

The length along the member from the start or end joint, or both, which is to be considered infinitely stiff. Rigid zones are specified either as a proportion of the length of the member, with a maximum value of 0.5 each (half of the length of the member), or as an absolute length. While the total member length is from joint to joint, the member length used in stiffness computation is the length of the elastic portion of the member.



Member Orientation Angle

### Orientation Angle

Members can be situated in any angle about their lateral axes, such as with flanges horizontal or vertical. The orientation angle of members 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. Member sectional properties are given in member local directions. Thus a member whose section's  $I_{zz}$  is greater than its  $I_{yy}$ , which is the case for standard I-shape sections, has its strong axis in its *local* z-axis direction.

For horizontal beams with standard library sections, specify an orientation angle of 90 degrees if the global z-axis is up.

See the section on member local axes below for a detailed explanation of local axes.

### **End Offsets (Rigid Links)**

End offsets can be applied to move the end-points of the member relative to the locations of the end joints. 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.

### **End Releases**

The end connections of beam elements are by default perfectly rigid. All directions except  $F_y$  and  $F_z$  may be released. Semi-rigid releases can be applied only for  $M_y$  and  $M_z$ . The release components are in the member local coordinate system. Semi-rigid end connections are specified as a percent of fixity. A value of 0 (Free) represents a perfect hinge and 100 (Fixed) represents full fixity.  $F_y$  and  $F_z$  can be released using slave/master constraints [p72].

### **Casting Day**

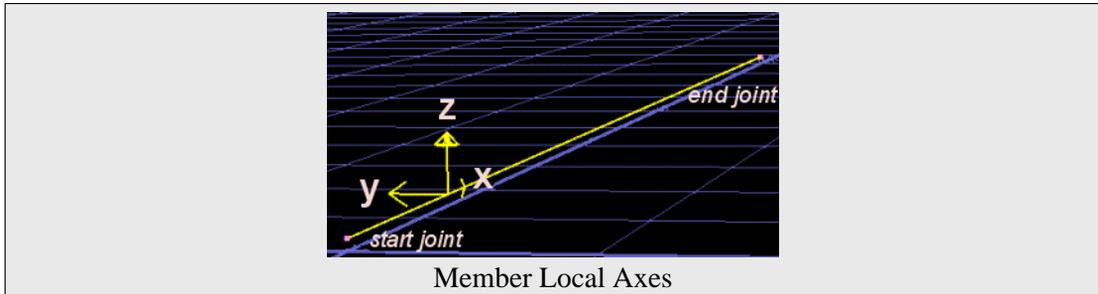
The day the member is cast, for use in the Time-Dependent Staged Construction Analysis. The age of the member when constructed is determined by subtracting the casting day from the day of construction set for the construction stage being analyzed. The casting day of a member must be less than the day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

### **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*.

## Member Local Axes

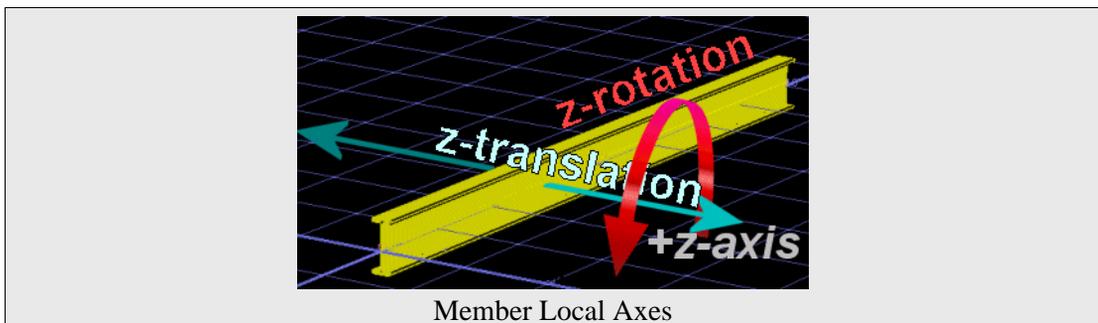


Members have local axes for the purposes of loading and reporting forces and stresses. Local axes of members determine the orientation of a member in space: which way the wedges and flanges are pointing, for instance. The orientation of the local axes is specified by the orientation angle property of members.

The origin of a local member coordinate system is at the member's start joint.

The positive direction of the local x-axis of a member is directed from its start joint to its end joint.

The local y-axis is perpendicular to the x- and z-axes.



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

For differentiation, call the global coordinate axes  $\mathbf{X}$ ,  $\mathbf{Y}$ ,  $\mathbf{Z}$  and the local member axes  $\mathbf{x}$ ,  $\mathbf{y}$ ,  $\mathbf{z}$ . The orientation angle is defined as follows:

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

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

- When the member  $\mathbf{x}$ -axis is in the same direction as the global  $\mathbf{Z}$ -axis, the orientation angle is defined as the angle from the minus  $\mathbf{X}$ -axis to the  $\mathbf{z}$ -axis of the member.
- When the member  $\mathbf{x}$ -axis is in the direction opposite to the global  $\mathbf{Z}$ -axis, the orientation angle is defined as the angle from positive  $\mathbf{X}$ -axis to the  $\mathbf{z}$ -axis of the member.

The local member axes are not necessarily principal axes. If the product of inertia  $I_{yz}$  is zero for the member, the local axes coincide with the principal axes of the member.

---

### For More Information

- The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements [p10].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Joints [p54].
- Materials [p29].
- Sections [p32].

## **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.

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 2000 User's Guide*] and for obtaining analysis results [p198].

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 [p57].

---

### **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Members [p57].
- Steel Design [in *LARSA 2000 User's Guide*].

## Plates

The plate and shell element can be used to model slabs, foundations, walls, and shells in structures. This element is a combination of membrane and pure bending elements and can be either triangular or quadrilateral. For general information on the plate element, see The Plate Element: Plates and Shells [p17].

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 a non-zero area and thickness.

### Properties 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.

#### Type

The element can be used as Bending Only or Membrane Only or as a Shell element to act in both bending and membrane action. The element can also be set as Inactive, and it will not contribute any stiffness to the model.

#### 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. The Elastic Modulus and Shear Modulus must be defined such that the Poisson's Ratio of the material is less than 0.50. Material properties are constant throughout the plate.

### Material Angle

The angle for elements with materials with orthotropic properties.

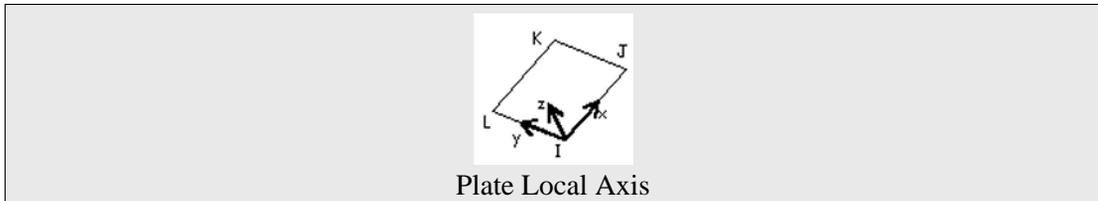
### Thickness

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

### Creep & Shrinkage

A time-dependent material definition [p44] specifying creep and shrinkage material properties.

## Plate Local Axes



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 *thickness/2* into the positive-z direction and *thickness/2* into the negative-z direction.

### **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Materials [p29].
- Joints [p54].

## Springs

The spring element types include both two-node springs and one-node grounded springs, both of which can have linear, nonlinear, and inelastic (hysteretic) material behavior. For general information about the spring element, see The Spring Element: A Multipurpose Element [p19].

### 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 The Spring Element: A Multipurpose Element [p19].

### Direction

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

**Local Axial & Local Torsional** - Used with 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 [p54], or along or about a user coordinate system [p49] 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.

**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.

**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.

## **Nonlinear Spring Properties**

The following fields are only used with nonlinear springs in the nonlinear analysis.

**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.

**Spring Properties Definition**

The string property definition specifies other spring properties [p36].

---

## **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- The Spring Element: A Multipurpose Element [p19].

## **LARSA 2000 Reference**

---

- Spring Property Definitions [p36].
- Joints [p54].

## Isolators and Bearings

Isolators and bearings are two-node elements often used for protection from seismic forces. For general information on these elements, see also *Isolators and Bearings: The Seismic Elements* [p22].

### ID

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

### Start (I) and End (J) Joints

An isolator/bearing's connectivity is described using two non-coincident joints. Isolators and bearings may not have zero length. The joints' IDs are used here.

### Type

The behavior of the isolator or bearing is determined by the isolator/bearing property definition [p40] selected for this field. The property definition determines the type and material constants associated with the isolator or bearing.

### Direction

The direction of action of the isolator or bearing.  
[This information has not yet been entered into the documentation. We apologize for the delay.]

---

## For More Information

- For help on using spreadsheets, see *Using the Model Spreadsheets* [in *LARSA 2000 User's Guide*].
- *Isolators and Bearings: The Seismic Elements* [p22].
- *Isolator/Bearing Property Definitions* [p40].

## 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 [p54] 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**

## LARSA 2000 Reference

---

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Joints [p54].

## Slave/Master Constraints

Slave/masters can be used to enforce equal-displacement behavior of two joints in any degree of freedom.

A practical use of slave/master 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 slaved 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 slave and master joints. When no displacement coordinate systems are specified, the global coordinate system is used.

When the displacement directions of a slave and master joint 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 slave 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 slave/master for X-Translation between these joints will cause the slave joint's global y-displacement to be equal to the master joint's global z-displacement.

### **Slave Joint ID**

The joint ID of the slaved joint. The slave joint will move together with equal displacement as of the master joint specified for each direction (degree of freedom). The slave joint will move independently in the degrees of freedom where a master joint is not specified.

### **Master Joint for X-Translation**

If a Joint ID is specified, then the X-Translation of the Slave Joint will be equal to X-Translation of this joint.

**Master Joint for Y-Translation**

If a Joint ID is specified, then the Y-Translation of the Slave Joint will be equal to Y-Translation of this joint.

**Master Joint for Z-Translation**

If a Joint ID is specified, then the Z-Translation of the Slave Joint will be equal to Z-Translation of this joint.

**Master Joint for X-Rotation**

If a Joint ID is specified, then the X-Rotation of the Slave Joint will be equal to X-Rotation of this joint.

**Master Joint for Y-Rotation**

If a Joint ID is specified, then the Y-Rotation of the Slave Joint will be equal to Y-Rotation of this joint.

**Master Joint for Z-Rotation**

If a Joint ID is specified, then the Z-Rotation of the Slave Joint will be equal to Z-Rotation of this joint.

