LARSA 2000 User's Guide

for

LARSA 2000
Finite Element Analysis and Design Software

Larsa, Inc.
Melville, New York, USA

Revised August 2004

Table of Contents

Overview

This is the LARSA 2000 User's Guide, which covers how to use the LARSA 2000/4th Dimension user interface. This is also the starting point for any new LARSA 2000 user.

About the Manual

Topics covered in this manual include:

- Files and Reports, including importing/exporting
- Graphics, Selection, Spreadsheets, and Explorers
- Modeling Tools, including drawing, transformations, generation, and grids
- Running an Analysis
- Getting Results, both visually and numerically, *and*
- Steel Design

Other manuals for LARSA 2000/4D cover more specialized or in-depth topics:

Samples and Tutorials
This manual includes the Quickstart Guide, which all new LARSA users should go through to get a basic understanding of how LARSA 2000 works. The manual also includes a number of samples and tutorials.

Reference Manual
The reference manual covers LARSA's element library and other input data, explains how analyses are performed, and defines how analysis output is reported.

Bridge Analysis Guide
An in-depth guide to using LARSA 2000/4D for bridge structures, including using tendons, influence-based results, and related tools. The manual also includes a tutorial for conforming to the AASHTO LRFD 1998 code.

Staged Construction Analysis Guide

An in-depth guide to using LARSA 2000/4D in staged or segmental construction, including a reference for LARSA's time-dependent staged construction analysis.

Developer's Guide
A reference for extending LARSA 2000 through user-written plugins and VBA macros.

Section Composer Reference
A guide to the LARSA Section Composer.

These manuals are available in several formats on our website at http://www.larsausa.com in the Support section.

Getting Help

For technical support,

- Please call us using one of the following telephone numbers:
- U.S. & Canada, Toll Free: 1-800-LARSA-01
- International: 212-736-4326 (our New York office)
- Email us at support@larsausa.com, or
- Visit our website www.larsausa.com and see the Support section.

We are open 9am to 5pm U.S. Eastern Time Monday through Friday.

You may also reach us by postal mail at:

About LARSA 2000 and LARSA 2000/4D

LARSA 2000 and LARSA 2000/4th Dimension are the most advanced multipurpose structural analysis package available today. The sixth version in the LARSA product line, LARSA 2000 features an all-new, powerful user interface and an engine with unmatched analytical features.

LARSA 2000/4D's new fourth-dimension features make the software uniquely able to handle the demands of modeling bridges: influence-line based moving load analysis, staged construction analysis, time dependent construction analysis, hysteretic and seismic elements, and progressive collapse.

And LARSA 2000's new user interface is just as astounding as its analytical engine. The interface features clear and powerful graphics windows, easy-to-use numerical spreadsheets for both input data and results, new *Explorers* for navigating project data, and a long list of modeling tools.

LARSA 2000 and LARSA 2000/4D are available as a program for Microsoft Windows 98/ME/2000/XP on Intel 386-compatible processors.

The LARSA structural analysis engine has been in commercial use for over 25 years. It was originally developed to perform nonlinear static analysis of structures that have large displacements, such as suspension and cable stayed bridges and guyed towers. The engine became popular for analyses of these types of structures because of its unprecedented accuracy at a reasonable price. The engine has been powerful since day one, using both tangent stiffness and the full Newton-Raphson method with iterations. LARSA has come a long way since it was first available on the VAX super-mini computers decades ago.

In 1986, LARSA expanded its engine to meet the needs of any structural analysis that a practicing engineer might find at that time, including seismic analysis and design. At the same time, a graphical user interface was released for DOS. LARSA was the first to offer an individual PC-based DOS structural analysis package with geometric nonlinear analysis capabilities.

LARSA again led the industry in 1994 by taking the next step to Microsoft Windows. That

year LARSA introduced its 16-bit Windows 3.x version of the software with a point and click graphical user interface. In 1996, LARSA was the first to offer elastic/perfectly plastic pushover analysis.

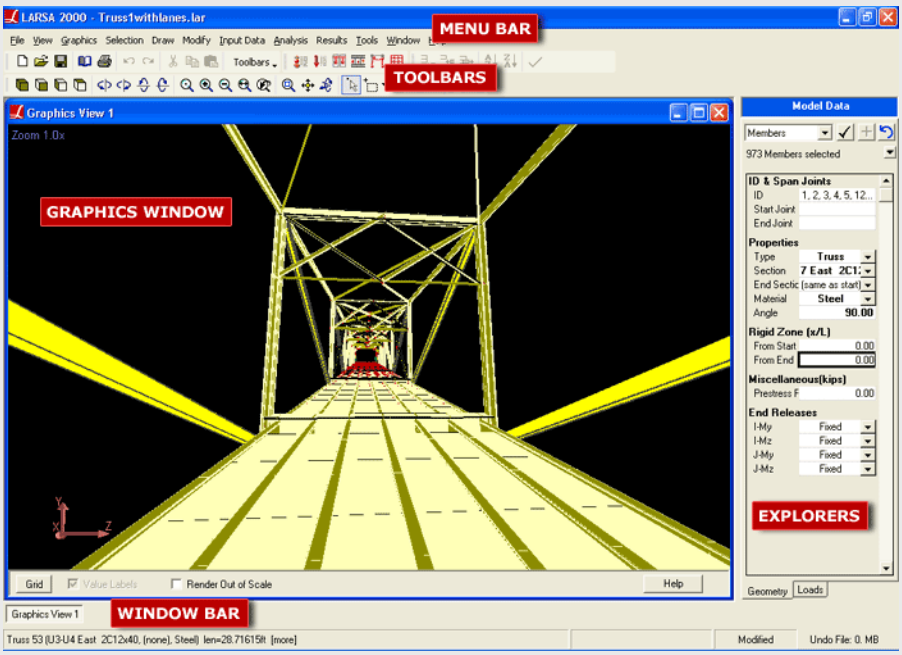
In 2001, LARSA 2000 and LARSA 2000/4th Dimension were released. LARSA continues to lead the industry in structural analysis.

Using LARSA Overview

This section provides an overview of the features needed to get started using LARSA 2000.

The LARSA 2000 Look and Feel

The LARSA 2000 screen looks like the window in the figure below.



The Layout of LARSA 2000

Menu Bar

Contains all commands that you can use.

Toolbars

Contains buttons for frequently used commands.

Graphics Window

Shows a graphical representation of the open project. See An Overview of Graphics & Selection [p38].

Explorers

Provides quick access to all aspects of the open project. See Explorers [p54].

Window Bar

Lists all open LARSA windows (e.g. graphics windows, spreadsheets). Click the name of a window to switch to it.

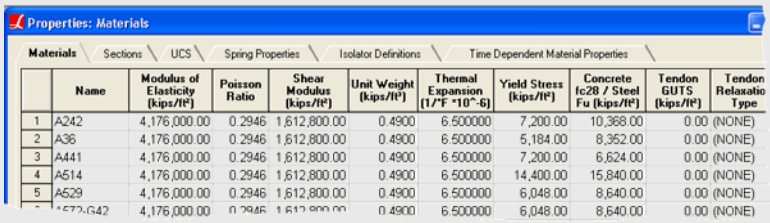
Defining Properties

The term "properties" in LARSA 2000 refers to the project input that defines the behavior of the geometry of the model, including material properties, sectional properties, nonlinear spring properties, and time-dependent material properties.

Materials and Sections can be added to the project using any of the following methods.

Materials and sections can be added using the materials and sections spreadsheets. All material and sections properties can be viewed/edited directly on the spreadsheet. For more information, please refer to Using the Model Spreadsheets [p91].

This tool can be found on the Input Data menu under "Properties."



Properties: Materials

Materials | Sections | UCS | Spring Properties | Isolator Definitions | Time Dependent Material Properties

	Name	Modulus of Elasticity (kips/ft²)	Poisson Ratio	Shear Modulus (kips/ft²)	Unit Weight (kips/ft³)	Thermal Expansion (1/°F *10^-6)	Yield Stress (kips/ft²)	Concrete fc28 / Steel Fu (kips/ft²)	Tendon GUTS (kips/ft²)	Tendon Relaxatio Type
1	A242	4,176,000.00	0.2946	1,612,800.00	0.4900	6.500000	7,200.00	10,368.00	0.00	(NONE)
2	A36	4,176,000.00	0.2946	1,612,800.00	0.4900	6.500000	5,184.00	8,352.00	0.00	(NONE)
3	A441	4,176,000.00	0.2946	1,612,800.00	0.4900	6.500000	7,200.00	6,624.00	0.00	(NONE)
4	A514	4,176,000.00	0.2946	1,612,800.00	0.4900	6.500000	14,400.00	15,840.00	0.00	(NONE)
5	A529	4,176,000.00	0.2946	1,612,800.00	0.4900	6.500000	6,048.00	8,640.00	0.00	(NONE)
	A572-G42	4,176,000.00	0.2946	1,612,800.00	0.4900	6.500000	6,048.00	8,640.00	0.00	(NONE)

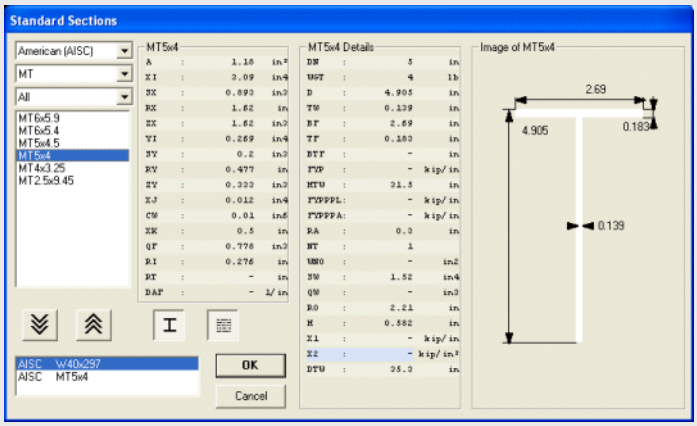
Materials Spreadsheet

Standard materials can be loaded using the *Standard Materials* dialog box. For more information, please refer to Loading Standard Materials [p103].

This tool can be found on the Input Data menu under "Standard Materials."

The *Standard Sections* dialog box can be used to import sections from a standard section database. For more information, please refer to Loading Standard Sections [p105].

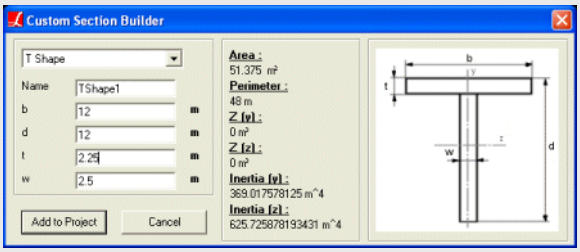
This tool can be found on the Input Data menu under "Standard Section."



Importing a Standard Section

The *Custom Section Builder* can be used to add sections by specifying the shape type and the dimensions. For more information, please refer to Creating Custom Sections [p107].

This tool can be found on the Input Data menu under "Custom Section."



Creating a Custom Section

Parametric sections can be added by attaching a user-created parametric section database to the project. Parametric section databases can be created using the *Section Composer*. For more information, please refer to Section Composer User's Guide [in *LARSA Section Composer*].

Spring Properties, Isolator Definitions, Time-Dependent Material Properties are out of the scope of this document. For more information, please refer to Properties [in *LARSA 2000 Reference*].

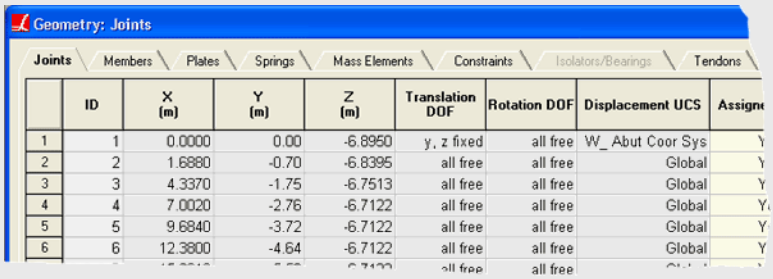
Creating Geometry

The term "geometry" in LARSA 2000 refers to the joints, members, plates, slave/masters and other physical elements that make up a structure.

A structure can be created using the four methods listed below.

The spreadsheets can be used to add new geometry elements. Properties, such as materials and sections, can be assigned to the structure using the spreadsheets if they have already been loaded into the project. (See above.) For more information, please refer to Using the Model Spreadsheets [p91].

This tool can be found on the Input Data menu under "Geometry."

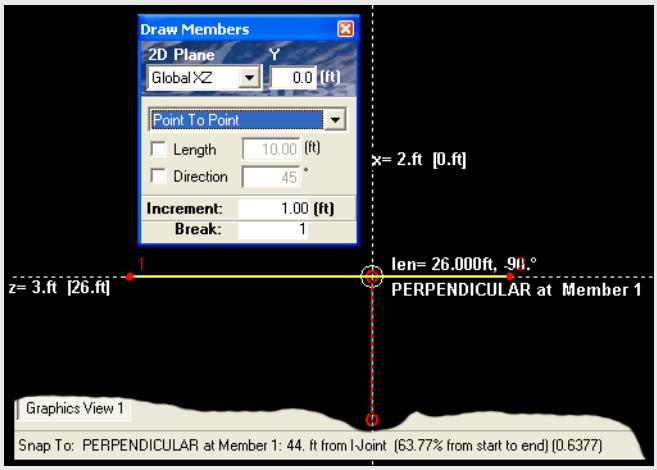


	ID	X (m)	Y (m)	Z (m)	Translation DOF	Rotation DOF	Displacement UCS	Assigne
1	1	0.0000	0.00	-6.8950	y, z fixed	all free	W_ Abut Coor Sys	Y
2	2	1.6880	-0.70	-6.8395	all free	all free	Global	Y
3	3	4.3370	-1.75	-6.7513	all free	all free	Global	Y
4	4	7.0020	-2.76	-6.7122	all free	all free	Global	Y
5	5	9.6840	-3.72	-6.7122	all free	all free	Global	Y
6	6	12.3800	-4.64	-6.7122	all free	all free	Global	Y

Input Spreadsheets

Structure geometry can also be drawn on the graphics view windows. For more information, please refer to Drawing Geometry and Loads [p70].

This tool can be found on the Draw menu under "Geometry."



Drawing Members

Structure geometry can also be created using the *Model Data Explorer*. Properties can be assigned while creating the geometry. The *Model Data Explorer* can also be used to edit the structure geometry data. For more information, please refer to Model Data Explorer [p56].

This tool can be found on the View menu under "Model Data."

Various generation tools can be used to create the geometry. For more information, please refer to Generation Tools [p81].

This tool can be found on the Draw menu under "Generation."