---

**For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Joints [p54].

## Tendons

Tendons can be included in a structure for modeling the post-tensioning of members. Tendons are generally used in the Staged Construction Analysis [see "Staged Construction Guide" in *LARSA 2000/4th Dimension: Staged Construction Analysis*]. For a general overview of tendons, see The Tendon Pseudoelement [p25].

Post-tensioning 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.

The definition of tendon paths is described in Tendon and Lane Paths [p77].

### **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 [p29] of the tendon.

### **Strand Area**

Cross-sectional area of each strand.

### **# of Strands**

Total number of strands in the tendon.

### **Initial Jacking Force**

The initial jacking force applied to the tendon.

### **Adjusted Jacking Force (at Start/End)**

Automatically computed based on the initial jacking force, anchor set, and peak stress. This field must be manually updated by the user by right-clicking the tendons spreadsheet.

### **Jacking Applied**

The application point or sequence of the jacking force: At Start, At End, or At Start

Then End.

### **Anchor Set**

The anchor set.

### **Wobble Coefficient**

The wobble coefficient. Wobble friction is due to unintentional misalignment. It is specified per unit length.

### **Curvature Friction Coefficient**

The curvature friction coefficient, which is due to the intentional curvature of the tendon. This coefficient is unitless.

### **Peak Stress**

The limiting stress for the tendon, used to compute the Adjusted Jacking Force.

### **Elongation**

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

---

## **For More Information**

- The Tendon Pseudoelement [p25].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- For help defining the path of tendons, see Tendon and Lane Paths [p77].

## **Lanes**

Lanes define loading paths for vehicles in the moving load analysis [p150] and for influence-based analysis [p152].

When lanes are used with the standard moving load analysis or for influence line-based results, lane paths must follow a set members in the model. When load patterns are marched over the lane, the loads are transferred directly to the members that define the lane path. When a lane is not directly over a member, eccentricities are accounted for through moment-type loads.

Lanes can also be used for influence surface-based results, in which case the lane path must be specified with respect to plates. In this case, the width of the lane should be specified.

Lanes are defined before specifying the vehicle types and parameters used in moving load cases.

The geometric location of the lane with respect to the structure is specified as a path [p77]. Paths can take any arbitrary path through 3D space, and automatic curve fitting may be applied.

### **Lane Name**

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

### **Lane Width**

The actual width of the lane where loads can be positioned. This data is used for influence surface computations.

---

## **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- For help defining the path of lanes, see Tendon and Lane Paths [p77].
- Moving Load Analysis [p150].

## Tendon and Lane Paths

This section describes how to define paths for both tendons and lanes.

There are two types of paths: those that follow a series of members and those that follow a series of plates. Influence surfaces require the use of the latter.

Paths consist of an ordered list of *geometry points* that determine the points in space that the path follows. A path can contain any number of geometry points. Geometry points are specified with reference to the member or plate that each point falls in.

For member-based paths, in addition to specifying the coordinates of geometry points, each member that the path passes through must also be specified. Placeholders called *path-only points* are used in the path to specify extra members that the path passes through without specifying any geometry points in those members. Plate-based paths do not have this restriction. Intermediate plates in a path do not need to be specified.

By default, the path will consist of straight line segments connecting each of the geometry points. Curve fitting using several methods may be applied at geometry points to smooth out transitions between segments.

For this example path,

	<b>Point Type</b>	<b>Member</b>	<b>Coordinate</b>
1	Geometry	10	(0, 1, 0)
2	Path Only	11-14	
3	Geometry	15	(0, -1, 0)
4	Path Only	16	
5	Geometry	17	(0, -1, -1)

the path starts in member 10, goes to member 11, then 12, then 13, and through 17. Each member that the path passes through is specified in order in the spreadsheet. It would be incorrect for row 2's *Member* cell to read *14-11*. It would also be incorrect for all of the path-only points to be specified after the geometry points, as the ordering of all of the members in the spreadsheet would be incorrect. In addition, the tendon or lane will pass through the three coordinates, which are specified in the point's member's local coordinate system by default.

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

### Point Type

Determines whether this is a geometry or path-only point.

- **Geometry.** This point will specify a coordinate through which the path will pass through.
- **Path-only.** This point will act as a placeholder to specify a member that the path will pass through. Path-only points are not required for plates
- **Span Break.** Used to indicate a break between two spans of a bridge, which is used for influence-based analysis. Span Break points do not affect the geometry of the path.

### Reference Object Type

Determines whether this point is within a member or a 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 coordinate system of this member or plate. Specify a range of IDs, such as *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 coordinates of this point in the **local** coordinate system of the reference member or plate.

- The **X** offset is specified in coordinate units, for convenience, while the **Y** and **Z** offsets are specified in section units.
- By default, the offset coordinates are with respect to the starting point of the member. This can be changed with the **X, Y, and Z Reference** fields.

The X, Y, and Z Reference fields are only applicable for member-referenced points. For plate-referenced points, the offset coordinate is always in the plate local coordinate system.

### X Reference

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

- **Start.** The **X Offset** is the x-distance from the start of the member.
- **End.** The **X Offset** is the x-distance from the end of the member. Positive values for **X Offset** are *within* the member.

### Y Reference

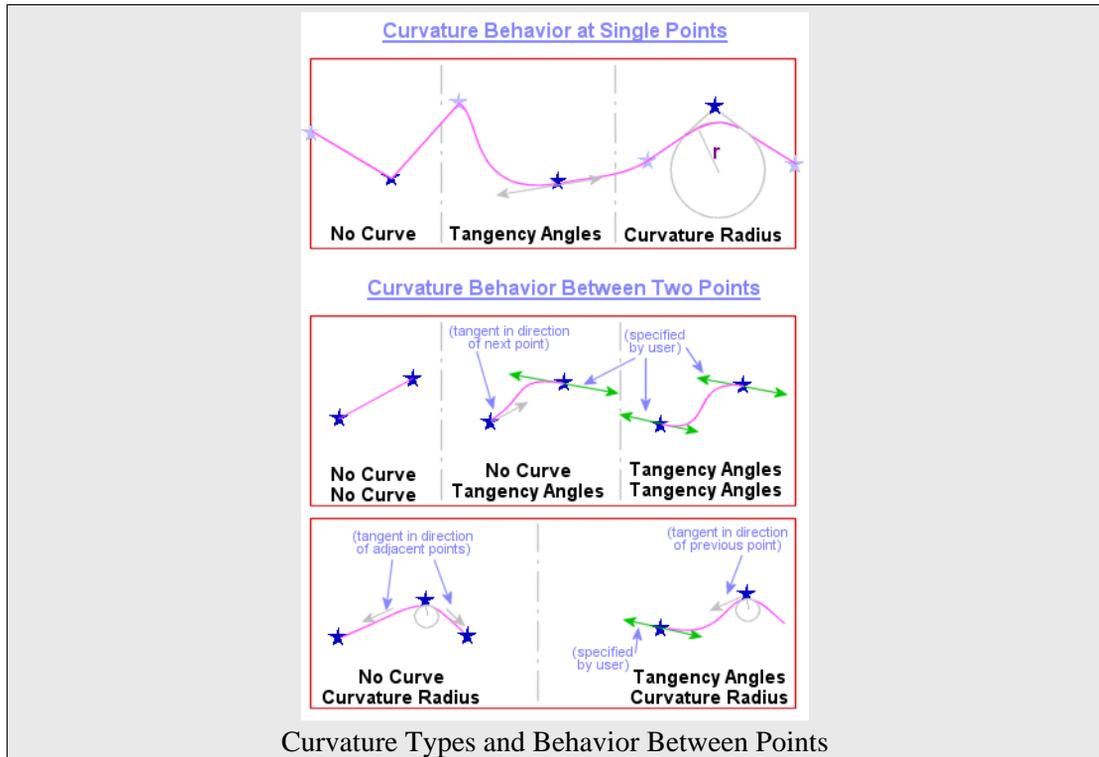
The position that the **Y Offset** starts from.

- **COG.** The **Y Offset** is the y-distance from the centerline of the member (the center of gravity, generally speaking). Positive values are "up" or in the positive direction of the member y-axis.
- **+Y1 (Top).** 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.
- **-Y3 (Bottom).** 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.

### Z Reference

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

- **COG.** 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 (Right).** 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 "left" into the member.
- **-Z3 (Left).** 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 "right" into the member.



### 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 path turns. The path is straight before and after this point.

**Tangency Angles.** The direction of a tangent line to the path at this point is specified. A cubic curve is fit to the path before and after this point so that the path makes a smooth transition from the left to the right of this point.

- The tangent line is specified as two angles, one in the local XZ plane and one in the local YZ plane.
- Two independent cubic equations are solved in separate planes and then combined to produce the three-dimensional curve. Tangent directions perpendicular to the planes

used to solve the equations may not be specified. Depending on what planes are used to solve the cubic equations, some angles will not be accepted.

**Curvature Radius.** A sharp corner at this point is smoothed using the arc of a circle. The radius of the arc is specified.

- The path will not actually pass through this point. The arc is fit into the corner so that its end-points are tangent with each side.
- The point specified is the PI point. The PT and PC points will be computed automatically. The PT point is the point on the line from the previous geometry point to this point where the generated arc is tangent to the line. The PC point is similarly defined for the line from this point to the next geometry point.

The behavior of curve fitting between two points depends on the type of curve fitting at each point.

---

### For More Information

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Tendons [p74].
- Lanes [p76].
- Sections [p32].
- Members [p57].

# **Loads**

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 [p108], P-Delta Analysis [p111], and Nonlinear Static Analysis [p114]) require static load cases and load combinations of static load cases. Eigenvalue analyses [p130] do not make use of load cases at all. The moving load analyses (Moving Load Analysis [p150] and Influence-Based Analysis [p152]) require moving load cases with moving loads. The time history analyses [p141] require a time history load case with time history excitation loads and, for some models, initial conditions.

The Response Spectra Analysis [p132] can be performed with or without static load cases, but it requires a response spectra case with setup information.

In Staged Construction Analysis [p172], 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.

<b>Static Load Cases</b>	<b>84</b>
<b>Load Combinations</b>	<b>86</b>
<b>Self Weight</b>	<b>89</b>
<b>Joint Loads</b>	<b>91</b>
<b>Support Displacements</b>	<b>92</b>
<b>Member Loads</b>	<b>93</b>
<b>Member Thermal Loads</b>	<b>96</b>
<b>Plate Loads</b>	<b>98</b>

---

## **LARSA 2000 Reference**

---

<b>Moving Loads</b>	<b>100</b>
<b>Time History Loads</b>	<b>102</b>

---

## Static Load Cases

Each static load case represents one scenario of static loading.

The following loads may be applied in static load cases.

- Self Weight [p89] for automatic computation of gravitational loading based on section and material properties of elements.
- Joint Loads [p91]: Forces and moments applied at joints
- Support Displacements [p92]
- Member Loads [p93] on beams (point loads, uniformly or partially distributed loads, and triangular loads)
- Member Thermal Loads [p96]: Uniform thermal loads for beam, truss, and cable elements, and thermal gradients for beam elements
- Plate Loads [p98]: Concentrated and uniform loads applied on plate and shell elements
- Thermal Plate Loads [p98]: Temperature change through the thickness of plate and shell elements resulting in uniform and thermal gradients
- Pre- and post-tensioning of members such as cables

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

### 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 2000 User's Guide*].

To create code-based load combinations, the Automatic Code-Based Load Combinations [in *LARSA 2000 User's Guide*] tool is useful. Class-based cumulative results can be obtained in the results spreadsheets [see "Results Spreadsheets" in *LARSA 2000 User's Guide*] in staged construction analysis.

### **For More Information**

- For help creating and editing load cases, see Load Cases Explorer [in *LARSA 2000 User's Guide*].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Analysis Reference [p105].

## 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 2000 User's Guide*] 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 2000 User's Guide*] can also be used to combine results. Automatic Code-Based Load Combinations [in *LARSA 2000 User's Guide*] can be used to automatically create code-based combinations.

### Including Response Spectra Cases

Response Spectra Analysis [p132] 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 [p150] 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-Based Analysis [p152] 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 [p114]**

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 [p120] 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**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000*]

## LARSA 2000 Reference

---

- User's Guide*].
- For help on using the Load Cases Explorer, see Load Cases Explorer [in *LARSA 2000 User's Guide*].
- Linear Result Combinations [in *LARSA 2000 User's Guide*].

## Self Weight

The automatic computation of self-weight (gravity loading) can be included in static load cases [p84] 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 [p70].

### Self Weight Computation

The weight of members [p10] are computed using the cross sectional area, element length and weight density. The weight of plates [p17] are computed using the element volume and the weight density.

The self-weight of beam elements are modeled using uniformly distributed loads. This

## **LARSA 2000 Reference**

---

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.

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.

---

### **For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## 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 [p84].

### 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.

---

## For More Information

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Joints [p54].

## Support Displacements

Displacement loads and support settlements 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.

Joint Displacement Loads can only be applied to joints in degrees of freedom that are fixed at the joint.

Support displacement loads are static loads and are applied in static load cases [p84]. A joint can have only one support displacement in a load case.

### **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

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## 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** - Pre-tensioning force, primarily applied to cable elements.
- **Post-Tension** - Post-tensioning force, primarily applied to cable elements.
- **Jacking Elongation** - Simulates the jacking operation in cable tensioning.

The pre- and post-tension load types are only used in a nonlinear analysis. They are 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. 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.

The jacking elongation load specifies the change in the length of the member, which is used only in nonlinear analyses.

### **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

## **LARSA 2000 Reference**

---

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.

---

## **For More Information**

- Members [p57].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## Member Thermal Loads

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

For linearly varying loads, the variation is from top to bottom or left to right, not along the length of the member. Along the length of the member, the temperature change is constant for both uniform and non-uniform thermal loads.

Uniform thermal loading produces axial strains only. When the thermal load is not uniform, thermal loading produces both axial and bending strains. Only uniform thermal loads should be applied to truss and cable elements.

For non-uniform thermal loads, the section stress points [p32] of the member must be specified as the most extreme points on the section.

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 for the element.

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

### Member

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

### Type

The type of the load: Uniform or Linear.

### Variation Direction

For linearly varying temperature change, 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 along the member. For linearly varying loads, specifies the temperature change at the top edge (+y) or right edge (+z) of the member.

### 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.

---

### **For More Information**

- Members [p57].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## Plate Loads

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

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.

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 assumed to be constant over 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.

Temperature increase is positive for uniform temperature change. The gradient is positive when the top (+z side) is subject to higher (warmer) temperatures than the bottom (-z side) of the element.

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

### Plate

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

### Type

Determines the type of the load.

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

### 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.

**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.

### **Magnitude**

The magnitude of the load. For uniform loads, the load is constant throughout the plate.

### **Relative X Location (x/XL)**

For point loads, the relative distance of the load with respect to the I-J side of the element. Specify 0 if the point is on the I-L side of the plate, 1 if the point is on the J-K side, or a value in between.

### **Relative Y Location (x/XL)**

For point loads, the relative distance of the load with respect to the I-L side of the element. Specify 0 if the point is on the I-J side of the plate, 1 if the point is on the K-L side, or a value in between.

### **Uniform Temperature**

The uniform temperature change.

### **Temperature Gradient**

Temperature gradient is the temperature change through the thickness of the element specified as per unit thickness. If a plate 3 inches thick is 60 degrees warmer on its top (+z edge) than its bottom (-z edge), then the temperature gradient is +20 degrees per inch.

---

## **For More Information**

- Plates [p63].
- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].

## **Moving Loads**

Moving loads are added into moving load cases for the Moving Load Analysis [p150]. These loads specify how to generate the result cases for a vehicle moving on a user-defined path (lane) on a structure. These loads are also used for Influence-Based Analysis [p152].

The analysis requires a database file containing the load patterns [see "Connecting Databases" in *LARSA 2000 User's Guide*], geometry definition of lanes [p76], 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 [p86] or after the analysis using Linear Result Combinations [in *LARSA 2000 User's Guide*].

For Influence-Based Analysis [p152], a separate moving load case is required for each lane to be considered. In each load case, a single moving load is applied to that lane. The load pattern must be "Influence Load," the factor must be 1.0, the direction of travel must be forward, and the start position must be 0.0. The direction and position increment may be set by the user. In some circumstance, it may be appropriate to apply two or more moving loads into a single moving load case to simulate the distribution of vehicle axle loads onto multiple girders. In these cases, the factors should sum to 1.0 and the lanes must all be the same length.

### **Lane Loaded**

The lane [p76] 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 2000 User's Guide*].

### **Applied Load Direction**

The direction of the load application. The direction can be in the local coordinate

## **LARSA 2000 Reference**

---

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, backward, or both, meaning 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.

### **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**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Lanes [p76].
- Standard Vehicles [p167].
- Moving Load Analysis [p150].
- Connecting Databases [in *LARSA 2000 User's Guide*].

## 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 [p141].

### Excitation Functions

Time-based excitation functions are applied at joints in any of the six directions. Multiple excitation functions can simultaneously be applied in more than one direction at the same joint. Displacement based excitation functions can be applied at the support joints and force based excitation functions can be applied at joints free to move.

Uniform base excitation can also be used for the nonlinear time history analysis. See Nonlinear Time-History Analysis Options [p147] for more information.

Before entering the load data for a time history case, the database file containing the time history curves must be connected to the project. See Connecting Databases [in *LARSA 2000 User's Guide*].

The data required for excitations is below:

#### Joint

The ID of the joint to which the excitation will be applied.

#### Time History Curve

The time history records curve to be applied at this joint, selected from all of the time history curves in databases connected to the project.

- Displacement Functions: May only be applied at joints restrained in the direction of the excitation, which are usually supports.
- Force Functions: May only be applied at joints free in the direction of the excitation.
- Acceleration Functions: May only be applied at joints free in the direction of the excitation. The equivalent force is computed as acceleration multiplied by nodal mass in the applied direction.

A single time history case must have either displacement or force/acceleration curve types.

### **Applied Direction**

This is the direction of the applied load. The excitation function is applied in the joint's displacement coordinate system, which is by default the Global Coordinate System.

- **Translation X/Y/Z** - Indicates the direction of the translational excitation function, for force, displacement, or acceleration curves.
- **Rotation X/Y/Z** - Indicates the direction of the rotational excitation function, for force, displacement, or acceleration curves.

## **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.

**For More Information**

- For help on using spreadsheets, see Using the Model Spreadsheets [in *LARSA 2000 User's Guide*].
- Linear Time History Analysis [p141].
- Connecting Databases [in *LARSA 2000 User's Guide*].

## *Part II*

# **Analysis Reference**

Each of the analysis types available in LARSA 2000 are described in the sections below.

<b>Static Analysis</b>	<b>107</b>
Linear Static Analysis	108
P-Delta Analysis	111
Nonlinear Static Analysis	114
Nonlinear Buckling Analysis	120
Nonlinear Pushover Analysis	122
Nonlinear Static Analysis Options	123
Restart Nonlinear and Nonlinear Buckling Analysis Options	125
Nonlinear Pushover Analysis Options	127
<b>Eigenvalue Analysis</b>	<b>129</b>
Eigenvalue and Stressed Eigenvalue Analysis	130
Response Spectra Analysis	132
Eigenvalue Analysis Options	138
Stressed Eigenvalue Analysis Options	139
<b>Time History Analysis</b>	<b>140</b>
Linear Time History Analysis	141
Nonlinear Time History Analysis	143
Time History Analysis Options	145
Nonlinear Time-History Analysis Options	147
<b>Live Load Analysis</b>	<b>149</b>
Moving Load Analysis	150
Influence-Based Analysis	152
Influence Method Options	159
Influence Lines	162

---

## **LARSA 2000 Reference**

---

Influence Surfaces	165
Standard Vehicles	167
Moving Load Analysis Options	171
<b>Staged Construction Analysis</b>	<b>172</b>

---

## Static Analysis

<b>Linear Static Analysis</b>	<b>108</b>
<b>P-Delta Analysis</b>	<b>111</b>
<b>Nonlinear Static Analysis</b>	<b>114</b>
<b>Nonlinear Buckling Analysis</b>	<b>120</b>
<b>Nonlinear Pushover Analysis</b>	<b>122</b>
<b>Nonlinear Static Analysis Options</b>	<b>123</b>
<b>Restart Nonlinear and Nonlinear Buckling Analysis Options</b>	<b>125</b>
<b>Nonlinear Pushover Analysis Options</b>	<b>127</b>

---

## 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 [p84] whose types are set to *Static* and factored combination of these load cases (load combinations [p86]). 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
- Member sectional forces, stresses, and displacements
- Plate center forces and stresses, and joint forces
- Spring forces

### **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:

$$\mathbf{K_t U} = \mathbf{R}$$

where

U = Joint displacement vector

R = Applied joint force vector

with the tangent stiffness matrix, in the following form:

$$\mathbf{K_t} = \mathbf{K_o} + \mathbf{K_g} + \mathbf{K_s}$$

where

- $\mathbf{K_t}$  = Tangent stiffness
- $\mathbf{K_o}$  = Small displacement stiffness
- $\mathbf{K_g}$  = Large displacement stiffness for geometry change
- $\mathbf{K_s}$  = 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 ( $\mathbf{K_g}$ ) and stress stiffening ( $\mathbf{K_s}$ ) are not considered. Then tangent stiffness matrix ( $\mathbf{K_t}$ ) becomes the same as the stiffness matrix corresponding to linear small displacement behavior:

$$\mathbf{K_t} = \mathbf{K_o}$$

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

- Running an Analysis [in *LARSA 2000 User's Guide*].