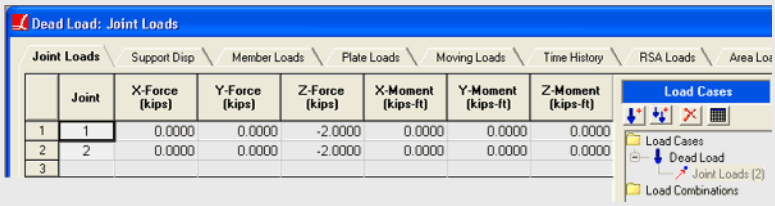
Applying Loads

Load Cases: All loads must be contained in a "load case." Therefore, before applying loads to the structure geometry, one needs to create the load cases. For help creating and editing load cases, see Load Cases Explorer [p58].

Loads: Once there are load cases, loads can be created using one of the methods listed below. For more information on loads, please refer to Loads [in *LARSA 2000 Reference*].

Spreadsheets can be used to create loads. The loads spreadsheet can be accessed by using one of the two methods listed below.

1. Open the load cases input spreadsheet from the **Input Data** menu. Right click on the load case row that you want to see the contained loads and click **Edit Loads**.
2. Double click on the load case in the *Load Case Explorer*.



Loads Spreadsheets

Loads can be created graphically using the graphics view. For more information, please refer to Drawing Geometry and Loads [p70]. Please note that not all load types can be drawn graphically, including time-history excitations and initial conditions. For those load types, use the spreadsheets.

This tool can be found on the Draw menu under "Loads."

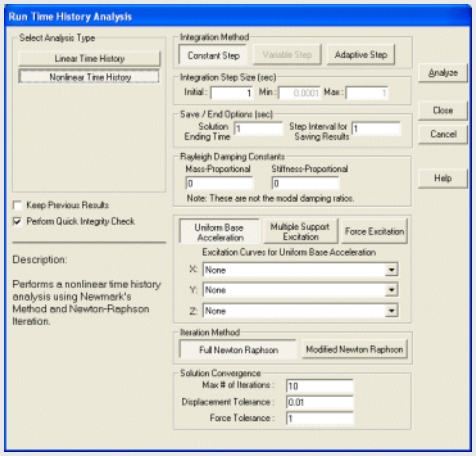
Analyzing the Model

Before analyzing your model, please use *Integrity Check* to ensure the validity of the project. For more information, please refer to Integrity Check [p100].

This tool can be found on the Input Data menu under "Integrity Check."

Run an analysis by selecting the proper analysis type under **Analysis** menu. For more information, please refer to Running an Analysis [p116].

This tool can be found on the Analysis menu under "[Choose an Analysis Type]."



Analysis Window

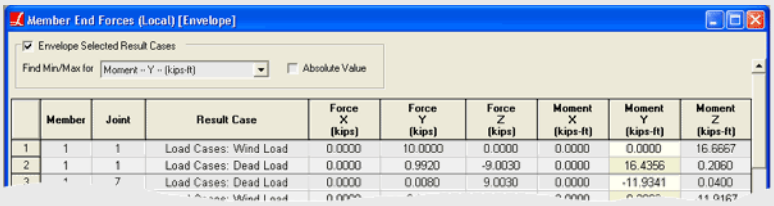
Viewing the Results

Once the model has been analyzed, all of the result cases will be available in the *Analysis Results Explorer*. For more information, please refer to Analysis Results Explorer [p66].

This tool can be found on the View menu under "Analysis Results."

Result data can be viewed in spreadsheet format using the results spreadsheets. First select the result case from the *Analysis Result Explorer* and select the appropriate results spreadsheet under the **Results** > **Spreadsheets** menu. For more information, please refer to Results Spreadsheets [p127].

This tool can be found on the Results menu under "Spreadsheets."

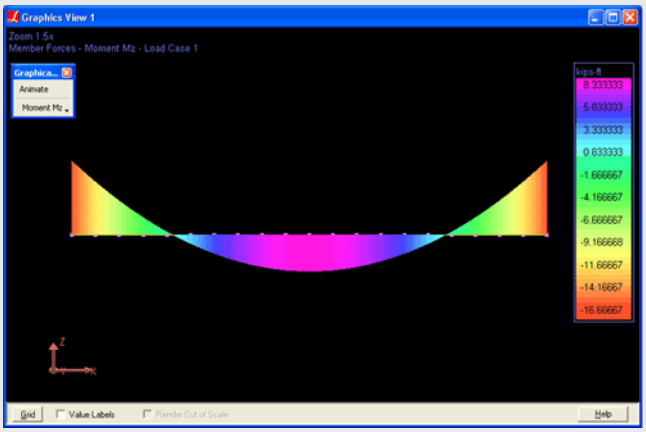


	Member	Joint	Result Case	Force X (kips)	Force Y (kips)	Force Z (kips)	Moment X (kips-ft)	Moment Y (kips-ft)	Moment Z (kips-ft)
1	1	1	Load Cases: Wind Load	0.0000	10.0000	0.0000	0.0000	0.0000	16.6667
2	1	1	Load Cases: Dead Load	0.0000	0.9920	-9.0030	0.0000	16.4356	0.2060
3	1	7	Load Cases: Dead Load	0.0000	0.0080	9.0030	0.0000	-11.9341	0.0400

Results Spreadsheet View

Result data can also be viewed graphically using the graphics view window. For more information, please refer to Viewing Results Graphically [p120].

This tool can be found on the Results menu under "Graphical."



Graphical Results

Results can also be graphed. For more information, please refer to Graphing Results [p136].

This tool can be found on the Results menu under "Graphs."

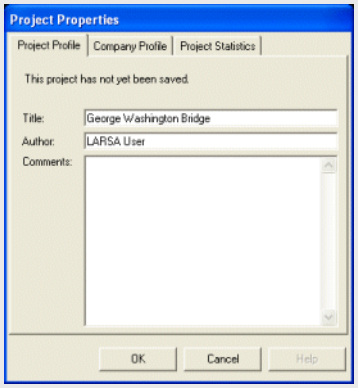
Files & Reports

The following sections describe using files and reports in LARSA, through the loading, saving, import/export and report/printing features.

Project Properties

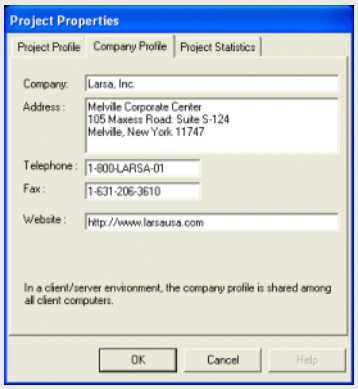
The Project Properties dialog box is used to edit the title, author, and comments fields of the open project, to edit the user's company information, and to display project statistics.

This tool can be found on the File menu under "Properties."



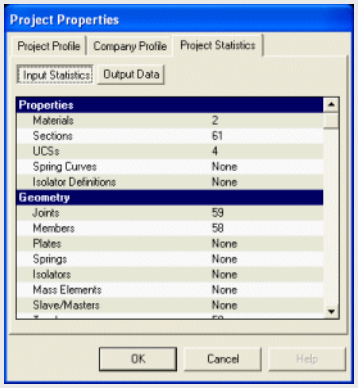
The first page of the Project Properties dialog box has the title, author, and comments fields.

The first page of the dialog box has three edit fields where the title, author, and comments fields of the open project can be set. This information is used for generating report and other exports.



The second page of the Project Properties dialog box has the company information fields.

The second page of the dialog box has several fields to edit the company information associated with the license of LARSA running on the computer. The company profile is used for generating reports and other exports. Because each installation of LARSA is expected to be used in one company, the company profile is stored as a global setting of LARSA and not in individual projects.



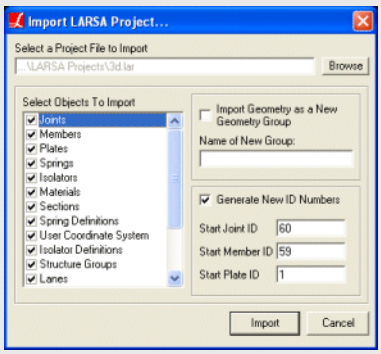
The third page of the Project Properties dialog box shows project statistics.

The third page of the dialog box shows statistics for the currently open project. For *Input Statistics*, the total number of each type of properties, geometry elements, and loads are listed. For *Output Statistics*, each result type is listed along with whether that result has been computed by an analysis and is available for viewing.

Import Project (Merge)

The Import Project tool is used to merge the contents of a second LARSA project into the open project. This is useful if different parts of a structure are stored in separate files and need to be merged into one project.

This tool can be found on the File menu under "Import > LARSA Project."



Import Project

To merge the contents of a project into the currently open project:

- Click the Browse button to select the file with the data that is to be imported.
- Uncheck types of data that you do not want merged. If *Members* is unchecked, for instance, the members in the selected file will not be imported.
- To add all of the imported geometry into a new structure group [p60], check the *Import geometry as new structure group* option and then enter the name of the new group.
- If items in the imported project have the same ID numbers as items in the current project, the IDs of the imported items must be changed. LARSA will automatically renumber conflicting items by default.
- To override the default renumbering, check *Generate New ID Numbers* and specify

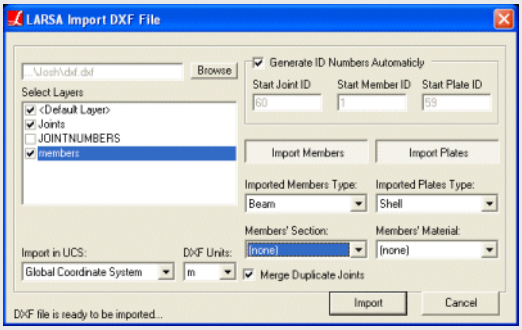
the starting ID number for the joints, members, and plates imported.

- Click *Import* to begin importing the contents of the selected file.

Import from DXF (AutoCAD)

The Import DXF tool is used to import the contents of a DXF file into the open project. DXF files are usually generated by drafting tools such as AutoCAD.

This tool can be found on the File menu under "Import > DXF File."



Import DXF File

To import the contents of a DXF file into the currently open project, first click the *Browse* button and select the file to import. Then choose the appropriate options, described below.

How the Import is Performed

LARSA 2000 imports following DXF objects from the selected layers.

- **LINE** and **LWPOLYLINE** objects are imported as members.
- **3DFACE** objects are imported as plates.

Import Options

Select Layers

Often the contents of DXF files are organized into layers, each layer containing a different set of drawing elements. LARSA will only import the contents of layers marked with a check in the selection list.

Import in UCS

The contents of the DXF file can be imported relative to a user coordinate system [see "User Coordinate Systems" in *LARSA 2000 Reference*]. If a coordinate system other than the global coordinate system is chosen, then the coordinates of the DXF entities are transformed as if they were specified in coordinates relative to the selected system.

DXF Units

Coordinates in DXF files are specified without units. By default, the coordinates in the DXF file are assumed to be in whatever coordinate units are being used in the open project. To override what units the coordinates in the DXF file are in, modify this option.

Generate ID Numbers Automatically

Items imported from a DXF file must be given IDs not already used by objects in the open project. LARSA will automatically number items by default. To override the starting ID numbers of the imported joints, members, and plates, uncheck this option and specify the starting IDs.

Import Members, Import Plates

Unpush these buttons if you do not want the members or plates imported from the DXF file.

Imported Types, Sections, Materials

Choose what member/plate types, sections, and materials to assign to the imported entities, if any.

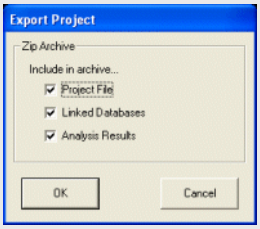
Merge Duplicate Joints

If checked, an extra scan of the imported coordinates will be performed to ensure no imported joints result in having two different joints at the same location in the project. If the DXF file takes a long time to import and if you are sure no coordinates in the DXF file overlap with coordinates of joints in the open project, this option may be unchecked.

Export to Zip Package

The Export to Zip tool compresses all of the files needed by the open project into a Zip archive which can be more easily and quickly sent by email than uncompressed project and output files.

This tool can be found on the File menu under "Export > ZIP Package."



Export to ZIP Package

Zip archives store multiple files in compressed form. Because analyzed LARSA projects involve many files on disk (the project file, analysis results, and linked databases), it is useful to compress a project into a single Zip archive file before sending the project by email or disk to others.

LARSA will automatically compress all of the files needed by a project into a Zip archive with the Export to Zip Package command. The user may select what parts of a project are to be exported into the Zip file.

Project File

Check this option to include the main project file (the LAR file) in the Zip archive.

Linked Databases

Check this option to include the databases linked to the project in the Zip archive. Linked databases generally are the moving load databases, response spectra databases, and time history databases used by the project. (See Connecting Databases [p89].)

Analysis Results

Check this option to include the analysis results in the Zip archive. Because analysis results for large projects often are very large on disk, leaving this option checked may result in the exported Zip file being too large to send by email or disk.

The contents of a Zip archive can be accessed in several ways.

- LARSA will automatically open a LARSA project in a Zip archive from the regular **File > Open** menu command.
- Users of Windows XP can extract the contents of a Zip archive from Windows Explorer.
- Other third-party tools, such as WinZip (from http://www.winzip.com), are available for creating and extracting Zip files.

Export to DXF (AutoCAD)

The Export to DXF File tool exports the geometry in the open project to a DXF file, which is readable by AutoCAD and other drafting software.

This tool can be found on the File menu under "Export > DXF File."

Export to DXF File

This tool has the following options.

Select DXF Version

Select what DXF format to use for export.

Export Members/Plates

Choose whether members and/or plates are exported to the DXF file.

Export Only Selected Objects

If this option is checked, only the selected geometry elements are exported to the DXF file.

How the Export is Performed

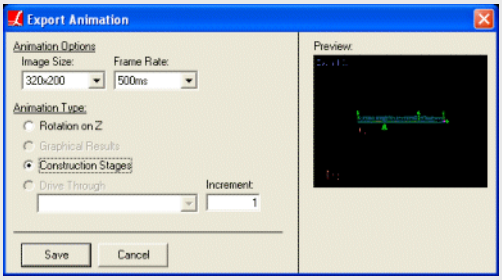
LARSA 2000 exports joints, joint numbers, members and plates to DXF format.

- **Joints** are exported as *POINT* objects. All joints are placed in a layer called "Joints." Joint numbers are exported as *MTEXT* and placed in a 'JointNumbers' layer.
- **Members** are exported as *LINE* objects. All members are placed in a "Members" layer.
- **Plates** are exported as *3DFACE* objects. All plates are placed in a "Plates" layer.

Export AVI (Animation)

The Export AVI Animation tool exports an animation of the open project to an AVI-format file, which is playable on all Microsoft Windows computers.

This tool can be found on the File menu under "Export > AVI Animation."



Export AVI Animation

This tool has the following options.

Image Size

The width and height in pixels of the animation. Larger image sizes will take longer to generate and will take up more disk space. 640x480 is generally a good size.

Frame Rate

The rate at which frames in the animation will be presented when the exported animation is played. The lower the frame rate, the smoother and faster the animation.

Animation Type

Choose what type of animation to export.

Rotation on Z: Rotates the structure about the z-axis through 360 degrees.

Graphical Results: This option is available if results are currently being displayed graphically. The results being displayed graphically will be animated for the export. If mode shapes are being displayed, an animation of the current mode shape will be exported. For other results, each result case in the current result group will be exported

as a frame in the animation. This is useful for staged construction, moving load and time history analysis results. See Viewing Results Graphically [p120].

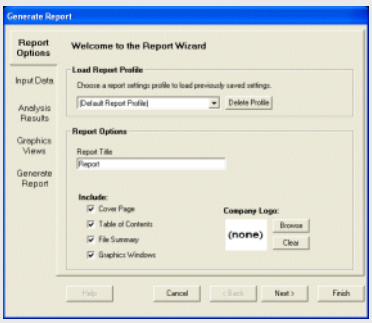
Construction Stages: Each step in the construction of the structure will be exported as a frame in the animation. See Staged Construction Guide [in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

Drive Through: An animation of a drive-through of the structure will be exported. The drive-through will follow a lane [see "Lanes" in *LARSA 2000 Reference*] already defined in the project, and the position of the vantage point will move at the specified increment (in coordinate units) per frame.

Print a Report

The Report tool prints out a hard copy of the model data or results in the open project, or saves a report to a file in various formats.

This tool can be found on the File menu under "Report."



Create a Printed Report

The Report Wizard takes you through the steps to print out a report of the model and/or results. Step forward and back through the wizard using the Back and Next buttons at the bottom of the dialog.

Report Options

Load Report Profile

At the very end of the Report Wizard, you have the option of saving the choices you made so that you do not need to go through the wizard again to generate the same report. To load in previously saved settings, choose the profile to load at this step.

Report Title

Give a title for the report. The title will be printed on the cover page.

What Pages to Include

You may optionally include each of the following pages:

- Cover Page - A cover page will be printed.
- Table of Contents - A table of contents of the report will be printed.
- File Summary - A summary of the project's contents will be printed.
- Graphics Windows - All open graphics windows will be printed.

Company Logo

To print your company's logo on your report, click the Browse button and select an image file on your computer with your company's logo. Standard image format files are supported, including BMP and GIF.

Model Data

The Model Data tab is divided into three sections: Properties & Geometry, Groups & Stages, and Load Cases & Combinations.

Each section contains a list of the different parts of your project that you can print. Sections are printed as spreadsheets, with the same rows and columns you normally see on screen.

Check each spreadsheet to include in the report.

Analysis Results

In this section, you can choose which analysis results to print in the report. Analysis results are only available if they have previously been solved in an analysis.

To print a result spreadsheet, follow these steps.

1. In the *Result* field, choose the type of result to print.
2. Select which result cases to print the results for. Select the result case on the left and then click *Add Result Case* to add it to the list of cases on the right.
3. If you want the results enveloped, select the aspect of the result to envelope on. For instance, if you are printing member end forces, you might want to report the maximum starting x-force for each member. In that case, choose Start Force in X to envelope on. The envelope will be taken over the result cases chosen in the previous

step.

4. When all of these choices are finalized, choose *Add Report Item*. Your choices will be listed in the box at the bottom.
5. Repeat this process for each result type to print.

Graphics Views

If you have saved Named Graphics Views, you may print a graphics page for each of the views. Select which named views to print from the list.

Generate Report

This is the final step before printing the report.

At this point, you may save the choices you made as a report profile so that you can recall the choices the next time you want to print the same report again. To save your choices, enter a name that describes your choices and then click *Save*.

Output Format

Choose which type of output format you would like.

- Printer - The report will be printed.
- PDF Format - The report will be saved to a PDF file, which is readable by the free Adobe Acrobat Reader provided by Adobe.
- HTML Document - The report will be saved in an HTML-format file, which is readable by any web browser.
- Microsoft Excel - The report will be saved as a Microsoft Excel spreadsheet. Because Excel spreadsheets are limited to 65,535 rows, large projects may not export successfully.
- Text-Only - The report will be saved as a plain text file, readable on any computer with any text editor.

Click *Finish* to print the report. You will first be presented with a preview of the report.

File Types

The various files used by LARSA 2000 are explained in this section.

Project Files (*.lar)

All of the information that makes up a model is stored in a *.lar file. This file includes all units, properties, geometry, loads etc.

Please note that moving load patterns, time history curves, and response spectra curves are stored in external databases. See below.

Database Files (*.drs,*.dth, *.dml, *.lpsx)

These files are database files that can be used by attaching them to a project.

- ***.drs:** Stores response spectra curve data. Use *Database Editor* to edit, create or view the database.
- ***.dth:** Stores time history curve data. Use *Database Editor* to edit, create or view the database.
- ***.dml:** Stores moving load pattern data. Use *Database Editor* to edit, create or view the database.
- ***.lpsx:** Stores parametric section data. Use *Section Composer* to edit, create or view the database.

For information on how to connect databases to a project, see Connecting Databases [p89].
For information on how to create and edit databases, see The Database Editor [p110].
For information on how to create and edit section databases, see Section Composer User's Guide [in *LARSA Section Composer*].

Analysis Results Files

The LARSA 2000 analysis engine writes results into result files with the same file name as the project file but with different extension. Each result file holds the information for a specific result type.

It is not uncommon for an analysis to generate dozens of results files.

Analysis Results Archive (*.lan)

If the option *Pack results into LAN archive* is checked in **Tools > Options**, then LARSA will automatically pack together all of the result files when LARSA closes a project or exists into a file with the same name as the project file but with the extension LAN. LARSA will unpack LAN files and then delete the LAN file when projects with LAN files are opened.

Graphics & Selection

The sections below describe the various ways of using LARSA's graphics windows and how to select and unselect objects.

An Overview of Graphics & Selection

The graphics views are the primary means of viewing and modeling structures in LARSA 2000. In order to use the graphics views effectively, one must understand the six graphics tools and the concept of selection.

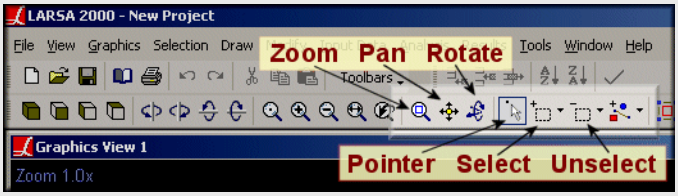
Selection

Every structural object in LARSA is either *selected* or *unselected*. LARSA provides various tools to modify the selection state of objects, including the Selection/Unselection graphics tools (below), Select Special [p42], Select by Plane [p44], and Select/Unselect All. Selected objects are generally displayed solid, while unselected objects are generally displayed dashed.

The selection state of objects is important because most modeling operations, such as modifying the properties of objects, applying transformations, and meshing, operate *only* on the selected objects. Therefore, the objects to which you want to apply an operation must be selected before applying the operation.

The Graphics Tools

There are six graphics "tools" that define the behavior of the mouse in the graphics windows. The tools are displayed together on the graphics toolbar, shown below. Exactly one tool is active at all times, and the active tool is shown highlighted. In the picture to the right, the Pointer Tool is active.



The Graphics Tools

The **Pointer** tool is the primary tool of the graphics windows. It is used to examine objects quickly (click) and to quickly manipulate objects (right-click). The **Select** and **Unselect** tools select and unselect objects. Click or click-and-drag windows around objects to select/unselect them.

Use the **Zoom** tool to zoom in by click-and-dragging a window around objects you wish to zoom in on. Right-click to zoom out. The **Rotate** and **Pan** tools are used to rotate the structure and pan the structure left/right/up/down on the screen.

For the pointer and selection tools, right-clicking empty space in the graphics window will open up the graphics shortcut menu, which provides quick access to graphics window commands.

The Zoom Tool.

Zoom Window
Click-and-drag a window around a portion of the structure to zoom in onto that portion of the structure.

One-Click
Click an empty area of the graphics window to zoom in by 2.5x. (Or right-click to zoom out.) Or, click a structural object to zoom in by 2.5x and center the display on that object. (Or right-click to zoom out.)

The Pan Tool.

Click-and-Drag
Move the structure on the screen to pan left, right, up, and down.

The Rotate Tool.

Click-and-Drag
Move the mouse in the graphics window to rotate the structure. Drag horizontally to rotate the structure about the y-axis. Drag vertically to rotate the structure about the x-axis.

Hold `SHIFT`
And drag horizontally to rotate the structure about the z-axis.

The Pointer Tool.

Quick Info
Click structural objects to get more information about the objects. Right-click a structural object to access the shortcut menu for the object.

Quick Select
Hold `SHIFT` while clicking on an object to select *only* that object. All other objects will be deselected.

The Selection and Unselection Tools.

Select Window
Click-and-drag a window around structural objects to select/unselect the objects. (Other objects will remain as they were.) Dragging from right to left will include all objects that are at least partially within the window. Dragging from left to right will include only the objects that are entirely within the window.

One-Click
Click objects to select just one object at a time. If more than one object is under the mouse cursor, you will be prompted to choose which one to select.

Reverse
Hold `SHIFT` to do the opposite. If the selection tool is active, object will be unselected, and if the unselection tool is active, objects will be selected.

Quick Group
Hold `CTRL` and drag objects into the Structure Groups Explorer [p60] or Construction Stages Explorer [p64] to create a new group from the currently selected objects.

Switching Between Tools

In addition to clicking on their icons, there are two ways to quickly switch between the tools: shortcut keys and using the middle mouse button.

The shortcut keys are as follows:

Zoom
`F5`

Pan
`F6`

Rotate
`F7`

Select
`F8`

Unselect
`SHIFT+F8`

In addition, these other shortcut keys are provided for similar tasks:

Select All
`SHIFT+F5`

Unselect All
`SHIFT+F6`

Clicking the middle mouse button (which is generally a scrollwheel button) in the graphics window has a special behavior: it temporarily changes the current tool. Click the middle button once to switch to the rotation tool. Click it again, and whatever tool was previously active will be restored. Holding shift and control while clicking modifies this behavior:

Neither `SHIFT` nor `CTRL`
The rotation tool is temporarily activated. If the rotation tool was already active, then clicking switches to the pan tool.

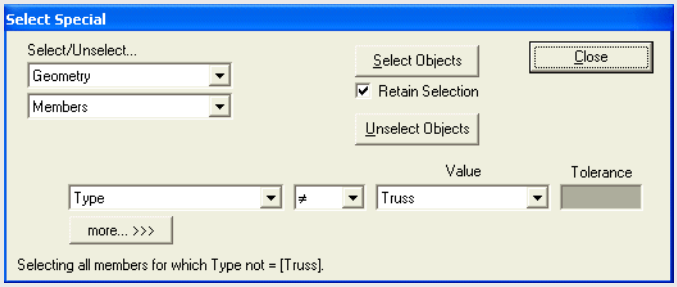
`SHIFT`
The pan tool is temporarily activated.

`CTRL`
The pointer tool is temporarily activated.

Select Special

The *Select Special* tool selects or deselects objects that have certain properties. For instance, unlike *Select All Joints*, Select Special can select "all joints whose x-coordinate is less than 10."

This tool can be found on the Selection menu under "Select Special."

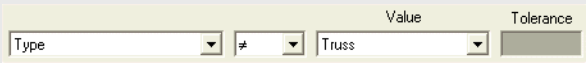


Select Special

Before you begin, instruct Select Special which type of objects to select/deselect. Choose Geometry or Loads from the first drop-down choice, and then a geometry type (joints, members, plates, springs, isolators) or a load type (joint loads, member loads, plate loads). To select/deselect loads, you must first activate the appropriate load case in the Load Cases Explorer [p58].

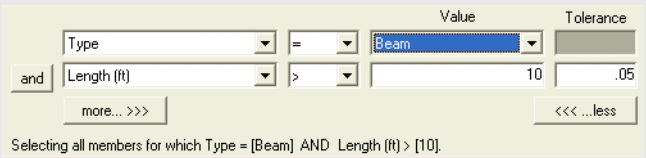
To use Select Special, you must specify criteria. Select Special will select (or deselect) only those objects that match the criteria.

Initially, you are given the option to provide one criterion, as shown in the figure below. In the figure, the user has already chosen to select/deselect members. The criterion is that a member's type is not truss. If the user were to hit the *Select Objects* or *Unselect Objects* button now, *only* the members whose type was not truss would be selected/deselected. The selection state of members which did meet the criterion would not be changed.



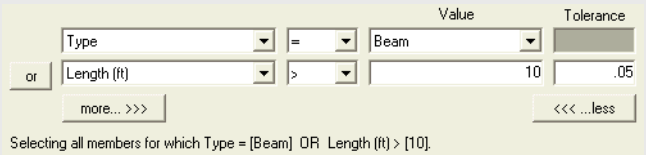
A Select Special Criterion

To specify more complex criteria, click the *more...* button. A second criterion can be specified. In the figure below, the user has specified two criteria: 1) the member's type is beam and 2) the member's length is greater than 10 ft (with a tolerance of .05 ft). Only the members that match both criteria 1 **and** 2 will be selected/deselected.



Complex Criteria

By clicking on the *and* button, the filter mode can be changed to *or*, as in the figure below. In this mode, the members that match either criterion 1 **or** criterion 2 (or both) will be selected/deselected. In this example, a **truss** member with length greater than 10 would be selected, but in the previous example it would not.



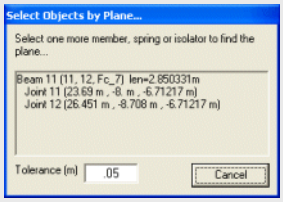
AND versus OR

As a reminder of what Select Special is about to do, a description of which objects are about to be selected/deselected is written at the bottom of the dialog.

Select by Plane

The *Select by Plane* tool selects joints and elements that lay on a particular 2D-plane.

This tool can be found on the Selection menu under "Select by Plane."



Select by Plane

Tolerance

Objects which are not exactly on the plane are selected if they fall within the given tolerance.

The plane to select can be chosen three ways.

Using 3 Joints

Click three joints one-by-one to select the plane that those three joints lay on. The coordinates of the joints must determine a plane. As you click the joints, their coordinates will appear in the Select by Plane window.

Using 2 Elements

Click two members, plates, or springs one-by-one to select the plane that the two elements lay on. The elements must lay on one plane. As you click the elements, their end-joint coordinates will appear in the Select by Plane window.

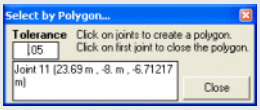
Using 1 Plate

A single plate defines a plane. Click a plate to select all objects that lay on the same plane.

Select by Polygon

The *Select by Polygon* tool selects joints and elements that fall within a 2D polygon.

This tool can be found on the Selection menu under "Select by Polygon."



Select by Polygon

Tolerance

Objects which are not exactly on the plane are selected if they fall within the given tolerance.

Click the joints one-by-one that define the boundary of the polygon. Click the first joint, again, when done to finish the polygon. As you click the joints, their coordinates will appear in the Select by Polygon window.