## **P-Delta Analysis**

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  $1.2*(\text{Dead Load}) + 1.6*(\text{Live Load})$ , 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**

- Running an Analysis [in *LARSA 2000 User's Guide*].

## 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 [p122] or in a nonlinear buckling analysis [p120].

The load cases and applied loads are defined exactly in the same manner as for the linear static analysis.

---

### **For More Information**

- Nonlinear Static Analysis Options [p123].
- Nonlinear Buckling Analysis [p120].
- Nonlinear Pushover Analysis [p122].

## **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 [p84].

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 [p86] 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

- Nonlinear Static Analysis [p114].
- Restart Nonlinear and Nonlinear Buckling Analysis Options [p125].

## 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 [p132] 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 [p10] and springs [p19], 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

- Nonlinear Pushover Analysis Options [p127].
- Nonlinear Static Analysis [p114].

# 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**

- Nonlinear Static Analysis [p114].
- Running an Analysis [in *LARSA 2000 User's Guide*].

## Restart Nonlinear and Nonlinear Buckling Analysis Options

The Restart 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 [p123] 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:

	<b>Description</b>	<b>Case</b>	<b>Factor</b>
1	Dead	Dead	1.0
2	Dead + 0.10*Lateral	Lateral	0.10

	<b>Description</b>	<b>Case</b>	<b>Factor</b>
3	Dead + 0.20*Lateral	Lateral	0.10
4	Dead + 0.30*Lateral	Lateral	0.10
5	Dead + 0.40*Lateral	Lateral	0.10

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**

- Nonlinear Static Analysis [p114].
- Running an Analysis [in *LARSA 2000 User's Guide*].

## **Nonlinear Pushover Analysis Options**

Solution convergence control from the nonlinear static analysis [p123] 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**

## **LARSA 2000 Reference**

---

- Running an Analysis [in *LARSA 2000 User's Guide*].

# Eigenvalue Analysis

<b>Eigenvalue and Stressed Eigenvalue Analysis</b>	<b>130</b>
<b>Response Spectra Analysis</b>	<b>132</b>
<b>Eigenvalue Analysis Options</b>	<b>138</b>
<b>Stressed Eigenvalue Analysis Options</b>	<b>139</b>

---

## **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 effect of the self-weight of the elements if weight density is supplied with material data. Mass elements, which are ignored in static analysis, are included in the eigenvalue analysis mass matrix.

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**

- Eigenvalue Analysis Options [p138].
- Stressed Eigenvalue Analysis Options [p139].

## **Response Spectra Analysis**

Response spectra analysis (RSA) is a statistical type of analysis where the peak response of a multi-degree-of-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 2000 User's Guide*] and Connecting Databases [in *LARSA 2000 User's Guide*].

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  $i$ th mode and  $j$ th 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 2000 User's Guide*].

### **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 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 and 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 [p86].

The standard Response Spectra Analysis will require an eigenvalue analysis that has been previously analyzed and saved.

---

## **For More Information**

- Running an Analysis [in *LARSA 2000 User's Guide*].
- Connecting Databases [in *LARSA 2000 User's Guide*].

# Eigenvalue Analysis Options

## 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

- Running an Analysis [in *LARSA 2000 User's Guide*].
- Eigenvalue and Stressed Eigenvalue Analysis [p130].

## Stressed Eigenvalue Analysis Options

Solution convergence control from the nonlinear static analysis [p123] and mode shape options from the standard eigenvalue analysis [p138] 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

- Running an Analysis [in *LARSA 2000 User's Guide*].
- Eigenvalue and Stressed Eigenvalue Analysis [p130].

# Time History Analysis

<b>Linear Time History Analysis</b>	<b>141</b>
<b>Nonlinear Time History Analysis</b>	<b>143</b>
<b>Time History Analysis Options</b>	<b>145</b>
<b>Nonlinear Time-History Analysis Options</b>	<b>147</b>

---

## Linear Time History Analysis

A time history analysis can be performed either as linear or nonlinear time history analysis. This section describes the common aspects of both types.

Time history analysis is for computing the response of the structure to time-dependent loads which are specified as excitation records or curves. Excitation curves may be in the form of force, displacement, or acceleration. Multiple curves can be applied to one or more joints, and there is no upper limit on the number of points in a curve.

### Time History Load Cases

The computations in a time history analysis are organized into time history load cases [p102], which are similar to load cases in the static analysis. Various response cases can be defined as separate load cases and an analysis can be performed on each case. Each time history load case generates a group of result cases for all the integration time steps solved and saved.

Time history loads are specified as excitation curves assigned to joints. Excitation curves are loaded from external time-history databases. See Connecting Databases [in *LARSA 2000 User's Guide*].

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.

### Applied Loads

Time-dependent excitation functions are applied at the joints in any of the six directions. Multiple excitation functions can be applied to the same joint and there is no limitation on the number of joints with force functions applied. The following restrictions apply.

Force Functions applied at a joint restrained in the direction of the applied force are ignored.

Displacement Functions may only be applied at joints restrained in the direction of the

excitation.

For Acceleration Functions applied at joints, the equivalent force is computed as acceleration multiplied by mass in the applied direction.

Uniform Base Acceleration excitations are available in the nonlinear time history analysis.

The time history loads (curves) applied at the joints are specified using the load case's *Time History: Applied Excitation Functions* spreadsheet.

### **Multiple Support Excitation**

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.

---

### **For More Information**

- Nonlinear Time History Analysis [p143].
- Time History Analysis Options [p145].

# **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 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 program has the option for either using the modified Newton-Raphson or full Newton-Raphson solution procedure. Both use iterations within each Newmark-Beta integration time step.

In modified Newton-Raphson method, the element tangent stiffness matrices are evaluated

only at the beginning of each time step. If the stiffness of the elements in the structure do not change much during the time step, the modified Newton-Raphson can satisfy the equilibrium in a few iterations. If the equilibrium can not be satisfied within a few iterations due to changes in the stiffness during a time step, full Newton-Raphson can be used.

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 2000 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 [p141], 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 [p147].

---

### **For More Information**

- Linear Time History Analysis [p141].
- Nonlinear Time-History Analysis Options [p147].

## **Time History Analysis Options**

### **Time Stepping Method**

The integration method is for choosing the type of time stepping used in the integration. Throughout the integration, the time step can be kept constant or can vary using an adaptive procedure.

#### **Constant Step**

The same step size is used for all integration steps.

#### **Adaptive Step**

Available only for the nonlinear time history analysis. The time step size is adjusted within user-specified limitations. Step sizes are based on the number of iterations required for previous steps to converge. Though the step size is automatic, an excessive number of iterations can be prevented.

### **Integration Step Size**

Specify the time step sizes used in Newmark Integration. For constant step size, enter only the initial step size which will be used throughout the integration. For adaptive stepping, provide initial, minimum (smallest), and maximum step sizes to be employed in numerical integration.

#### **Initial**

This is the initial step size for variable and adaptive integration methods and it is the step size used if Constant Step is chosen for the stepping method.

#### **Minimum**

The minimum step size that can be used in the adaptive integration method.

#### **Maximum**

The maximum step size that can be used in the adaptive integration method.

## 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,  $\mathbf{c}$ , is defined as  $a_0\mathbf{m} + a_1\mathbf{k}$ , 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

$$a_0 = (2 r w_i w_j) / (w_i + w_j) \quad a_1 = 2 r / (w_i + w_j) \quad \text{where } w_i \text{ and } w_j \text{ are the two natural frequencies in rad per sec.}$$

Note: Rayleigh damping constants are not the modal damping ratios.

---

## For More Information

- Running an Analysis [in *LARSA 2000 User's Guide*].
- Linear Time History Analysis [p141].

## **Nonlinear Time-History Analysis Options**

Solution convergence control from the nonlinear static analysis [p123] and time history options from the linear time history analysis [p145] apply to this analysis type as well.

### **Excitation Functions**

Uniform base acceleration, multiple support excitation, and force excitation are the options for applied excitation in the nonlinear time history analysis.

#### **Uniform Base Acceleration**

If Uniform Base Acceleration is selected, acceleration type excitation curves must be selected for at least one of the global directions. Curves can be specified for the global x, y, and z directions.

The acceleration is applied uniformly throughout the structure.

#### **Multiple Support Excitation**

Select this option when excitations are displacement functions applied at the support joints.

#### **Force Excitation**

Select this option when excitations are force functions applied at the joints.

### **Iteration Method**

The options are Full Newton-Raphson or Modified Newton-Raphson. The choice depends on the degree of nonlinearity in the structure response; i.e. the more nonlinear the response, the more often updating should be performed. Full Newton-Raphson is recommended for structures with material nonlinearity.

#### **Full Newton Raphson**

A new tangent stiffness matrix is calculated in each iteration.

### **Modified Newton Raphson**

This option involves fewer stiffness reformations than the full Newton-Raphson iteration.

---

### **For More Information**

- Running an Analysis [in *LARSA 2000 User's Guide*].
- Nonlinear Time History Analysis [p143].

## Live Load Analysis

<b>Moving Load Analysis</b>	<b>150</b>
<b>Influence-Based Analysis</b>	<b>152</b>
<b>Influence Method Options</b>	<b>159</b>
<b>Influence Lines</b>	<b>162</b>
<b>Influence Surfaces</b>	<b>165</b>
<b>Standard Vehicles</b>	<b>167</b>
<b>Moving Load Analysis Options</b>	<b>171</b>

---

## Moving Load Analysis

The moving load analysis simulates the movement of a vehicle or multiple vehicles passing over a structure. It provides a powerful method of generating the load cases for a vehicle moving on a user-defined path or vehicles on multiple paths.

This analysis automatically generates all of the load cases for various vehicle positions and performs a linear static analysis for the cases generated. The results generated are normally the ones generated in any linear static analysis.

The results of the generated moving load cases can be combined with any other load case using Load Combinations [p86] or Linear Result Combinations [in *LARSA 2000 User's Guide*], and envelopes are simplified with Extreme Effect Groups [in *LARSA 2000 User's Guide*].

The moving load analysis requires definitions of lanes [p76] and the availability of moving load vehicle patterns in external databases [see "Connecting Databases" in *LARSA 2000 User's Guide*].

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 replaced with an equivalent linear element without modifying the actual model data. See Linear Static Analysis [p108].

### Vehicle Paths: Lanes

Vehicle paths (lanes) [p76] can be defined using beam elements or plate elements.

The torsion effects on beam elements are computed and included if lane eccentricity is specified.

By loading multiple lanes simultaneously with loading factors, the effect of distributing the load of a vehicle across lanes can be simulated.

## **Load Cases for Moving Load Analysis**

The computations in a moving load analysis are organized into load cases similar to load cases in the static analysis. Various moving load cases can be defined as separate load cases and an analysis can be performed on each case in the same analysis run. Each moving load case generates a group of result cases for all the vehicle positions solved and saved.