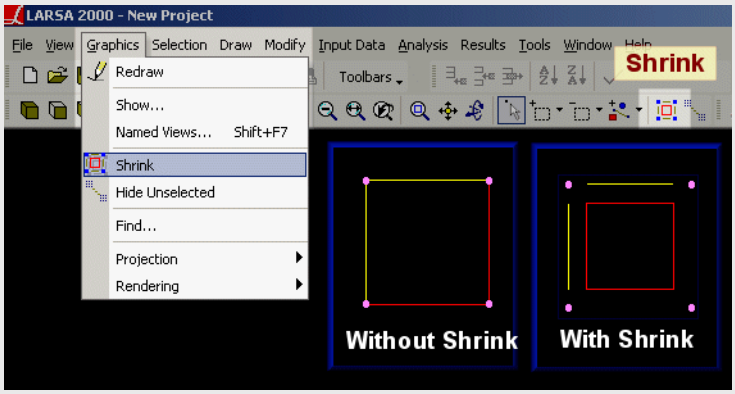
Graphics Display Options

The graphics display options control how structures are displayed in the graphics window. These options are located in various parts of LARSA 2000.

Shrink

When Shrink is turned on, members, springs, isolators, and plates are displayed at a fraction of their true size. Shrink is useful to clearly see the end points of members, springs, and isolators and the boundaries of bordering plates. When Rendering is turned on, Shrink is especially useful for preventing member sections from overlapping at joints. The Shrink default is to display elements at 75% of their true size when Shrink is turned on. The percentage can be changed from the **Tools** > **Options** dialog. To turn Shrink on and off, click the Shrink tool from the **Graphics** menu or the graphics toolbar.

This tool can be found on the Graphics menu under "Shrink."



Shrink

Orthographic/Perspective Projection

The graphic window may present the structure in one of two projections: orthographic or perspective.

Under **orthographic** mode, structural elements do not become smaller as they become farther away (there is no perspective). Orthographic mode is most useful while modeling the structure, as it does not have any distortions.

On the other hand, **perspective** mode is closer to a realistic display of the structure and generally makes the structure easier to view. Because of the distortions inherent in a perspective view, it is sometimes confusing when modeling. Perspective view can also be highly distored when looking at certain structures from certain directions.

This tool can be found on the Graphics menu under "Projection."

Icon Size

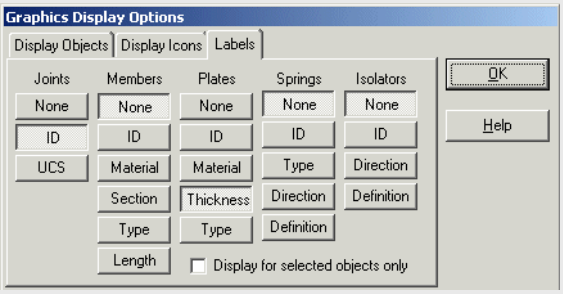
Certain elements are displayed as icons, such as supports and loads. The size of these icons may need to be adjusted when working on very large structures, as the icons may overcrowd the display. The Icon Size may be adjusted from the **Show** dialog, which is on the Graphics menu or on the graphics right-click menu when using the Pointer Tool. The Icon Size may be Small or Large. Large is the default and is suitable under most circumstances.

This tool can be found on the Graphics menu under "Show."

Element Labels and Icons

Element labels (such as joint numbers, plate thicknesses, spring types) can be displayed beside each element. Each element type (joint, member, plate, spring, isolator) may only have at most one label type turned on at a time. Orientation and Rigid Zone Icons may also be displayed on members. These options are available in the **Show** dialog, which is on the **Graphics** menu or on the graphics right-click menu when using the Pointer Tool.

This tool can be found on the Graphics menu under "Show."



Element Labels

The labels are arranged into columns for each type of object: joints, members, plates, springs, and isolators. In each column are the types of labels that can be shown for the objects, such as ID, material, and thickness. Click a label name to turn on that label.

Rendering

Members, plates, and other elements may be displayed "rendered," where they are drawn with their actual dimensions, instead of as simple lines and polygons.

This tool can be found on the Graphics menu under "Rendering."

There are three rendering modes:

None

No special rendering is used. The structure is drawn as a wireframe model.

Quick

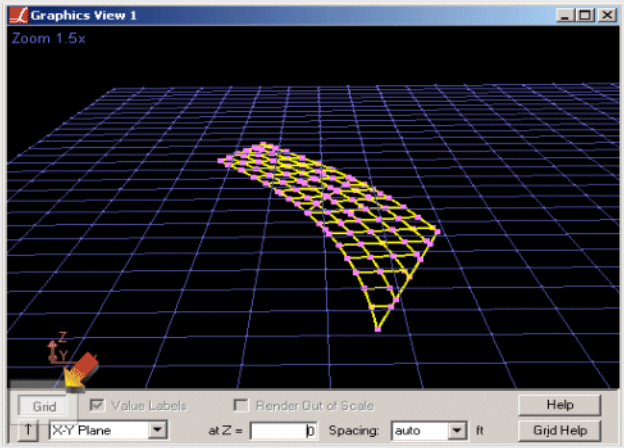
Quick rendering is used. Members are drawn with their actual shape and dimensions. To draw members out of scale so that their sections are more easily seen, click the *Render Out of Scale* checkbox at the bottom of the graphics window.

Complete

Full rendering is used. Both members and plates are drawn with their actual shape and dimensions, and lighting effects are also used. To draw members out of scale so that their sections are more easily seen, click the *Render Out of Scale* checkbox at the bottom of the graphics window.

Graphics Window Grid

When using the graphical windows, LARSA can provide a two-dimensional grid. The grid is toggled on and off using the *Grid* button at the bottom of each graphics window.



The Graphics Window Grid

The grid defaults to the X-Z plane (where y = 0), with grid extents and spacing dependent on the size of the structure and the current zoom factor.

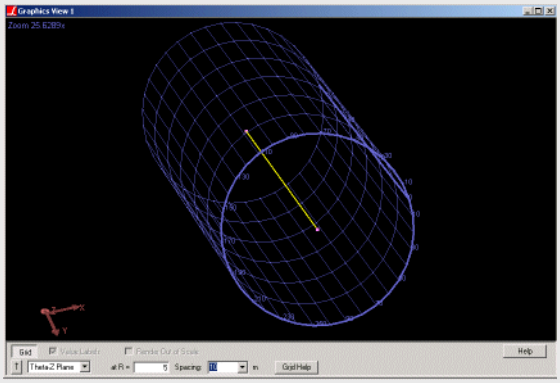
The grid may be shifted in and out of the X-Z plane (or whatever plane is currently active) by using the grid level input, which is marked by the label "at Y =."

Grid line spacing can be entered manually or selected from the drop-down list, or it can be set to automatic. On automatic mode, the spacing is chosen based on the current zoom factor. Automatic spacing is useful when the exact grid spacing is not important, but it should be turned off (by entering or selecting a number for the spacing) when modeling the structure.

> Values for grid line spacing that result in too many grid points being displayed will result in no grid being displayed, and a "grid size too small" error shown in the grid panel. In this case, decrease the extents of the grid or increase the line spacing.

Grid extents are also chosen automatically by default. Before modeling the structure, entering values for the extents manually will prevent the extents from changing automatically. To set the extents manually, click the up arrow, which is directly below the Grid button. In the Grid Extents box which will appear at the lower-left of the graphics display, the grid extents may be entered. Additionally, separate grid spacing values can be entered for each of the three axes. Automatic extents may be turned on and off by clicking the "Auto" checkbox in the Grid Extents box. To close the Grid Extents box, click the up arrow again.

The grid is drawn in the active user coordinate system, which might not be a rectangular system. In the cases of cylindrical or spherical systems, the grid planes are R-Theta, Theta-Z, and R-Z and R-Phi, Phi-Theta, and R-Theta, respectively. Setting grid extents (which will control R, Z, Theta, and Phi instead of X, Y and Z) and separate grid spacing for each axis is usually necessary for these coordinate systems.



A Cylindrical Grid

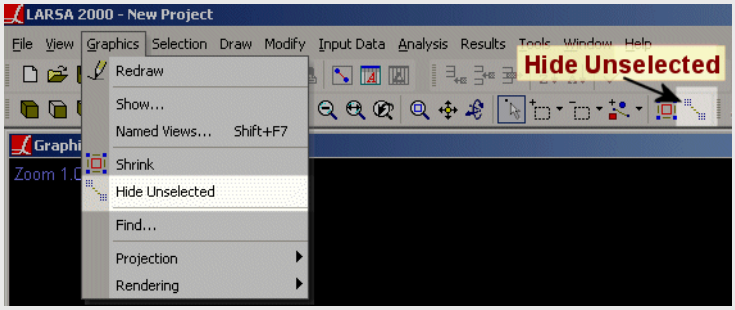
In addition, in non-rectangular coordinate systems there are "degenerate" grids, such as the Phi-Theta plane at R=0, which result in a grid being drawn at a single point or along a single line. In the case of this example, there is only one point at R=0, so there is no grid to be drawn. The grids displayed in these cases may be misleading.

Also misleading are grids displayed perpendicular to the viewing plane when orthogonal projection is being used. In this case, the grid is drawn "in to" the computer screen, resulting in a single line being displayed, or nothing at all. In this case, perspective display should be turned on. See Graphics Display Options [p46].

Hide Unselected

When modeling a structure or investigating results in a particular region of the structure, it is often useful to hide part of the structure so that the remaining elements are unobscured. The Hide Unselected feature does just that.

This tool can be found on the Graphics menu under "Hide Unselected."



Hide Unselected

Hide Unselected takes all of the unselected objects and makes them invisible. They will come back when Hide Unselected is turned off.

To hide part of the structure...

- Select the part of the structure that you want to remain visible (or unselect the part of the structure that you wish to hide.)
- Activate Hide Unselected from the Graphics menu or the graphics toolbar. The unselected structural elements will be hidden.
- The Hide Unselected icon in the menu and toolbar will be highlighted to indicate that Hide Unselected has been turned on.

To restore the hidden part of the structure, click the Hide Unselected button again.

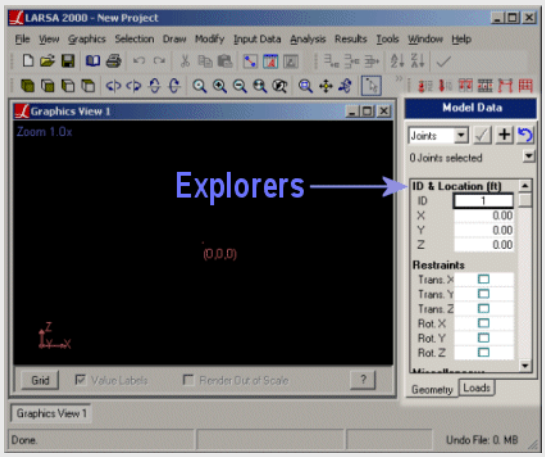
Invisible items that later become selected will be turned visible again so that selected items

are always shown.

Explorers

Navigating through a large project can be difficult, no matter what software package is used. While invaluable, spreadsheets often present too much data at once. Introducing the solution to this problem, LARSA 2000 uses *Explorers* to summarize large quantities of data into manageable units which can be quickly edited. The Explorers are explained in these sections.

Using LARSA 2000's Explorers



LARSA 2000 Explorers

LARSA 2000's Explorers are shown at the right side of the screen. There are five explorers: Model Data, Load Cases, Structure Groups, Construction Stages, and Analysis Results.

Above the active Explorer are five small buttons, one for each Explorer: Model, Load, Group, Stage, Results. Click one of these buttons to activate that Explorer.

Below the first row of buttons is a second row for choosing whether to view one or two Explorers at once. The left-most button turns on one-Explorer mode. The next button turns on vertical-split mode, in which two Explorers are shown with one above the other. The third

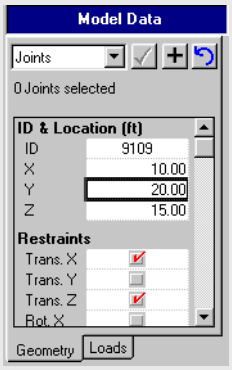
button turns on horizontal-split mode, in which two Explorers are shown side-by-side. When two Explorers are show, the last button is enabled and when clicked will reverse the positions of the two Explorers.

To resize explorers, bring the mouse over the left border of the explorer. When the mouse becomes a horizontal arrow, drag the border sideways.

If screen space is limited, the explorers can be hidden. To hide the explorers, choose the **View** menu and uncheck the command *Show Explorers*. To show the explorers again, choose the same command again.

Model Data Explorer

The Model Data Explorer is used for quickly adding new elements to the structure and modifying the properties of existing elements. This Explorer is similar to the Properties window of AutoCAD and other applications.



The Model Data Explorer

The explorer, displayed on the right-hand side of the screen when LARSA starts, can display properties of geometry entities (joints, members, plates, etc.), load entities (joint loads, member loads, etc.), as well as custom properties through the use of plugins, such as steel design parameters. Each category of entities (geometry, loads) is separated into tabs at the bottom of the explorer. The class of entity to be modified is shown at the top.

The values displayed in the explorer reflect the properties of the currently selected objects. If more than one object is selected, the properties that all selected objects have in common are displayed, and properties which are not common are left blank.

To make a modification to the properties of selected objects, make the appropriate changes in the listing of properties, and then click the checkmark button. Changes are not finalized until the checkmark is clicked to confirm them. Properties which have been changed are shown in bold, until they are confirmed.

The reset button (the arrow bending backward) will abort any changes and update the listing to display the current properties again.

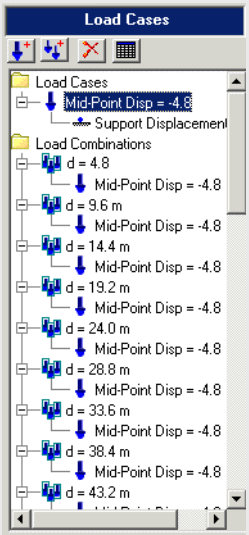
To add a new object, enter the properties of the object, and then click the add button (the plus sign). Be sure to set the properties first, as the Explorer will continue to display the properties of *all* selected objects afterwards, not just the new object. For geometry items, the add button will not be available unless a unique ID for the object has been supplied.

If no objects of the type being modified are currently selected, then the explorer will display default values, and the checkmark will not be available.

The Model Data Explorer can be used to quickly select objects by their ID. If the Explorer is showing properties of selected joints, enter the ID of a joint in the ID field and hit enter, and that joint will be selected. All other objects will be deselected. You may also enter ranges and lists of IDs (e.g. 1-10,25,26).

Load Cases Explorer

The Load Cases Explorer is used to rapidly create and edit load cases and load combinations.



The Load Cases Explorer

The Explorer lists all load cases and combinations in the project and indicates the contents of each load case beneath the case. In the explorer shown here, the blue load icons denote a load case or combination.

Load case properties can be edited directly from the Explorer. The Explorer is also used to select the *active* load case. Before modifying a load case, it must be selected in the Explorer.

Using the Load Cases Explorer

To select a load case or combination, click it. It becomes the active load case for viewing and editing. Graphics windows will show the loads within the active load case.

To Create a New Load Case, click the New Load Case button, which has a plus sign next to a single load icon.

To Create a New Load Combination, click the New Load Combination button, which has a plus sign next to a load group icon (two arrows).

Right-click a load case or combination to modify its name, properties (including self-weight application), or to duplicate the case.

Double-click a load case, or click the spreadsheet button in the Explorer, to edit the contents of the load case numerically in a spreadsheet.

Load cases can be added to load combinations by **clicking-and-dragging** the load case onto the appropriate load combination. To remove a load case from a load combination, simply drag the case out of the combination. To edit a load case's factor as applied in a load combination, right-click the load case.

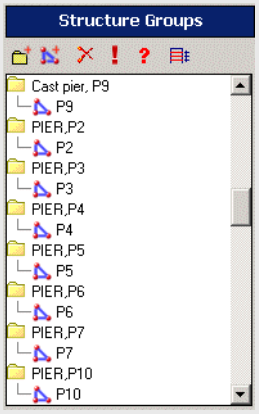
To delete a case, select the case and then click the delete (X) button.

For More Information

- Static Load Cases [in *LARSA 2000 Reference*].
- Load Combinations [in *LARSA 2000 Reference*].
- Self Weight [in *LARSA 2000 Reference*].

Structure Groups Explorer

For medium- to large-sized structures, breaking down a structure into groups can save time by allowing quick access to a particular part of the structure. Structure Groups do just this.



The Structure Groups Explorer

Structure groups are groups of elements in your structure: members, plates, and springs. Groups allow you to quickly select and unselect various geometric portions of a structure, for using the Model Data Explorer [p56], or for viewing results [p119] in particular portions of the structure. Groups are also used in staged construction analysis [see "Setting Up the Model" in *LARSA 2000/4th Dimension: Staged Construction Analysis*] and as design groups for Steel Design [p157].

Forming structure groups is as simple as hitting the new-group button, or as simple as clicking-and-dragging the contents of the graphics window into the Structure Groups Explorer. (See An Overview of Graphics & Selection [p38].)

Structure groups may be organized into **group folders**, and folders may be organized within folders as well.

Clicking on a structure group in the explorer will unselect all objects and then select only the objects within that group. **Clicking a group folder** will select all of the objects contained in all groups within that folder.

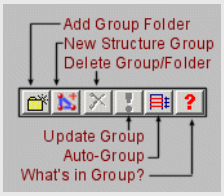
Hold `CTRL` while clicking structure groups to select more than one group.

Geometry elements may be members of multiple groups.

Groups can also be used to color different parts of a structure.

Using the Structure Groups Explorer

The Structure Groups Explorer commands are explained below.



Structure Groups Explorer Commands

Adding Groups

Click the New Structure Group icon to add a new structure group to the project.

If no group or folder is selected, the folder will be added at the top level. If a folder is selected, the new group will be added within the selected folder. If a group is selected, the new group will be added at the same level as the selected group.

Or, select the pieces of the structure that you would like to group together. Then, hold `CTRL` and drag the selection from the Graphics Window into the Structure Groups Explorer. The selected geometry elements are added to a new group.

Adding Folders

Click the Add Group Folder icon to add a new structure group folder.

If no group or folder is selected, the folder will be added at the top level. If a folder is selected, the new folder will be added within the selected folder. If a group is selected, the new folder will be added at the same level as the selected group.

Updating a Group

To change the contents of a structure group, **first** select the group in the explorer. Then, select only the object and all of the objects that you would like to be the new contents of the group. Finally, click the Update button to update the group with the selected objects.

For example, to remove objects from a group, simply select the group in the explorer, unselect the objects to delete, and click Update.

Setting a Group's Color

To change the color of all objects in a group, right-click the group and choose **Color...**. To reset the color of a group, click **Color...** and choose *Reset*.

If an object is in multiple groups with colors, the color for the object is chosen arbitrarily.

Viewing and Editing Groups

To view and edit the contents of a structure group, select it in the explorer and click the What's in Group? button. You can add and remove objects from groups here as well.

Managing Groups and Folders

Groups and folders may be moved into folders by **clicking-and-dragging** them on top of folders. Bring a group or folder to the top level by dragging it into the empty whitespace of the explorer.

While dragging a group, **hold `CTRL`** to make a copy of the group in the folder where you drop the group. No new members/plates will be created. Only a new structure group will be formed with the same contents as the old group.

To delete a group or folder, select the folder and then click the Delete button.

Auto-Grouping

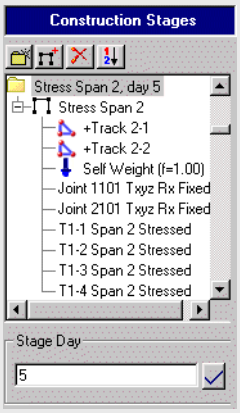
LARSA can automatically divide a structure into groups of floors and walls if the floors and walls are in standard planes (planes parallel to the global x-y, x-z, or y-z planes).

Click the auto-group button and choose which axis represents elevation. (If the floors are on planes parallel to the x-y plane, choose the z-axis.)

Set a tolerance level so that objects at nearly-identical elevations are grouped together.

Construction Stages Explorer

The Construction Stages Explorer is used for Staged Construction Analysis to define the order in which a structure is built.



The Construction Stages Explorer

The Construction Stages Explorer shown here is a for a typical time dependent stage analysis of a 2-span cast-in-place bridge structure construction. It included assembling, applying loads, activating supports and stressing tendons. Each stage has a day assigned as well.

The Explorer is explained in detail in the Staged Construction Reference [see "Setting Up the Model" in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

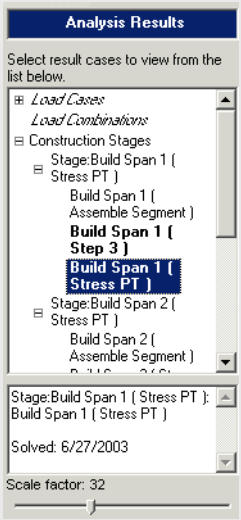
For More Information

- Construction Stage Editor [p109].
- For an overview of construction analysis, see Staged Construction Guide [in *LARSA*

2000/4th Dimension: Staged Construction Analysis].

Analysis Results Explorer

To efficiently explore results from analyzed load cases, each case of results is listed in the Analysis Results Explorer.



The Analysis Results Explorer

Result cases (to be distinguished from input load cases [see "Static Load Cases" in *LARSA 2000 Reference*] in the Load Cases Explorer [p58]) are the different scenarios that have been processed by the analysis engine. These cases are the post-processing counterparts of load cases, load combinations, and construction stages [see "Staged Construction Guide" in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

Result cases do not always correspond directly to input cases. In Moving Load Analysis [in *LARSA 2000 Reference*], one input load case is used to define the application of moving load patterns to the structure, and the analysis engine generates numerous result cases representing the possible positions of the load patterns on the structure. In these cases, the resulting generated cases are grouped together in the Analysis Results Explorer.

Result cases are displayed in a tree in the Analysis Results Explorer. Groups are indicated by

a plus or minus sign next to them. Click a plus sign to expand the group and see what result cases and groups it contains. Click a minus sign next to a group to collapse the group and hide its children.

Using the Analysis Results Explorer

Results for a case can be viewed by clicking the result case, and then selecting either a graphical result to display or a result spreadsheet from the **Results** menu. You can also **right-click** a case to access its spreadsheet results quickly. For more on accessing results, see Getting Results [p119].

Select cases by clicking them. Hold `CTRL` while clicking to select multiple cases. Hold `SHIFT` to select a range of cases. `CTRL`+click a group to select or unselect all of the cases in the group. Selected cases are shown in bold. The last case clicked is highlighted.

When viewing results in Results Spreadsheets [p127], results are displayed for each case selected in the Analysis Results Explorer. When *envelope* is turned on in a results spreadsheet, then the spreadsheet displays an envelope of the result cases selected in the Analysis Results Explorer.

When viewing results graphically [p120], the results are shown for *either* the last result case selected or for an envelope of all of the result cases selected in the Analyis Results Explorer. See Viewing Results Graphically [p120].

The date and time that a result case was solved is displayed at the bottom of the Explorer, and the display scale factor for displaying deformed structure, mode shape and force diagrams can be easily adjusted from the explorer by dragging the slider at the bottom.

Use with Special Result Cases

Result cases created after an analysis is run (e.g. Linear Result Combinations [p131], Extreme Effect Groups [p134] and Influence-Based Analysis [in *LARSA 2000 Reference*]) are also shown in the Analysis Results Explorer.

Their properties can be edited by right-clicking on the case and choosing **Properties**.

They can also be deleted by right-clicking on the case and choosing **Delete**.

For More Information

- For more on accessing results, see Getting Results [p119].

Modeling Tools

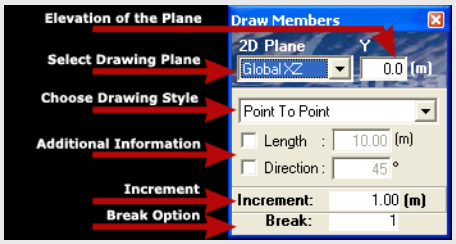
LARSA provides a handful of modeling tools to help in the creation of your structure. The modeling tools are in the Draw and Modify menus.

The **Draw** menu's tools allow you to create new objects in your structure.

The **Modify** menu's tools are for modifying existing objects.

Drawing Geometry and Loads

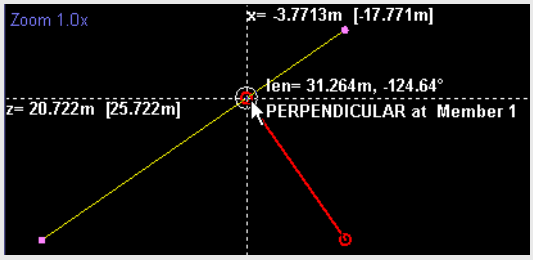
There are number of methods to create a model in LARSA 2000, one of which is drawing.



Draw Toolbox

With the draw toolbox, you can select the drawing plane and the elevation in which to draw. As you move the mouse, a crosshair will continuously show the coordinates of the mouse.

The mouse snaps to objects, midpoints, and perpendiculars. Useful information is displayed right next to the mouse pointer or on the status bar as the mouse is moved over these targets.



Graphics View: Drawing a Member

On the toolbox, set the increment (grid snap size) for the movement of the mouse. If the increment is 2 feet, when the mouse is not over any of the above targets it will always snap to locations that are multiples of 2 feet on both axes.

During the draw process, the Model Data Explorer [p56] automatically becomes active to let you specify the properties of the geometry you are drawing. Set properties in the model data explorer *before* drawing so that the new objects you create take on those properties.

Right-clicking during the draw process will generally reset the current drawing command. When drawing plates, however, right clicking after specifying the third joint of the plate will create a three-node plate. For springs, right clicking after specifying the first node will create a grounded spring. Right clicking also has a special behavior when changing the support conditions of a joint (see below).

Drawing Joints

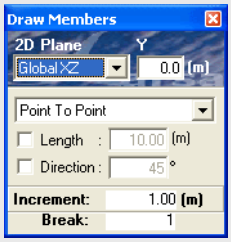
Draw Joint Toolbox

This tool can be found on the Draw menu under "Geometry > Joints."

Click in the graphics window to draw a joint at the location under the mouse. The Graphics Window Grid [p50] may be useful.

An existing joint can be moved by clicking on that joint and then clicking its new location. It is also possible to change the support conditions of a joint with a right-click: Change the support conditions in the Model Data Explorer [p56] and then right-click joints to apply the support conditions to joints.

Drawing Members



Draw Member Toolbox

This tool can be found on the Draw menu under "Geometry > Members."

Drawing members can be accomplished by clicking a start location and then an end location. If you do not click on joints, joints will be automatically created at the location of the click.

Members can be broken into multiple parts while drawing by specifying a *Break* number on the draw toolbox. You can also draw members with a pre-set length and/or angle by turning on the option on the draw toolbox (clicking it so it is checked) and then entering the length and/or angle.

There are three drawing styles available:

Point to Point
Click on a start location and then an end location. The member will be drawn between the points. Click the next member's start location to draw another member.

Polyline
Click on a series of points one-by-one to draw a continuous line of members. Right-click to stop drawing the polyline.

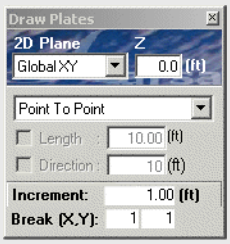
Arc
Use this to draw a circular arc of members.

1. Click the center of the circle that the arc will go around.
2. Click the starting location of the arc. You will see a line from the first point to the

mouse indicating the radius of the arc.

3. Click the ending location of the arc. It must be on the circumference of the same circle.

Drawing Plates



Draw Plate Toolbox

This tool can be found on the Draw menu under "Geometry > Plates."

There are two ways to draw plates. With the *point-to-point* method, click on joints in counter-clockwise order to form a plate. New joints will automatically be created if there is no joint at a location clicked. If you are drawing a triangular plate, right-click after drawing the third point. (If you do not right-click, LARSA assumes you are drawing a quadrilateral plate.)

Or, with the *rectangular* method for quadrilateral plates, click on the two opposing points of the plate to create the plate.

You can also draw plates with a pre-set width and height. Turn on a dimension restriction by clicking it (so that it is checked) and then enter the appropriate value.

You can also break plates while drawing by entering *Break* X and Y values. Plates will be broken in X pieces along its width and Y pieces along its height. The width of a plate is from the first joint clicked to the second joint clicked. The height of a plate is from the second joint clicked to the third joint clicked.

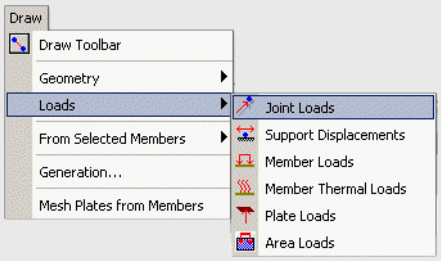
Drawing Springs

This tool can be found on the Draw menu under "Geometry > Springs."

You can draw springs by clicking on locations in the graphics window as you would for drawing members (see above).

To draw grounded springs, click on a joint and then right-click to let LARSA know that you are done drawing.

Drawing Loads



Draw Toolbox

This tool can be found on the Draw menu under "Loads."

Joint loads, support displacements, member loads, member thermal loads, plate loads, and area loads can all be drawn with the mouse.

Begin drawing by first clicking on the appropriate menu, as shown to the right.

Enter the properties of the loads you are about to draw in the Model Data Explorer [p56].

Then click on individual joints, members, or plates, as appropriate, to apply the load. To apply the load to many objects at once, window the objects as if you were selecting [p38] them.

Undo/Redo

The Undo/Redo capability of LARSA 2000 is one of the defining features of the program. Undo/Redo allows the user to reverse a change to the project — perhaps undoing an accidental change — and then put it back again.