A moving load case [p100] is required for the moving load analysis. The moving loads are specified as load patterns applied to user-defined lanes. Moving load patterns (vehicle or train loads) are loaded from external moving-load databases. See Connecting Databases [in *LARSA 2000 User's Guide*].

---

### **For More Information**

- Moving Load Analysis Options [p171].
- Influence-Based Analysis [p152].
- Lanes [p76].
- Standard Vehicles [p167].

## Influence-Based Analysis

Influence-based analysis is an extension of the moving load analysis which uses the effects of a unit load to rapidly compute the effects of arbitrary complex load patterns on design lanes.

Influence-based results are often required by bridge design codes, such as the AASHTO and Indian Roads Congress codes. In these codes, it is not sufficient to envelope the results of a single vehicle placed at each station along a girder. That is, the one variable, *position*, is not sufficient. In the case of the AASHTO code, *vehicle length* must also be considered. The design truck varies from 28 to 44 feet, with the length that induces the most forces considered. In the case of the IRC code, *number of vehicles* must be considered. Bridges are loaded with as many vehicles as will fit to produce the most extreme effects.

These extra variables make it impossible to compute results for every possible value of each variable; there are too many possibilities. Influence is a mechanism for finding the most extreme values without permuting through all possible conditions.

The influence-based method provides considerable power and flexibility, completely eliminating the need for guesswork and the generation of many load conditions. It handles accurate placement of any load pattern, including UDL-type loads for extreme effects. American AASHTO LRFD 2000 standards, British (UK) BS5400, Indian Roads Congress, and other codes are supported.

### Overview

The influence-based analysis is performed in the following manner:

- Lanes [p76] are defined specifying the path of vehicles on the structure.
- A moving load [p100] is set up on each lane using the special load pattern "Influence Load." The influence load is simply a point load with a magnitude of 1.
- LARSA will generate an influence line for lanes defined relative to members, and influence surface for lanes defined relative to plates.
- Influence-based cases are set up for each influence line and surface. Vehicle types and loading parameters are specified. These result cases will compute and show the extreme values for the vehicle loading requested.

## Method and Assumptions

The core of an influence-based analysis is the computation of influence coefficients. These coefficients represent the effect on the structure of a unit load at each of the positions where a vehicle axle may fall. The effect of a vehicle at an arbitrary position is computed by scaling the influence coefficients corresponding to the vehicle's axles, and then adding them together. From just an analysis of unit loads, the effects of all possible vehicle loading can be determined.

Influence coefficients are computed at equally spaced stations along a path. Because axles can only be applied at positions where unit loads have been computed, axles are always "rounded" to the nearest influence coefficient. The smaller the station spacing, the closer the unit loads will be to the vehicle axles and the less "rounding" will occur.

Influence-based results assume loading of vehicles is linear. That is, the effect of a 10 kip axle is 10 times the effect of a 1 kip axle. And, the effect of multiple axles together is the sum of the effects of each of the axles individually. It is this assumption that allows influence coefficients, the effects of 1-unit axles, to be used to compute the effects of any vehicle.

Load patterns with variable-position axles can be used in influence-based results. In that case, the results will reflect the axle spacing that produced the most extreme effect.

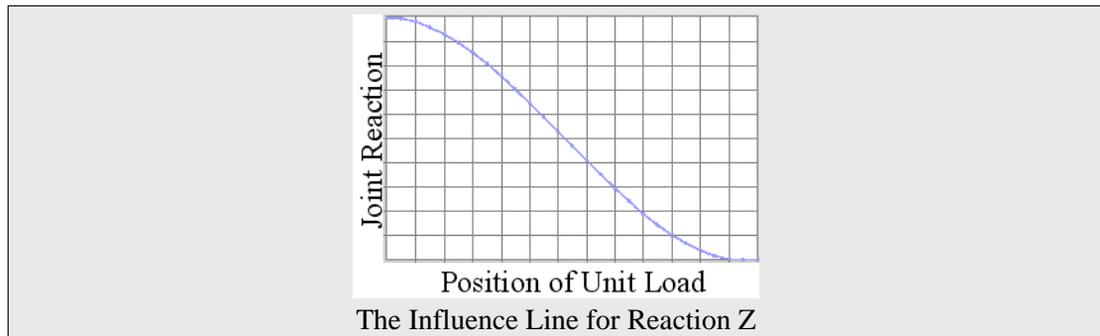
Because of the large number of possible loading situations considered in an influence-based analysis, the influence engine cannot try every possible permutation of the variables involved. Instead, the influence engine uses several reliable techniques for eliminating vehicle scenarios expected to not produce the most extreme values. In some circumstances, the influence engine will yield a loading scenario very close to, but not exactly matching, the true worst-case condition.

## Examples

Example 1. If a 1-kip load at the center of a particular member produces a 5-kip reaction in joint 1, then a 50-kip load on that same member will produce a 250-kip reaction in that same joint. Furthermore, if a 1-kip load on member A produces a 5-kip reaction in joint 1 and separately a 1-kip load on member B produces a 12-kip reaction in joint 1, then a 50-kip load on member A and simultaneously a 15-kip load on member B produces a  $50 \cdot 5 + 15 \cdot 12 = 430$ -kip reaction in joint 1.

Example 2. Consider a member of length 50 ft. Unit loads are placed at 7 ft. intervals and are analyzed separately. The unit loads result in the following reactions for one of the joints:

	<b>Load</b>	<b>0</b>	<b>7</b>	<b>14</b>	<b>21</b>	<b>28</b>	<b>35</b>	<b>42</b>	<b>49</b>
	<b>Position:</b>								
1	Rx (kip)	5.0	6.5	9.0	14.2	17.0	18.0	18.5	19.0
2	Ry (kip)	0	0	0	0	0	0	0	0
3	Rz (kip)	72.0	65	50	30	22	5	-12	-18



The above table is called the influence coefficients. These are the effects on the structure of unit loads placed at various positions. The values for one particular result, say Reaction Z, are called an influence line. The influence line for Reaction Z is plotted in the graph above. The horizontal axis is the position of the unit load, and the vertical axis is the effect on the structure (the reaction) from that load.

Once the influence coefficients are computed, it is the job of the influence engine to find the positions of vehicles that cause the most extreme reactions. When enveloping for the maximum z-reaction at the joint in this example, the influence engine will place a vehicle as far to the left as possible. This will yield the highest reactions.

If a design truck with axles 8 kip, 32 kip, and 32 kip, spaced 14 feet apart is applied, the axles will be placed at stations 0, 14, and 14. Using only the influence coefficients, the z-reaction caused by the vehicle is computed as  $8*72 + 32*50 + 32*22 = 2880$  kip.

When enveloping for the minimum reaction, the influence engine will place the vehicle as far to the right as possible, because in this example that will produce the most negative reaction. The axles will be placed at stations 21, 35, and 49. The z-reaction will be computed as  $8*30 + 32*-5 + 32*-18 = -496$  kip.

## Procedure: Preparing the Analysis

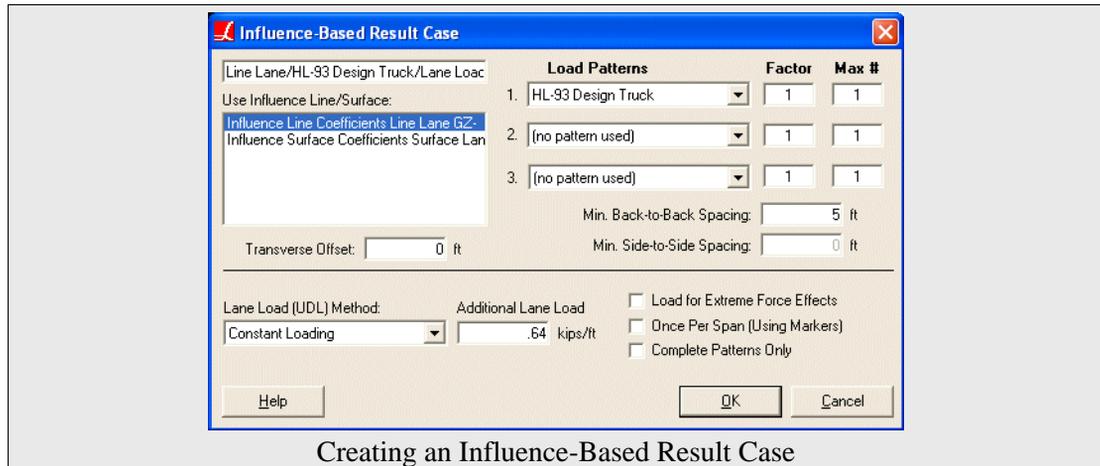
To get influence-based results, a lane definition must be created to represent the path along which vehicles will travel. Lanes for influence-based results can be curved. To use the influence-line method, the lane must follow members only and must have a width of zero. To use the influence-surface method, the lane must follow plates, and the lane must have a non-zero width. (See Lanes [p76] and Paths [p77].)

Moving load cases [p100] are used to specify how the influence coefficients are to be created. Moving load records indicate the lane definition to load and the influence coefficient spacing (*position increment*). Apply the special "Influence Load" load pattern to the lane to invoke the influence-based analysis. The special pattern is found in all of the standard moving load database files, one of which must be connected to the project. (See Connecting Databases [in *LARSA 2000 User's Guide*].)

LARSA will generate all of the influence coefficients when a Moving Load Analysis [p150] is run. Only after the analysis is run is the vehicle type determined.

## Procedure: Vehicle Loading Parameters

The analysis computes influence coefficients. Influence-based result cases are used to find the worst-case conditions for vehicle loading. To create an influence-based case, open the **Results** menu and choose **Influence-Based Case**. This is where the lane and load pattern, and other options, are chosen.



### Use Influence Line/Surface

Choose the lane from the list on the left, which will show all influence coefficient results already generated.

### Load Patterns

Choose up to three different load patterns to apply. Load patterns, such as HL-93 Design Truck, are presented from the connected databases. LARSA will apply as many as can be placed on the lane simultaneously so as to produce the most extreme effect, starting with pattern 1 going down the list.

Normally codes require only one load pattern to be used at a time. In this case, only make a selection for pattern 1.

### Factor and Max #

For each load pattern, a factor may be applied to the magnitudes of the loads. In addition, specify the maximum number of times to apply the load pattern to the lane. If *Max #* is given as 5, then LARSA will place *at most* five "trucks" based on that load pattern on the lane.

Many codes, including AASHTO LFD, only require placing one truck on a lane at a time. In this case, set *Max #* to 1.

### Min. Spacing

A minimum distance between load patterns can also be specified. Min. Back-to-Back

Spacing is the smallest space allowed between the rear-most axle of one vehicle and the start of another. For influence surfaces, the Min. Side-to-Side spacing is the smallest space allowed between the sides of two nearby vehicles. When 0 is set for these options, vehicles can be placed next to each other, but in no case will vehicles overlap.

### **Transverse Offset**

This option is used for influence lines and is discussed in Influence Lines [p162].

### **Lane Load**

Lane loading (a.k.a. patch or UDL loading) can also be entered. The value is the magnitude of the uniformly distributed force. The method of lane loading is described in Influence Method Options [p159].

### **Load for Extreme Force Effects**

When this option is checked, vehicle axles that do not contribute to extreme forces are not considered. For example, when enveloping for maximum forces, axles that cause negative forces are ignored. In addition, regions of lane loading that would not contribute to extreme forces are not loaded. Normally this option should be checked. For more information, see Influence Method Options [p159].

### **Once Per Span**

This option causes LARSA to split the lane into segments as indicated by Span Break Markers in the lane path [p77]. The worst condition for each segment is solved independently, and then combined.

### **Complete Patterns Only**

Often the worst position of a vehicle is when it is only partially on the lane. When this option is checked, only vehicle positions where all axles fall on the lane are considered.

Click OK to create the new case. The case will appear in the Analysis Results Explorer.

## **Procedure: Getting Results**

The influence-based 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 search for a minimum and maximum, LARSA must know which column from a result spreadsheet to envelope on. If LARSA 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 knows which value to envelope on.

Not all graphical results use envelopes automatically. These results, which include deformed model, cannot be viewed for influence-based cases. The other results, such as member stresses, automatically envelope on the variable being displayed. If axial force is being shown graphically, then the influence results will envelope on axial force.

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 time or hard drive space.

---

### **For More Information**

- Moving Load Analysis [p150].
- Lanes [p76].

## Influence Method Options

This section describes lane (patch) loading and the extreme force effects option used in influence-based analysis.

### Lane (Patch) Loading

Lane loads can generally be modeled as a uniform member load. Some codes, such as AASHTO and BS5400, require that the location and/or magnitude of lane loading be dependent on where the loading contributes to the extreme effect. Rather than manually applying uniform member loads to just the members that contribute to the extreme effect for each desired result, influence-based cases compute lane/patch loading according to several codes.

In influence-based cases, the lane load is effectively computed by integrating the area under the influence line and multiplying the result by the magnitude of the lane load.

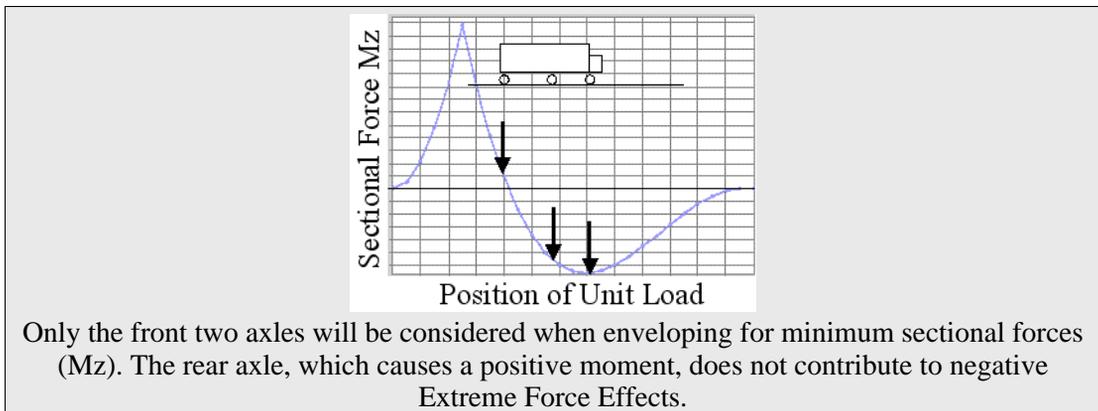
The lane load can optionally be applied as per the BS5400 code, which adjusts the magnitude of the lane load depending on the length of loaded region.

Lane loading is computed by adding together the effect of all of the unit loads within the loaded regions, and scaling the effect appropriately. When used with influence surfaces, lane loading will not be applied at the same location as a vehicle.

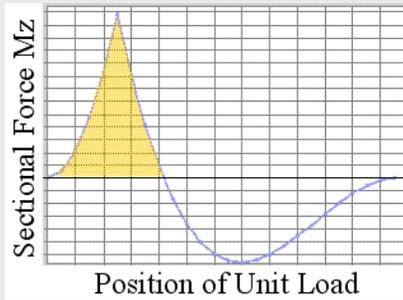
If the Extreme Force Effects option is not specified, then lane loading is applied along the entire length of the lane, without regard to whether the loading will produce the most extreme values.

## Extreme Force Effects

In certain bridge design codes, extreme force effects must be used. When using extreme force effects, loading that does not contribute to the value under consideration must be ignored. In the figure below, the left-most axle would not be included under extreme force effects because a minimum (most negative) moment was being sought, but the axle produced a positive moment. The influence-based moment will only be a combination of the front two axles.



Extreme force effects applies to lane loads as well. For lane loads, only the portion of the lane which contributes to the minimum or maximum under consideration are included in the integration. The shaded area of the figure below depicts the area under the influence line that reflects the lane load for the maximum z-moment in the member using extreme force effects.



The highlighted portion shows the range of influence coefficients used to compute a maximal value when the Extreme Force Effects option is on.

## Influence Lines

The influence line method is appropriate for bridges modeled as girders without a plate deck. Vehicles take straight or curved paths along a series of beams. Influence line results represent the worst-case scenario for vehicles loaded in this way.

To use the influence line method, follow the procedure outlined in Influence-Based Analysis [p152], but note the following restrictions on how to model the lane. See also Influence Surfaces [p165].

### Influence Line Lanes

The paths of lanes for influence lines must consist of only member elements. Influence lines cannot be used to place vehicles directly on a plate-modeled deck. When used with plate-deck models, cross-beams and rigid members will transfer load to other elements as they usually would.

Each axle of the loaded vehicles will be lumped on the centerlines of the members in the lane path. Load distributions across multiple girders can be accomplished either through rigid cross-beams connecting the members in the loaded lane to other girders, or by loading multiple lanes simultaneously in the moving load case used to generate the influence coefficients. When loading multiple lanes simultaneously, specify a set of factors for the lanes that sum to 1.0.

### Transverse Loading

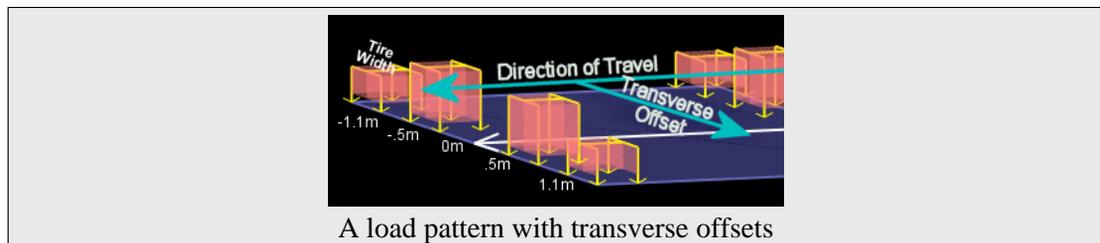
It is possible to use the influence line method when vehicle axles do not fall directly on the lane, but instead have eccentricity relative to the lane. This is called transverse loading, referring to the transverse distance from the lane to the load.

Transverse loading occurs in two circumstances. In the more common circumstance, a moving load pattern is offset from the lane using the *Transverse Offset* option of influence-based result cases. (See Influence-Based Analysis [p152].) When a transverse offset is given, the axle is offset by the distance specified, in the direction that generates

positive axial moment on the member.

A transverse offset given in this manner is used to model a design lane that is offset from the girder that takes its load. This is useful because it eliminates the need to create a lane definition [p76] for each design 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 second instance of transverse loading is in vehicle load patterns whose axles are defined as being transversely offset from the member centerline. Such a load pattern is depicted in the figure below. The axles are distributed loads offset from the centerline. These types of load patterns are called "two dimensional" because they act on a surface.



*Most load patterns in the standard load pattern database do not including any transverse loads: all loads are lumped on the centerline of the lane. Use The Database Editor [in LARSA 2000 User's Guide] to check whether a load pattern is specified as lumped on the centerline or with transverse loads.*

The effects of transverse loading is computed through a second set of influence coefficients which represent the effect of a unit moment at each position along the lane.

If a transverse offset is specified for the influence-based case and a load in a load pattern has a transverse offset, then the sum of the transverse offsets is used when computing the effect of the load. That is, the transverse offset for the influence-based case shifts the transverse offsets of all loads applied.

In both cases of transverse loading, the effect  $R$  of a point load with magnitude  $P$  and transverse offset  $E$  is computed as  $R = Pf + PE_m$ , where  $f$  is the influence coefficient for force at the location of the load, and  $m$  is the influence coefficient for moment at the location of the load.

Influence-based transverse loading **cannot** be computed from an influence line that itself included eccentricity. If the lane is offset from the member centerlines, then the eccentric

## **LARSA 2000 Reference**

---

effect of loads placed directly on that lane are already computed. Additional transverse offsets from the lane should not be specified in the influence-based case.

---

### **For More Information**

- For a tutorial on running an influence-based analysis, see Influence-Based Moving Load Analysis Tutorial [in *LARSA 2000 Samples and Tutorials*].

## Influence Surfaces

The influence surface method is appropriate for bridges with decks modeled as plates. Vehicles take straight or curved paths through the deck of the bridge. Influence surface results represent the worst-case scenario for vehicles loaded in this way.

To use the influence surface method, follow the procedure outlined in Influence-Based Analysis [p152], but note the following restrictions on how to model the lane.

### Influence Surface Lanes

The paths of lanes for influence surfaces must consist of only plate elements. Influence surfaces cannot be used to place vehicles directly on a member.

The lane definition [p76] on which vehicles will be placed must have a non-zero width. Vehicle axles may be placed anywhere within this width. Design code requirements may indicate that loading must only be considered within a region with a width less than the physical width of the bridge. This is the *design width*, and it is what should be specified here.

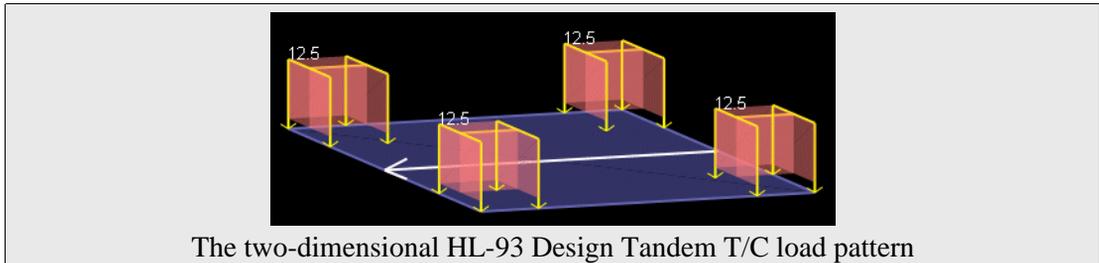
The surface of the lane extends half the width in each direction from the lane centerline. The tilt of the surface, such as in banks, will match the surface of the plates on which the lane is defined.