In LARSA 2000, all operations can be undone, save a small handful. Undo/Redo works seamlessly with the spreadsheets, graphical editing tools, and Explorers. There is no limit to the "Undo History," allowing you to undo all operations in the open project since the current LARSA session began. Undo/Redo data is not saved with the project, so it lasts only as long as LARSA 2000 is running.

The Undo and Redo commands are on the **Modify** and **Edit** menus. (Only one of the two menus is visible at a time. The **Modify** menu is available for the Graphics windows, and the **Edit** menu is available for the spreadsheet windows.)

When the Undo and Redo commands are available, a small description of the action that can be undone appears in the menu item. The Undo command, for instance, might read "Undo Add Joint," indicating the last operation that can be undone is adding a new joint. Choosing Undo will delete that joint, as if it had never been created.

Redo puts back the last operation that was undone. As in the last example, Redo would recreate that joint with the properties that it had before Undo removed it.

Undo/Redo data is stored in a temporary file. The size of that file is displayed in the status bar. Only on very large models does the Undo/Redo file grow beyond a megabyte.

A small number of operations cannot be undone, including applying a unit conversion to the entire project. After such operations, all undo/redo data is cleared. Earlier operations can no longer be undone. The user is prompted to confirm these operations and is warned of the consequences.

Erase and Delete

The Erase and Delete commands, found on the **Modify** menu, remove geometric and load entities from the model.

Erase

The Erase command deletes all selected geometric elements: joints, members, plates, springs, and isolators. All loads applied to the erased geometry are deleted.

If a joint is an end-point of a member and that member is *not* selected, then the joint cannot be erased.

Delete Loads

The Delete Loads command removes all selected loads. Loads, like geometric elements, have a selected state. Only the selected loads are removed. Geometric elements that are not selected cannot have selected loads, so loads on unselected geometric elements are guaranteed to be ignored. No geometric elements are modified by Delete Loads.

Break, Merge, and Join

Break, merge, and join are three tools for manipulating joints, members, and plates. These tools are on the **Modify** menu.

Break Members, Break Plates

These tools are used to break the selected members and plates into pieces. All of the loads placed on the initial members and plates will be appropriately reassigned or redistributed onto the pieces.

For **breaking members**, the number of segments *n* to break each member into is specified. By default, the members will be split evenly *n* ways. However, the position of the *n*-1 break points can also be specified. Break points are given in relative terms, where 0 is the beginning of the member (the start joint) and 1 is the end of the member (the end joint). 0.5 specifies the middle. To specify break points, enter the relative positions and separate each position with a comma.

For **breaking plates**, the number of horizontal and vertical segments to break each plate into is specified. If 3 horizontal segments and 2 vertical segments are specified, each plate will be broken into 3*2 = 6 pieces. The direction meant by horizontal (X) is the direction of each plate's local x-axis. A plate's local x-axis is from its first joint to its second joint. (See Plates [in *LARSA 2000 Reference*].)

Repeatedly breaking square plates can serve as a simple form of plate meshing.

Merge Joints

The Merge Joints command scans the selected joints looking for joints that have the same location (coincident). A tolerance can be specified to allow for rounding error.

After selecting *Merge Joints* from the menu, click *Check* to begin scanning. This process may take some time if thousands of joints are selected.

Once the check is done, a list of each group of coincident joints is presented. Each row lists all of the joints that were found at a particular location. Uncheck a row if you do *not* wish to merge those joints into one joint. Because a tolerance may be used, the coordinates of coincident joints may not be identical. In that case, for each row, you can choose which joint's coordinates to merge the joints into.

Finally, click *Merge* to perform the merge on any checked rows.

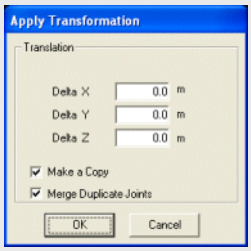
Join Members

Join Members looks through all of the selected members for any chain of members that forms a straight line. Join Members then joins all of those contiguous members into one, deleting any intermediate joints. If any intermediate joints cannot be deleted, then the chain will not be joined.

All of the loads placed on the initial members will be appropriately reassigned or redistributed onto the new member.

Transformations

LARSA provides various commands found on the **Modify** menu to alter and duplicate parts of a structure. Among those commands are the four transformations: Translate, Rotate, Mirror, and Scale.



Transformations

These commands operate on all of the selected objects in your project. For example, when using the Translate command, only the selected members will be moved. The unselected members will not be affected.

The four commands have a *Make a Copy* option. The *Make a Copy* option instructs LARSA to first duplicate all of the selected objects, and then perform the operation. When making a copy, the *Merge Duplicate Joints* option instructs LARSA to check that it does not create new joints where joints already exist.

All transformations are with respect to the active coordinate system [p98]. If no user coordinate system is active, the global coordinate system is used.

Translate

Moves the selected objects a certain distance in a given direction. The specified x, y, and z values will be added to the x-, y-, and z-coordinates of all of the selected joints.

Rotate

Rotates the selected objects about the origin by the angles specified. Specifying an X Rotation of 30 degrees will rotate the selected objects around the x-axis by that

angle.

Mirror

Reflects the selected objects over the specified plane.

Scale

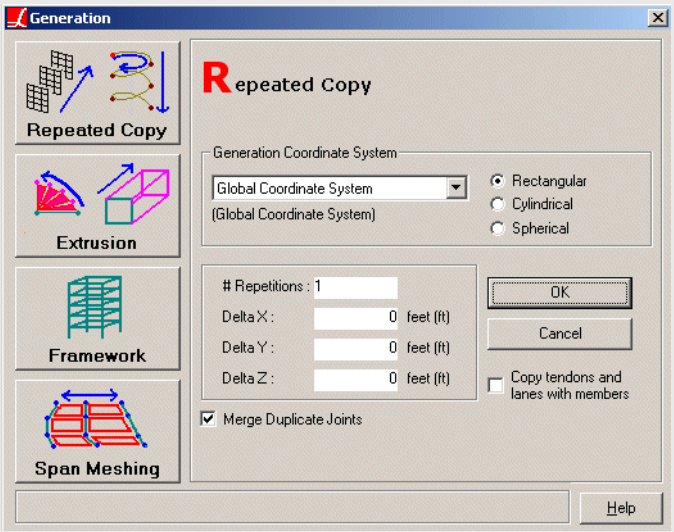
Multiplies the x-, y-, and z-coordinates of the selected objects by the specified scale factors.

Generation Tools

The Generation tool quickly generates new structural elements based on existing ones. Generation includes forms of copying, extrusion, and meshing.

This tool can be found on the Draw menu under "Generation."

There are four types of Generation commands. Select the command by clicking on its name in the listing on the left of the dialog.



The Generation Tools

Repeated Copy

Repeated Copy 1) clones the selected objects, 2) translates (moves) the objects in a specified direction, and then 3) repeats. The number of copies (repetitions) and the offset of each subsequent copy (Delta X, Delta Y, Delta Z) can be specified.

Repeated Copy can be applied in either the Global Coordinate System or in any user coordinate system [p98] of any type. In a rectangular UCS, the Delta X/Y/Z values are applied in the directions of the UCS's axes. In a cylindrical or spherical coordinate system, Delta R/Theta/Z or Delta R/Phi/Theta is specified, respectively. In those cases, the translations of each object will *not* all be parallel, but instead they will curve about the axes of the UCS. Arcs, spirals, and helixes can be created using Repeated Copy in a non-rectangular coordinate system.

The *Merge Duplicate Joints* option instructs LARSA to not create new, separate joints at locations where joints already exist. For instance, if a joint at (10, 0, 0) is selected and you specify a repeated copy with Delta Y = 5 *and* a joint already exists at (10, 5, 0), then the existing joint at (10, 5, 0) will be used as the copy, rather than creating a second joint at that location. If this option is not checked, coincident joints may be created.

Extrusion

Extrusion will extrude selected joints into members or selected members into plates along the given direction. The extrusion will repeat a specified number of times.

The direction of extrusion follows the coordinate system specified in the *Generation Coordinate System* box. See Repeated Copy (above) for more information on using a UCS or non-rectangular coordinate system.

Select whether to extrude members, plates, or both in the *Element Type* drop-down choice.

If only joints are being extruded into members, then all of the selected members will be copied in tandem with the extrusion in the same manner as Repeated Copy.

When extruding members, the new members will be given the type specified in the *Member Type* drop-down choice.

See the description of Merge Duplicate Joints, above.

Framework

Framework generates a framework of members based on a set of floor height and bay width/depth parameters. Framework can be used to quickly set up a building model.

The resulting framework has one base corner at (X Start, Y Start, Z Start). It's width, depth, and height are `# of Bays (X) * Bay Width (X)`, `# of Bays (Y) * Bay Width (Y)`, and `# of Stories * Story Height`, respectively.

Check the *Ground Floor Members* option to generate members in the plane of the ground floor. If the option is not checked, the framework will start with columns coming out of the ground, and no members resting on the ground.

See the description of Merge Duplicate Joints, above.

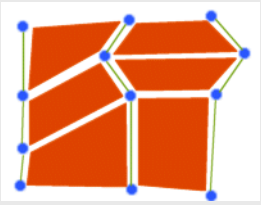
Span Meshing

Span Meshing is a specialized tool to generate a mesh of plates between two parallel chains of members. Span Meshing will also work for a series of parallel chains of members.

A chain of members is a series of contiguous members, such as from joint 1 to joint 2, from joint 2 to joint 3, from joint 3 to joint 4.... For span meshing, the chains must be in a a straight line, or close to it. In addition, each of the chains must be approximately parallel. Furthermore, each chain *must have the same number of members*.

Given a pair of member chains that satisfy these constraints, Span Meshing will create one plate between the chains for each pair of corresponding members. The plate will span the existing joints on the member chains.

The simplest case of Span Meshing is for two parallel members: member 1 goes from joint 1 to joint 2, and member 2 goes from joint 3 to joint 4. Span meshing will create a plate around joint 1, joint 2, joint 4, and joint 3. The diagram below may illustrate Span Meshing better.



Span Meshing

Here there are three member chains (in grey), each composed of three members. Each of the three chains are close to straight lines, and the three chains are close to parallel. Span Meshing generated the six plates (in red) by connecting the adjacent chains at their corresponding joints.

See the description of Merge Duplicate Joints, above.

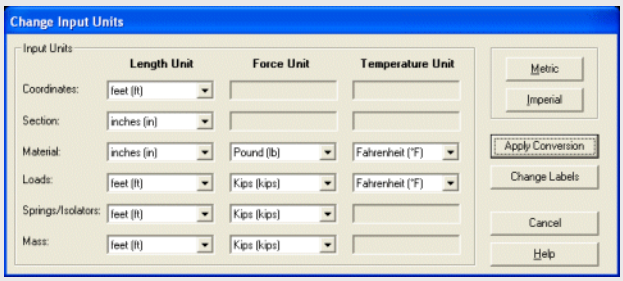
Project Options

The follow sections describe global project options.

Model Units

Numeric entries in LARSA are tied to several categories of units used by the project. The units used by the open project can be edited using the Units dialog.

This tool can be found on the Input Data menu under "Units."



Changing Model Units

The Units dialog only applies to numeric entries for model data. Results units are selected in the results units dialog [p141].

The Units dialog is arranged into a simple matrix. The rows represent categories of units:

Coordinates
Used for joint locations and other measurements of coordinates.

Section
Used for section properties, including section dimensions, and tendon offsets within section bounds.

Material
Used for material properties.

Loads
Used for load positions and magnitudes.

Springs/Isolators
Used for spring and isolator properties.

Mass
Used for mass elements and other properties.

The units for each category can be independently changed so that, for instance, joint locations need not be in the same units as section dimensions.

The columns of the matrix represent unit types: Length, Force, and Temperature. The unit in the cell in the Force column and Loads row will be used for Force (or derived) units for load entities.

A unit can be selected for most cells in the matrix. Some cells are gray, such as Coordinates/Force, for which unit selection is not applicable.

Click the *Metric* or *Imperial* buttons to immediately change all units to the standard metric or imperial units.

Click *Apply Conversion* to convert all numeric values in the projects to the new units you have specified.

Click *Change Labels* to set the project's units without applying a unit conversion. This is useful if you have entered all numeric values in the wrong units. For instance, if you have entered coordinates in meters but coordinate length units were set to feet, you can change the coordinate length units to meters and then click *Change Labels*. The coordinates will be left unchanged but they will then be labeled with the correct unit.

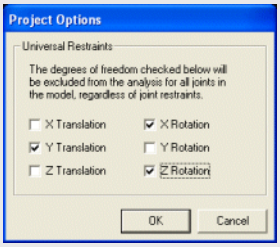
Click *Cancel* to discard any changes you have made to the units.

The Units dialog only applies to numeric entries for model data. Results units are selected in the results units dialog [p141].

Universal Restraints

Universal restraints are restraints in translation and rotation applied to all joints in the project. They are useful for constraining a model into two-dimensions.

This tool can be found on the Input Data menu under "Universal Restraints."



Setting Universal Restraints

The degrees of freedom selected as universal restraints are excluded from all analyses of the open project for all joints in the model, regardless of the joint's own restraints.

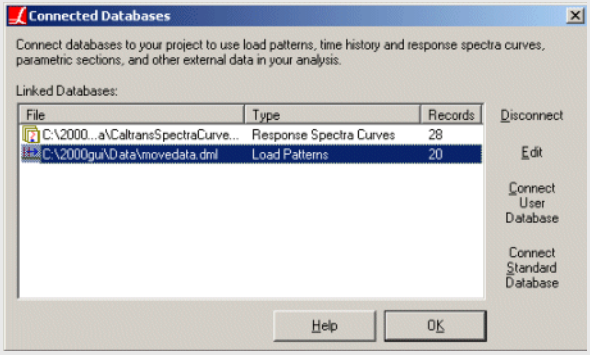
Click a degree of freedom to make it universally restrained.

The standard six degrees of freedom can be restrained:

- X Translation
- Y Translation
- Z Translation
- X Rotation
- Y Rotation
- Z Rotation

Connecting Databases

Some types of data which may be common to more than one project are saved externally to a project in LARSA database files. You can connect databases to your project using the *Connect Databases* dialog box.



Connect Databases Dialog Box

This tool can be found on the Input Data menu under "Connect Databases."

Moving load pattern data is one of the types of data that gets saved into an external database. By attaching a moving load pattern database, you make all the patterns in that database available to the project. To connect LARSA's standard database of American load patterns, click *Connect Standard Database* and open the file "movedata.dml."

Also all **time history curves** and **response spectra curves** are saved to an external database and can be connected to the open project.

The *Connect Databases* dialog box is also used to connect section databases that are created in the Section Composer [see "Section Composer User's Guide" in *LARSA Section Composer*].

For moving load, time history, and response spectra databases, only one database of each type can be connected to the project.

To connect a database: Click *Connect User Database* and select the database file using the browse dialog box. Once the database is connected, it will appear in the *Linked Databases* list.

To disconnect a database: Select the database from the list and click *Disconnect*.

To edit a database: Select the database from the list and click *Edit*. This will activate the The Database Editor [p110]. Section databases can only be edited in the Section Composer [see "Section Composer User's Guide" in *LARSA Section Composer*].

For More Information

- For information on how to create and edit databases, see The Database Editor [p110].

Using the Model Spreadsheets

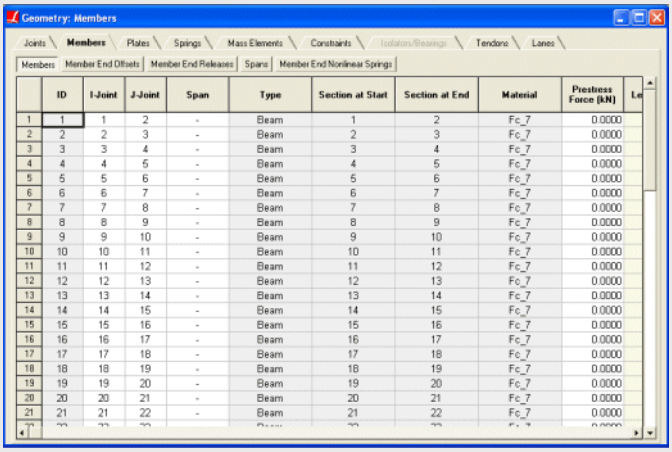
Model data spreadsheets provide direct, numerical access to all of the data used to model a structure.

This tool can be found on the Input Data menu.

The spreadsheets are divided into several categories, listed on the **Input Data** menu. Choose one of the categories to view that category's spreadsheets. (Which categories are available to you depends on your license for LARSA.)

The categories are . . .

- Properties [in *LARSA 2000 Reference*]: Materials, sections, coordinate systems, and other element properties
- Geometry [in *LARSA 2000 Reference*]: Joints, members, plates, and other geometry elements
- Loads [in *LARSA 2000 Reference*]: Load cases, combinations, and stages
- Load spreadsheets can also be accessed from the Load Cases Explorer [p58]
- Construction stages can also be accessed from the Construction Stages Explorer [p64]



	ID	I-Joint	J-Joint	Span	Type	Section at Start	Section at End	Material	Prestress Force (kN)
1	1	1	2	-	Beam	1	2	Fc_7	0.0000
2	2	2	3	-	Beam	2	3	Fc_7	0.0000
3	3	3	4	-	Beam	3	4	Fc_7	0.0000
4	4	4	5	-	Beam	4	5	Fc_7	0.0000
5	5	5	6	-	Beam	5	6	Fc_7	0.0000
6	6	6	7	-	Beam	6	7	Fc_7	0.0000
7	7	7	8	-	Beam	7	8	Fc_7	0.0000
8	8	8	9	-	Beam	8	9	Fc_7	0.0000
9	9	9	10	-	Beam	9	10	Fc_7	0.0000
10	10	10	11	-	Beam	10	11	Fc_7	0.0000
11	11	11	12	-	Beam	11	12	Fc_7	0.0000
12	12	12	13	-	Beam	12	13	Fc_7	0.0000
13	13	13	14	-	Beam	13	14	Fc_7	0.0000
14	14	14	15	-	Beam	14	15	Fc_7	0.0000
15	15	15	16	-	Beam	15	16	Fc_7	0.0000
16	16	16	17	-	Beam	16	17	Fc_7	0.0000
17	17	17	18	-	Beam	17	18	Fc_7	0.0000
18	18	18	19	-	Beam	18	19	Fc_7	0.0000
19	19	19	20	-	Beam	19	20	Fc_7	0.0000
20	20	20	21	-	Beam	20	21	Fc_7	0.0000
21	21	21	22	-	Beam	21	22	Fc_7	0.0000

LARSA's Spreadsheets

At the top of the spreadsheet window, there is a list of tabs for each type of data in the category chosen. For the geometry [see "Geometry" in *LARSA 2000 Reference*] category, shown above, the tabs list joints, members, plates, springs, mass elements, constraints, tendons, and lanes. Click a tab to see that particular type of data.

For information on what each column means in each of the spreadsheets, see Model Data Reference [in *LARSA 2000 Reference*].

Using the Spreadsheet

Each row in the spreadsheet represents one "object" that exists in the project. For the materials spreadsheet, this means that each row represents one available material. For members, each row represents one member in the project. Creating a member graphically will add a row to the spreadsheet. Conversely, deleting a row in the spreadsheet will erase that object from the project. (But, don't fret, there is always Undo/Redo [p75]) This differs from other spreadsheet applications, such as Microsoft Excel, that have a seemingly endless number of rows from the start. To add a row, which adds a new object into the project, see the section on adding rows, below.

One cell is active in a spreadsheet at any given time. The active cell, which is indicated by a black box around the cell, will receive user input. Typing into the spreadsheet will affect the

active cell. Clicking a cell makes it active. The active cell can be moved using the arrow keys. Pressing TAB moves the active cell to the right.

When a spreadsheet is active, typing will replace the contents of the active cell. To edit the active cell's contents, double-click the cell, press ENTER, or press F2 like in Microsoft Excel.

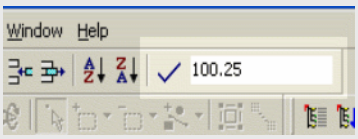
Many cells can be selected at once. To select a block of cells, click-and-drag the mouse from the upper-left cell to be selected to the lower-right cell to be selected. Alternatively, hold the SHIFT key as you reposition the active cell. To select all cells in the spreadsheet, choose *Select All* from the **Edit** menu, or press CTRL+A. Selecting many cells is useful for cut/copy/paste (see below) and editing multiple cells at once.

Simple formulas can be entered into spreadsheet cells, similar to using Microsoft Excel. Unlike in Excel, the formulas are evaluated immediately, and the result of the formula is stored in the spreadsheet, not the formula itself. These formulas are useful for performing quick computations and conversions. To enter a formula, precede it with an equal sign. Formulas cannot reference other cells. Some example formulas are:

=5/2.54 *(converts 5 centimeters to inches)*
=10*sin(.1) *(sin expects an angle in radians)*

To enter more complex formulas or apply formulas to many cells at once, see Spreadsheet Formulas [p113].

Editing Multiple Cells at Once



Spreadsheet Edit Bar

Normally you can only change one cell at a time, but it is possible to change many cells in one go. To do so, select the cells you want to edit together. The cells must form a contiguous block, and they must all be the same type (all numeric, for instance). Then enter the new value for the cells in the Edit Bar in the LARSA toolbar, shown to the right. Finally, click the checkmark to update the cells.

Adding Rows

To add a new row of data to the end of a spreadsheet, choose *Add Row*, which can be found in several locations:

- In the **Edit** menu, choose *Add Row*.
- Click the *Add Row* tool on the toolbar.
- Press the key combination `SHIFT+INS`.

A new row may be added to a spreadsheet by typing directly into the blank row at the end of every spreadsheet or by double-clicking a cell in that row. The blank row will then become promoted to a real row of data.

Inserting Rows

Rows of data can be inserted into the middle of a spreadsheet. Position the active cell *below* where you want the new row to be inserted. Then do one of the following:

- In the **Edit** menu, choose *Insert Row*.
- Click the *Insert Row* tool on the toolbar.
- Press the key combination `CTRL+INS`.
- Right-click the spreadsheet and choose *Insert Rows*.

To insert multiple rows into a spreadsheet, select a block of cells so that the number of rows selected is equal to the number of new rows you want to insert. Choosing *Insert Rows* will insert that number of rows *above* the selection block.

Deleting Rows

To delete rows, select the rows you want to delete, or position the active cell in the row you want to delete. Then do one of the following:

- In the **Edit** menu, choose *Delete Rows*.
- Click the *Delete Rows* tool on the toolbar.
- Press the key combination `SHIFT+DEL`.
- Right-click the spreadsheet and choose *Delete Rows*.

Cut and Copy

To copy cells to the Windows clipboard (from which you can paste the data elsewhere in LARSA or in other Windows applications), select the block of cells to be copied, or position the active cell in the one cell to be copied. Then do one of the following:

- In the **Edit** menu, choose *Copy*.
- Click the *Copy* tool on the toolbar.
- Press the key combination `CTRL+C`.
- Right-click the spreadsheet and choose *Copy*.

Although individual cells can be copied, only entire rows may be "cut" from the spreadsheet and placed into the clipboard. To cut a row or block of rows from the spreadsheet, position the active cell in the one row to be cut, or select the block of rows to be cut. Then do one of the following:

- In the **Edit** menu, choose *Cut*.
- Click the *Cut* tool on the toolbar.
- Press the key combination `CTRL+X`.
- Right-click the spreadsheet and choose *Cut*.

Selecting an entire row is not necessary to cut cells. If a block of cells is selected, all rows that the block intersects will be cut.

Paste

There are two modes in which LARSA will paste data from the Windows clipboard into a spreadsheet: Overwrite mode and Insert mode.

In **Overwrite** mode, the contents of the clipboard will replace the contents of the cells which are selected. If only one cell of data is in the clipboard, LARSA will always overwrite the contents of the active cell with the contents of the clipboard. Otherwise, to paste in overwrite mode, select the block of cells to receive the data from the clipboard.

In **Insert** mode, the contents of the clipboard will be inserted into new rows in the spreadsheet. Multiple cells of data must be in the clipboard. To paste in insert mode, do not select a block of cells. The contents of the clipboard will be inserted above the active cell.

After positioning the active cell and selecting a block of cells, if necessary, do one of the following:

- In the **Edit** menu, choose *Paste*.
- Click the *Paste* tool on the toolbar.
- Press the key combination `CTRL+V`.
- Right-click the spreadsheet and choose *Paste*.

For More Information

- Spreadsheet Formulas [p113].
- Model Data Reference [in *LARSA 2000 Reference*].

Special Data Tools

LARSA 2000 provides several specialized tools for working with materials, sections, and coordinate systems and a special tool for checking for modeling errors.

Working with User Coordinate Systems

User coordinate systems are used to enter and view coordinates in systems other than global coordinates.

UCSs are used in the following ways:

Active Coordinate System

There is always an *active coordinate system*. When LARSA 2000 starts, the active coordinate system is the global coordinate system. Joint coordinates in the spreadsheets [p91] and the Model Data Explorer [p56] are always reported with respect to the active coordinate system. Transformations [p79] are also applied in the active coordinate system, and the graphics window grid [p50] is displayed in the active system.

A user coordinate system can be set as the active coordinate system. This is explained below.

Changing the active coordinate system does not affect the positions of joints in the project. It only affects how joint coordinates are displayed and how modeling tools operate.

Generation

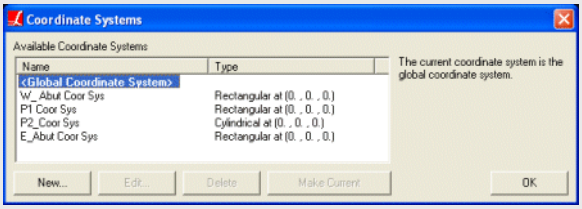
Coordinate systems are used in Generation Tools [p81].

Joint Displacement Coordinate Systems

When joints are restrained, the restraints are applied in the directions of a user coordinate system rather than in global directions. See Joints [in *LARSA 2000 Reference*] for more information.

Coordinate systems are accessed through the Using the Model Spreadsheets [p91] or through the Coordinate Systems dialog box.

This tool can be found on the Input Data menu under "Coordinate Systems."



Working with Coordinate Systems

The dialog box shows all of the coordinate systems in the open project. The *active coordinate system* is listed in bold.

To make a coordinate system the active coordinate system, click the system and then click *Make Current.*

For an explanation of how user coordinate systems are defined, see User Coordinate Systems [in *LARSA 2000 Reference*].

For More Information

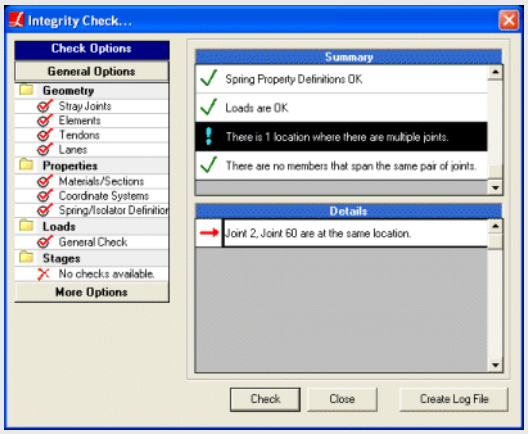
- User Coordinate Systems [in *LARSA 2000 Reference*].

Integrity Check

The Integrity Check command checks a model for common modeling errors.

When one is modeling a structure, it is easy to accidentally enter numbers which are invalid, or forget to enter values where they are required. The Integrity Check scans a model for common, detectable errors, reports the errors to the user and suggests ways to fix the problem.

This tool can be found on the Input Data menu under "Integrity Check."



Integrity Check

Select which integrity checks to run on the model by clicking on each item so that it has a checkmark next to it. By default, all tests are performed. Click *More Options* to see a list of more advanced tests

Click *Check* to scan the model. This process can take several minutes on large models, especially ones with long tendons.

A summary of results for each test will be listed in the upper-right box. Each test will have next to it either a green check indicating the test found no problems or a red exclamation

point indicating errors or warnings were found.

Click an item in the summary list to see details in the lower-right list. Details will report the exact errors that were found.

Integrity Tests

The following checks are performed for each test.

See the Model Data Reference [in *LARSA 2000 Reference*] for more information about correctly entering data.

Stray Joints
Checks for joints that are not connected to any elements.

Elements
Checks that...

- Members, springs, and isolators are assigned a distinct start and end joint.
- Members have non-zero length.
- Members have been given a section and material.
- Member rigid zones are defined from 0 to 1 and do not overlap.
- Plates are either triangular or quadrilateral.
- Plates have been given a material and have non-zero thickness.
- Isolators have been assigned an isolator definition.
- Master joints are not also slaved.
- Slave/Masters and Mass Elements are properly defined.

Tendons
Checks that tendons have a material, a positive strand area, jacking force, and number of strands, and a well-defined path.

Lanes
Checks that lanes have a well-defined path.

Materials/Sections

Checks that materials have a non-zero shear modulus and a Poisson ratio between 0 and 0.5. Also checks that members have unique names and sections have unique names.

User Coordinate Systems

Checks that user coordinate systems are well-defined.

Spring Definitions

Checks that spring curves have at least two points, no point through the origin if it is hysteretic, and have x-values that are only increasing.

Checks hysteretic spring definitions for Beta1, Beta2 between 0 and 1, and Alpha at least 2. Checks that bilinear backbone spring definitions have 4 points on the curve and trilinear 7.

Loads

Checks that loads are properly defined.

Coincident Joints

Checks whether multiple joints are coincident (at the same location).

Overlapping Members

Checks for multiple members that span the same two joints.

Overlapping Plates

Checks for multiple plates that span the same set of joints.