Every point on the surface must be clearly within a plate element in the model. If a point on the surface is not within a plate, or is on the exact edge of a plate, LARSA will not be able to compute the effect of a unit load at that location. It may be necessary, especially with curved paths, to specify a lane width slightly less than the physical width of the bridge to ensure the lane surface does not extend beyond the edges of plates.

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 an axle throughout the deck, and to any connected members.

## Two-Dimensional Loading

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. The axles can also be 2D-surface loads, rather than point or linearly distributed loads. One such load pattern, the AASHTO HL-93 Design Tandem, is depicted below.



Surface loads in two-dimensional load patterns are applied over the influence surface by summing the influence coefficients that are contained within the loaded region. Because of the granularity of the influence coefficients, the area represented by the coefficients within a loaded region will not exactly match the area of the loaded region specified in the load pattern (transverse width multiplied by the length of the load). The effect of a surface-load axle is always scaled so the intended total load, in force units, is preserved.

## Standard Vehicles

Various standard vehicle types are supplied with LARSA 2000. 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 file `movedata.dml`.

Some of these patterns are two dimensional: Tire contact area is specified as uniformly distributed loads over a rectangular surface at the appropriate transverse coordinates. These load patterns can only be used with influence-based analysis.

- HL-93 Design Truck has three axles of 8, 32, and 32 kips. The spacing between the front two axles is 14 feet, and the spacing between the rear axles varies between 14 and 30 feet.
- HL-93 Design Tandem consists of two 25 kip loads spaced 4 feet apart.
- HL-93 Design Two Trucks consists of two HL-93 design trucks at variable spacing. 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.
- 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.
- HL-93 Design Tandem T/C 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.
- 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.

## 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

## **LARSA 2000 Reference**

---

- Moving Load Analysis [p150].

## Moving Load Analysis Options

Solution convergence options (Nonlinear Static Analysis Options [p123]) apply only to the Nonlinear/Linear and Full Nonlinear moving load analysis types.

---

### For More Information

- Running an Analysis [in *LARSA 2000 User's Guide*].
- Moving Load Analysis [p150].

# 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 [p114] 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.

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 [p114] 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:** Joints are restrained or released (globally or slaved to other joints).
- **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 [p25].

---

### **For More Information**

- Staged Construction Guide [in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

*Part III*  
**Analysis Results  
Reference**

This section is a reference for each type of result produced by the analysis engine.

---

**For More Information**

- Getting Results [in *LARSA 2000 User's Guide*].

<b>Joint Results</b>	<b>176</b>
Joint Displacements	177
Joint Reactions	179
Joint Velocities & Accelerations	181
<b>Member Results</b>	<b>183</b>
Member End Forces	184
Member Sectional Forces	187
Member Stresses	190
Member Displacements	192
Member Plastic Deformation	194
Member Yield and Strains	196
Span Displacements and Forces	198
Analyzed Member Loads	200
<b>Plate Results</b>	<b>202</b>
Plate Forces on Center	203
Plate Forces at Joints	205

---

## LARSA 2000 Reference

---

Plate Stresses	207
<b>Spring Results</b>	<b>210</b>
Spring Forces	211
Spring Yield	213
Connection Beam Yield	214
<b>Tendon Results</b>	<b>215</b>
<b>Eigenvalue Results</b>	<b>217</b>
Modal Frequencies	218
Mode Shapes	220
Modal Reactions and Modal Member/Plate Forces	222

---

## Joint Results

Joint Displacements	177
Joint Reactions	179
Joint Velocities & Accelerations	181

---

## 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 2000 User's Guide*].)

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 [p54])

*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 [p54])

*positive values* are about the corresponding axis using the right-hand rule

*units:* displacement-category units (see Results Units)

---

**For More Information**

- Joints [p54].

## 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 [p54])

*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 [p54])

*positive values* are about the corresponding axis using the right-hand rule

*units:* force-category units (see Results Units)

**For More Information**

- Joints [p54].

## 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 [p54])

*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 [p54])

*positive values* are about the corresponding axis using the right-hand rule

*units:* displacement-category units (see Results Units)

---

### For More Information

## **LARSA 2000 Reference**

---

---

- Joints [p54].

## Member Results

<b>Member End Forces</b>	<b>184</b>
<b>Member Sectional Forces</b>	<b>187</b>
<b>Member Stresses</b>	<b>190</b>
<b>Member Displacements</b>	<b>192</b>
<b>Member Plastic Deformation</b>	<b>194</b>
<b>Member Yield and Strains</b>	<b>196</b>
<b>Span Displacements and Forces</b>	<b>198</b>
<b>Analyzed Member Loads</b>	<b>200</b>

---

## 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 [p57])

*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 [p57])

*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 [p57])

*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 [p57])

*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 [p57])

*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 [p57])

*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**

- Joints [p54].
- Members [p57].

## 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 2000 User's Guide*]. 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 2000 User's Guide*] (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 [p57])

*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 [p57])

*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 [p57])

*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 [p57])

*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 [p57])

*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**

## LARSA 2000 Reference

---

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 [p57])

*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**

- Joints [p54].
- Members [p57].

## Member Stresses

Member stresses are the internal stresses along the length of the member, in member local directions.

Stresses 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 2000 User's Guide*]. Because numerical integration is an approximation, results toward the ends of members may have a small degree of error.

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.

### 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 2000 User's Guide*] (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.

*positive values* are tensile

*specified w.r.t.* the local coordinate system of the member (see Members [p57])

*units:* stress-category units (see Results Units)

### **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 [p32]. Normal stress is the sum of extensional stresses caused by the axial force and bending moments at the station.

*positive values* are always tensile

*specified w.r.t.* the local x-axis of the member (see Members [p57]) at the member's section's stress recovery points (see Sections [p32])

*units:* stress-category units (see Results Units)

### **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 [p57]) at the member's section's stress recovery points (see Sections [p32])

*units:* stress-category units (see Results Units)

### **Shear Stress (Y/Z)**

The shear stresses are  $F_y/A_y$  and  $F_z/A_z$ . Shear stresses can only be reported if sectional shear areas have been specified (see Sections [p32]). Shear stresses caused by torsion are not included.

*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 [p57])

*units:* stress-category units (see Results Units)

---

## **For More Information**

- Members [p57].

## 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 2000 User's Guide*]. 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 2000 User's Guide*] (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 [p57])

*units:* displacement-category units (see Results Units)

#### Rotation (X/Y/Z)

## LARSA 2000 Reference

---

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 [p57])  
*units:* displacement-category units (see Results Units)

---

### **For More Information**

- Joints [p54].
- Members [p57].

## Member Plastic Deformation

Member plastic deformations are the components of the deformation of a members 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 [p57])

*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 [p57])

*units:* displacement-category units (see Results Units)

---

**For More Information**

- Members [p57].
- Materials [p29].

## 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 [p214] and Spring Yield [p213], 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 [p57])  
*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 [p192] 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 2000 User's Guide*].
- 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 [p187] 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 2000 User's Guide*].

---

### For More Information

## **LARSA 2000 Reference**

---

- Members [p57].
- Spans [p62].
- Member Displacements [p192].
- Member Sectional Forces [p187].

## Analyzed Member Loads

LARSA reports back what member loads were included in the analysis for verification purposes. The reported loads include input loads [p82] entered by the user, computed self-weight [p89], moving load patterns [p100] for each generated moving load case, and tendon [p74] equivalent forces.

For an explanation of member loads, see Member Loads [p93] and Member Thermal Loads [p96].

### 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**

- Member Loads [p93].
- Moving Load Analysis [p150].

## Plate Results

Plate Forces on Center	203
Plate Forces at Joints	205
Plate Stresses	207

---

## 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 always tensile

*specified w.r.t.* the local coordinate system of the plate (see Plates [p63])

*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 always tensile

*specified w.r.t.* the local coordinate system of the plate (see Plates [p63])

*units:* force-category units (see Results Units)

#### In-Plane Shear Fxy

The in-plane shear force, caused by the membrane behavior of the element.  $F_{xy}$ , the in-plane shear force on the face normal to the x-axis and in the direction of y, is equal to  $F_{yx}$ , 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 [p63])  
*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 [p63])  
*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 [p63])  
*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 [p63])  
*units:* force-category units (see Results Units)

---

## **For More Information**

- Plates [p63].

## Plate Forces at Joints

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.

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 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**

- Plates [p63].

## Plate Stresses

Plate stresses are the internal stresses of plates.

### 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.

#### 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 [p63])

*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 [p63])

*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 [p63])

*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:

$$\begin{aligned} T_{\max} &= (\text{Principal}_{\max} - \text{Principal}_{\min}) / 2 \\ T_{\min} &= -(\text{Principal}_{\max} - \text{Principal}_{\min}) / 2 \end{aligned}$$

*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 [p63])

*units:* stress-category units (see Results Units)

---

## For More Information

## **LARSA 2000 Reference**

---

---

- Plates [p63].

## Spring Results

Spring Forces	211
Spring Yield	213
Connection Beam Yield	214

---

## Spring Forces

Spring forces report the forces and moments acting on 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.

## 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.

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 has a displacement coordinate system assigned, in which case the displacement system directions are used.

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.

### Force $F_x/F_y/F_z$

The translational forces in the x-, y-, and z-directions.

*positive values* are in the direction of the positive axis

*units*: force-category units (see Results Units)

### Moment $M_x/M_y/M_z$

The moments about the x-, y-, and z-axes.

*positive values* are about the axis in the direction of the right-hand rule

*units*: force-category units (see Results Units)

---

## For More Information

- Springs [p66].

## **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.

**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.

---

**For More Information**

- Tendons [p74].
- Tendon Results Tools [in *LARSA 2000 User's Guide*].

## Eigenvalue Results

<b>Modal Frequencies</b>	<b>218</b>
<b>Mode Shapes</b>	<b>220</b>
<b>Modal Reactions and Modal Member/Plate Forces</b>	<b>222</b>

---

## 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 [p138]) 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 [p132].

Mode Shapes [p220] 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

## **LARSA 2000 Reference**

---

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 [p218].

### 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.

*positive values* are in the positive directions of the global coordinate system

*specified w.r.t.* global coordinates

*units:* displacement-category units (see Results Units)

#### Rotation (X/Y/Z)

The rotation of the joint about the x-, y-, and z-axes.

*positive values* are about the global axes using the right-hand rule

*specified w.r.t.* global coordinates

*units:* displacement-category units (see Results Units)

---

### For More Information

## **LARSA 2000 Reference**

---

- Modal Frequencies [p218].

## **Modal Reactions and Modal Member/Plate Forces**

Modal internal forces and reactions are computed for each mode [p218] using the mode shapes [p220] as the displacements of the structure.

For more information on these modal results, see the corresponding results for a static/dynamic analysis:

- Joint Reactions [p179]
- Member End Forces [p184]
- Plate Forces on Center [p203]

# Index

- acceleration, 181
- analytical members, 62
- analyzed loads, 200
- anchor set, 74
- axial force
  - Member End Forces, 184
  - Member Sectional Forces, 187
  - Spring Forces, 211
- backbone, 36
- beams
  - Members, 57
  - The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements, 10
- bearings
  - Isolator/Bearing Property Definitions, 40
  - Isolators and Bearings, 69
  - Isolators and Bearings: The Seismic Elements, 22
- beta angle, 57
- buckling, 120
- cables
  - Members, 57
  - The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements, 10
- cement hardening, 29
- chains of members, 62
- characteristic vectors, 220
- coefficient of thermal expansion, 29
- collapse, 120
- combining results, 86
- compression
  - Member End Forces, 184
  - Member Sectional Forces, 187
- compressive strength, 29
- concrete creep and shrinkage, 44
- constraints
  - Joints, 54
  - Slave/Master Constraints, 72
- construction steps, 172
- coordinate systems, 49
- coordinates, 54
- creep
  - Sections, 32
  - Time-Dependent Material Property Definitions, 44
- cross-sectional properties, 32
- curvature, 196
- cylindrical coordinate systems, 49
- deconstruction/demolition, 172
- deformation, 194
- displacement loads, 92
- displacements
  - Joint Displacements, 177
  - Member Displacements, 192
  - Span Displacements and Forces, 198
- distributed loads
  - Member Loads, 93
  - Plate Loads, 98
- ductility, 32
- dynamic analysis
  - Linear Time History Analysis, 141
  - Nonlinear Time History Analysis, 143
  - Response Spectra Analysis, 132
- earthquake elements, 22
- eigenvalue analysis
  - Eigenvalue Analysis Options, 138
  - Eigenvalue and Stressed Eigenvalue Analysis, 130
  - Response Spectra Analysis, 132
  - Stressed Eigenvalue Analysis Options, 139
- eigenvectors, 220
- elastic modulus, 29
- end offsets and releases, 57
- excitations, 102
- Fc28, 29
- follower forces, 114
- forces
  - Joint Reactions, 179
  - Member End Forces, 184
  - Member Sectional Forces, 187
  - Plate Forces at Joints, 205
  - Plate Forces on Center, 203
  - Spring Forces, 211
  - Tendon Results, 215
- frequencies, 218
- generated loads, 200
- groups of members, 62
- guaranteed ultimate tensile strength, 29
- HSXX vehicles, 167
- hysteretic members
  - Member Yield and Strains, 196
  - Members, 57
  - The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements, 10
- hysteretic springs
  - Spring Property Definitions, 36
  - Springs, 66
- incremental loading, 120
- inelastic material properties, 29
- inelastic members
  - Member Yield and Strains, 196
  - The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements, 10
- influence lines and surfaces
  - Influence Lines, 162

- Influence Surfaces, 165
- Influence-Based Analysis, 152
- initial conditions, 102
- in-plane shear, 203
- IRC vehicles, 167
- isolators
  - Isolator/Bearing Property Definitions, 40
  - Isolators and Bearings, 69
  - Isolators and Bearings: The Seismic Elements, 22
- isotropic material properties, 29
- joint displacement
  - Joint Displacements, 177
  - Joint Velocities & Accelerations, 181
- joint displacement loads, 92
- joint loads
  - Joint Loads, 91
  - Support Displacements, 92
- joint reactions, 179
- joints, 54
- lanes
  - Lanes, 76
  - Tendon and Lane Paths, 77
- large displacements, 114
- lateral offset, 162
- linear static analysis, 108
- live load analysis
  - Influence-Based Analysis, 152
  - Moving Load Analysis, 150
- live loads, 100
- load combinations, 86
- loads, 82
- local coordinate system
  - Members, 57
  - Plates, 63
- locking translation, 72
- mass elements, 70
- master joints, 72
- materials, 29
- maximum absolute stress, 190
- member deformation/displacements
  - Member Displacements, 192
  - Member Plastic Deformation, 194
- member displacements, 198
- member forces
  - Member End Forces, 184
  - Member Sectional Forces, 187
  - Span Displacements and Forces, 198
- member loads
  - Analyzed Member Loads, 200
  - Member Loads, 93
  - Member Thermal Loads, 96
- member sectional stresses, 190
- member stresses, 190
- member thermal loads, 96
- member yield and strains, 196
- members
  - Members, 57
  - Spans, 62
  - The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements, 10
- membrane force, 203
- modal frequencies, 218
- mode shapes
  - Eigenvalue and Stressed Eigenvalue Analysis, 130
  - Mode Shapes, 220
  - Response Spectra Analysis, 132
- modulus of elasticity, 29
- moment of inertia, 32
- moments
  - Joint Reactions, 179
  - Member End Forces, 184
  - Member Sectional Forces, 187
  - Plate Forces at Joints, 205
  - Plate Forces on Center, 203
  - Spring Forces, 211
- moving load analysis
  - Influence-Based Analysis, 152
  - Lanes, 76
  - Moving Load Analysis, 150
  - Moving Load Analysis Options, 171
- moving loads, 100
- natural frequencies
  - Eigenvalue and Stressed Eigenvalue Analysis, 130
  - Response Spectra Analysis, 132
- Newmark-Beta, 143
- Newton-Raphson, 143
- nodes, 54
- nonlinear buckling analysis, 120
- nonlinear pushover analysis, 122
- nonlinear restart analysis, 125
- nonlinear springs
  - Spring Property Definitions, 36
  - Springs, 66
- nonlinear static analysis
  - Nonlinear Static Analysis, 114
  - Nonlinear Static Analysis Options, 123
  - P-Delta Analysis, 111
  - Staged Construction Analysis, 172
- nonlinear time history analysis
  - Nonlinear Time History Analysis, 143
  - Nonlinear Time-History Analysis Options, 147
- normal modes, 220
- normal stress, 190
- orientation angle, 57
- orthotropic material properties, 29
- P/A, 190
- partial loads, 93
- participating mass, 218
- pastic section modulus, 32
- path, 77
- p-delta analysis, 111
- perimeter, 32

- physical members, 62
- pin connection, 57
- plastic deformation, 194
- plate forces
  - Plate Forces at Joints, 205
  - Plate Forces on Center, 203
- plate loads, 98
- plate stresses, 207
- plates
  - Plates, 63
  - The Plate Element: Plates and Shells, 17
- point loads
  - Joint Loads, 91
  - Member Loads, 93
  - Plate Loads, 98
- Poisson ratio, 29
- post-tensioning
  - Tendon Results, 215
  - Tendons, 74
  - The Tendon Pseudoelement, 25
- prestress force, 57
- pre-tensioned tendons, 74
- primary moments, 215
- principal stress, 207
- pushover analysis
  - Nonlinear Pushover Analysis, 122
  - Nonlinear Pushover Analysis Options, 127
- reactions, 179
- rehabilitation, 172
- relaxation, 44
- releases, 57
- residual strength, 32
- response spectra analysis, 132
- restraints
  - Joints, 54
  - Slave/Master Constraints, 72
- rigid zones, 57
- rotation
  - Joint Displacements, 177
  - Joint Velocities & Accelerations, 181
  - Member Displacements, 192
  - Mode Shapes, 220
  - Span Displacements and Forces, 198
- section properties and dimensions, 32
- seismic analysis, 132
- seismic elements, 22
- semi-rigid end connections, 57
- shear area, 32
- shear force
  - Member End Forces, 184
  - Member Sectional Forces, 187
  - Spring Forces, 211
- shear forces, 203
- shear modulus, 29
- shear stress, 190
- shells
  - Plates, 63
  - The Plate Element: Plates and Shells, 17
- shock loading, 132
- shrinkage
  - Sections, 32
  - Time-Dependent Material Property Definitions, 44
- slave/masters, 72
- solution convergence, 123
- span displacements, 198
- span forces, 198
- spans, 62
- spherical coordinate systems, 49
- spring forces, 211
- spring properties, 36
- springs
  - Spring Property Definitions, 36
  - Springs, 66
  - The Spring Element: A Multipurpose Element, 19
- staged construction analysis, 172
- static analysis
  - Linear Static Analysis, 108
  - Nonlinear Static Analysis, 114
  - P-Delta Analysis, 111
- steel relaxation, 44
- stiffness degradation, 36
- strain, 196
- strand area, 74
- stress
  - Member Stresses, 190
  - Plate Stresses, 207
- stress direction, 207
- stress recovery points, 32
- stress stiffening, 114
- stress trajectory, 207
- stressed eigenvalue analysis, 130
- support settlements, 92
- tendon GUTS, 29
- tendon relaxation, 29
- tendon results, 215
- tendons
  - Tendon and Lane Paths, 77
  - Tendon Results, 215
  - Tendons, 74
  - The Tendon Pseudoelement, 25
- tension
  - Member End Forces, 184
  - Member Sectional Forces, 187
- thermal expansion, 29
- thermal loads
  - Member Thermal Loads, 96
  - Plate Loads, 98
- time history analysis
  - Linear Time History Analysis, 141
  - Nonlinear Time History Analysis, 143
  - Nonlinear Time-History Analysis Options, 147
  - Time History Analysis Options, 145

## LARSA 2000 Reference

---

- time history loads, 102
- time-dependent material properties
  - Staged Construction Analysis, 172
  - Time-Dependent Material Property Definitions, 44
- torsion
  - Plate Forces on Center, 203
  - Spring Forces, 211
- torsional constant, 32
- translation
  - Joint Displacements, 177
  - Joint Velocities & Accelerations, 181
  - Member Displacements, 192
  - Mode Shapes, 220
  - Span Displacements and Forces, 198
- transverse offset, 162
- trapezoidal loads, 93
- trusses
  - Members, 57
  - The Member Element: Beams, Trusses, Cables, and other Beam-Like Elements, 10
- ultimate stress, 29
- uniform loads, 93
- unit weight, 29
- user coordinate systems, 49
- vehicles
  - Lanes, 76
  - Standard Vehicles, 167
- vehicular loads
  - Moving Load Analysis, 150
  - Moving Loads, 100
- velocity, 181
- Von Mises stress, 207
- wobble, 74
- yield ratio, 196
- yield stress, 29
- Young's modulus, 29