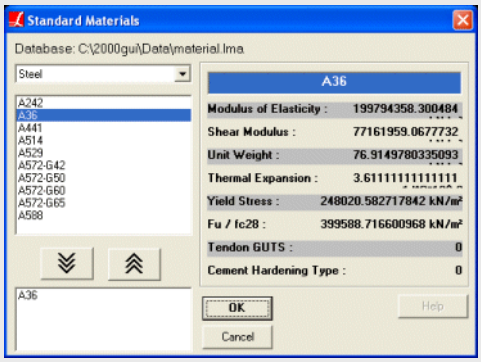
Spans

Checks that spans are properly defined and are collinear.

Loading Standard Materials

Before a member or plate can be given material [see "Materials" in *LARSA 2000 Reference*] properties, those properties must be loaded into LARSA. The properties can be entered manually in the materials spreadsheet [p91], or they can be imported from the standard materials database. The latter method is explained in this section.

This tool can be found on the Input Data menu under "Standard Materials."



Importing Standard Material Properties

Standard materials can be added into the open project by opening the Standard Materials tool.

Materials in the standard database are organized by type: Steel and Concrete. Select the type of material you want to add into the open project.

A list of available materials will be presented in the next box. Click on the material to add to the open project -- it's properties will be shown to the right -- and then click the down arrow to add it to the list of materials to add. Repeat for each material to add.

When all of the materials to add are in the bottom list, click OK.

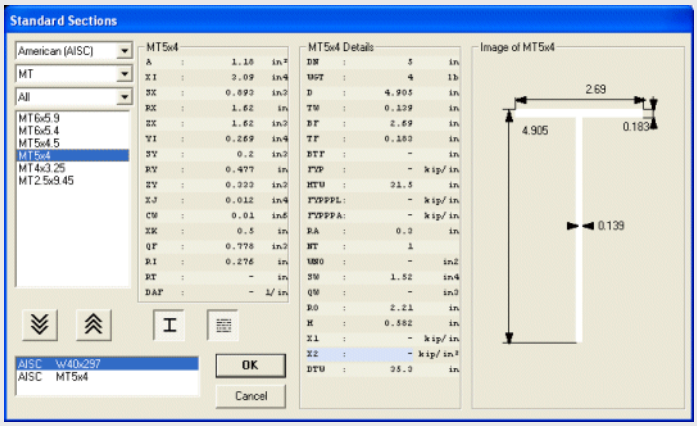
For More Information

- Materials [in *LARSA 2000 Reference*].
- Members [in *LARSA 2000 Reference*].
- Plates [in *LARSA 2000 Reference*].

Loading Standard Sections

Before a member can be given sectional [see "Sections" in *LARSA 2000 Reference*] properties, those properties must be loaded into LARSA. The properties can be entered manually in the sections spreadsheet [p91], by creating a custom section [p107], or they can be imported from the standard sections database. The latter method is explained in this section.

This tool can be found on the Input Data menu under "Standard Sections."



Importing Standard Sections Properties

Standard sections can be added into the open project by opening the Standard Sections tool.

Sections in the standard database are organized by database (AISC, UK, etc.) and type (I, T, C, etc.). They can also be filtered by name. Select these options in the first three drop-down lists.

A list of available sections will be presented in the next box. Click on the section to add to the open project -- its properties will be shown to the right -- and then click the down arrow to add it to the list of sections to add. Repeat for each section to add.

When all of the sections to add are in the bottom list, click OK.

For More Information

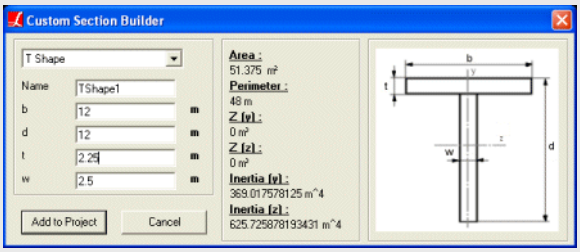
- Sections [in *LARSA 2000 Reference*].
- Members [in *LARSA 2000 Reference*].

Creating Custom Sections

Before a member can be given sectional [see "Sections" in *LARSA 2000 Reference*] properties, those properties must be loaded into LARSA. The properties can be entered manually in the sections spreadsheet [p91], imported from the standard sections database [p105], or created with the Custom Section tool. The latter method is explained in this section.

Use the Custom Section tool when you want to add sectional properties to the open project that are not found in the standard database but are based on standard shapes.

This tool can be found on the Input Data menu under "Custom Section."



Creating a Custom Section

Custom sections can be added into the open project by opening the Custom Section tool.

Section Shape

Select the shape of the section. Available shapes include I Shape, T Shape, and L Shape. The diagram on the right side of the dialog shows the orientation of the axes of the shape, which affect how you will set the member orientation angle [see "Members" in *LARSA 2000 Reference*] later.

Name

Give a name to the section you are creating. The name will be used later when assigning it to members.

Dimensions

A list of dimensions will be presented, appropriate for the section shape that you selected previously. Enter a value for each dimension. The diagram on the right side of the dialog shows the meaning of each dimension.

As you enter the dimensions, the calculated sectional properties will be updated on the right.

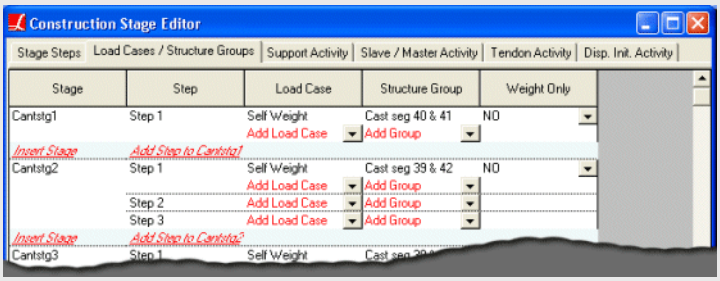
When all of the dimensions have been entered, click OK.

For More Information

- Sections [in *LARSA 2000 Reference*].
- Members [in *LARSA 2000 Reference*].

Construction Stage Editor

The Construction Stages Editor is used for Staged Construction Analysis to define the order in which a structure is built.



The Construction Stages Editor

The editor is explained in detail in the Staged Construction reference [see "Setting Up the Model" in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

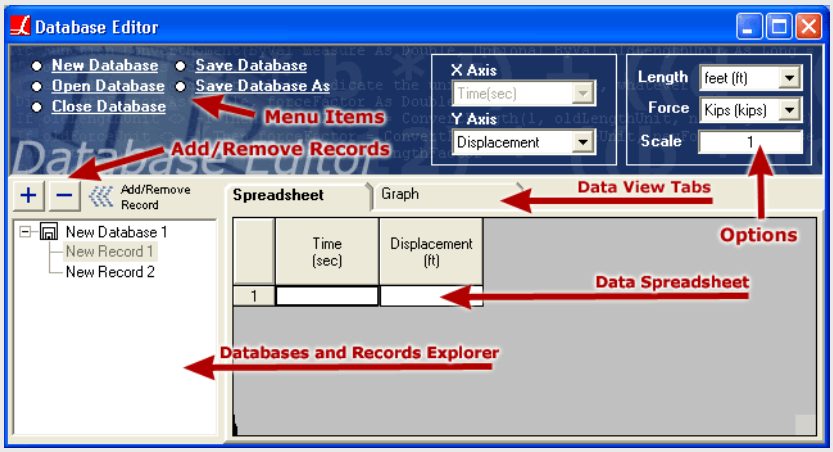
For More Information

- Construction Stages Explorer [p64].
- For an overview of construction analysis, see Staged Construction Guide [in *LARSA 2000/4th Dimension: Staged Construction Analysis*].

The Database Editor

The Database Editor is a tool within LARSA 2000 to create and edit external LARSA database files.

You can activate the *Database Editor* from the **Input Data** menu under **Edit Databases**, or you can select the database in the Connect Databases [p89] dialog box.



Database Editor View

Creating a Database

When you activate *Database Editor*, all databases connected to the active project automatically shows up in *Databases and Records Explorer*. Use the *New Database* link at the top-left side of the *Database Editor* and choose the type of the database. Currently you can create *Moving Load*, *Time History and Response Spectra* databases. Once the new database is created it will show up on the *Databases and Records Explorer* on the left.

Adding/Removing Records

To add a new record, click on the database name on the *Databases and Records Explorer* and click the button with the plus sign.

To remove an existing record, click on the record on the *Databases and Records Explorer* and click the button with the minus sign.

Adding/Editing/Removing Data

The data spreadsheet should be used to modify the records in the database. Click on the record you would like to edit from the list on the left. The spreadsheet tab will activate. Use this spreadsheet to insert/remove/edit data. Since this spreadsheet is like any other spreadsheet in LARSA 2000, you can use almost all menu items to format the spreadsheet. See Using the Model Spreadsheets [p91].

Other Data Views

Other than the spreadsheet view, each database may have different types of view options. For example *Moving Load Database* records can be edited on the spreadsheet and then can be viewed visually on the next tab called *Graphics*. This allows users to easily verify their data. For *Time History Curve* and *Response Spectra Curve* databases, *Database Editor* will give you the option to view a graph of the data.

Additional Options

Depending on the database type, every record may have some additional options other then the actual data. For example for a *Response Spectra Curve* database you can choose to input time vs displacement or frequency vs displacement.

It is not recommended to edit the records that are currently in use by the project.

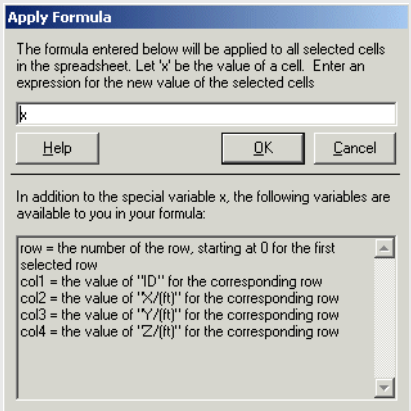
For More Information

- For information on how to connect databases to a project, see Connecting Databases [p89].

Spreadsheet Formulas

The Apply Formula tool allows the user to enter formulas to compute the value of spreadsheet cells.

Formulas are entered in the Apply Format dialog box, which is accessible by right-clicking the input spreadsheets [p91]. The formula will be used to find the values of the selected cells in the spreadsheet.



Apply Formula

Unlike the formulas that can be entered directly into spreadsheet cells, formulas entered this way can refer to other cells in the spreadsheet and can make use of each cell's row index and current value.

Formula

Enter a formula that will evaluate to the desired value for each cell. The formula can make use of a small set of variables, which are defined below. The special variable *x* represents the current value of any given cell. Thus, the default formula *x* will replace the value of each selected cell in the spreadsheet with the value currently in the spreadsheet: no change will occur. Some simple example formulas are:

5/2.54 (*converts 5 centimeters to inches*)
x (*leaves spreadsheet cells unchanged*)
2*x (*multiplies each spreadsheet cell by 2*)

-x (*changes the sign of the selected cells*)

Variables

The variables available to formulas include:

x
The current value of a cell.

row
The row number of a cell, starting at 0 for the first row in the selected block of cells.

col#
The value in the other columns of the spreadsheet, in the same row as the cell being updated. A list of col# variables appears in the Apply Formula dialog box.

Functions

The following functions are available to formulas:

- abs, sgn, fix (round toward zero)
- sqr, exp, log
- sin, cos, tan, atn (uses radians)

Some further examples follow.

row (*would set a single column of cells to the consecutive numbers starting at 0*)
2*col2 (*in the Joints spreadsheet, would set the selected cells to have twice the value of the x-coordinate value in the corresponding row*)
col2+col3 (*in the Joints spreadsheet, would set the selected cells to have the sum of the x- and y-coordinate values in the corresponding row*)
10+(25-10)*(col4-5)/(15-5) (*in the Joints spreadsheet, if the z-coordinates in the selected rows range from 5 to 15, then this function sets the values of the selected cells to a linear interpolation between 10 and 25 at the position of the corresponding z-coordinate.*)

For More Information

- Using the Model Spreadsheets [p91].

Running an Analysis

Running an analysis, the core feature of LARSA 2000, takes the structural model you have created and subjects it to various types of analyses. For more detailed information on how analyses are performed, see the Analysis Reference [in *LARSA 2000 Reference*].

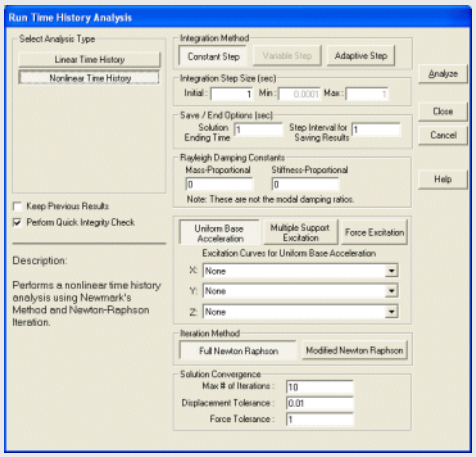
Selecting an Analysis

Analyses are grouped by type in the **Analysis** menu as follows:

- Linear Static [see "Linear Static Analysis" in *LARSA 2000 Reference*] and P-Delta [see "P-Delta Analysis" in *LARSA 2000 Reference*] Analyses
- Nonlinear Static Analysis [see "Nonlinear Static Analysis" in *LARSA 2000 Reference*]
- Nonlinear Plastic Pushover and Collapse
- Eigenvalue [see "Eigenvalue and Stressed Eigenvalue Analysis" in *LARSA 2000 Reference*] and Response Spectra [see "Response Spectra Analysis" in *LARSA 2000 Reference*] Analyses
- Time-History (linear/nonlinear) Analysis [see "Linear Time History Analysis" in *LARSA 2000 Reference*]
- Moving Load Analysis [see "Moving Load Analysis" in *LARSA 2000 Reference*]
- Staged Construction Analysis

A red arrow next to an analysis group indicates which type of analysis was last run for the open project. If no analysis has been run, the arrow points to the first group.

Selecting one of the analysis groups brings up the analysis dialog box.



Run an Analysis

Choosing Analysis Options

Each type of analysis within the chosen group will be listed on the left in the *Select Analysis Type* box. Click the analysis type desired.

All of the analysis options for the selected type of analysis will be listed on the right. Some analysis types have no options, while others have many. For descriptions of all analysis options, see the Analysis Reference [in *LARSA 2000 Reference*].

All analysis types share several common options:

Keep Previous Results

When checked, the LARSA analysis engine will try to hold on to the results of previous analyses when the next analysis is performed. In some cases, old results cannot be retained.

Perform Quick Integrity Check

Often problems in an analysis are a result of simple mistakes in the model. When this option is checked, a fast integrity check of the model is performed before the next analysis is run. The check is able to find common mistakes. It is highly recommended that you leave this option checked.

Running the Analysis

To run the analysis based on the options you have chosen, click *Analyze*. The analysis window will appear, showing the progress of the analysis as it happens. Analysis errors will also be shown in the window.

Click *Close* to save your analysis options but not perform an analysis.

Click *Cancel* to discard changes to the analysis options and abort running an analysis.

For More Information

- Analysis Reference [in *LARSA 2000 Reference*].

Getting Results

The sections below describe how to access results graphically and numerically.

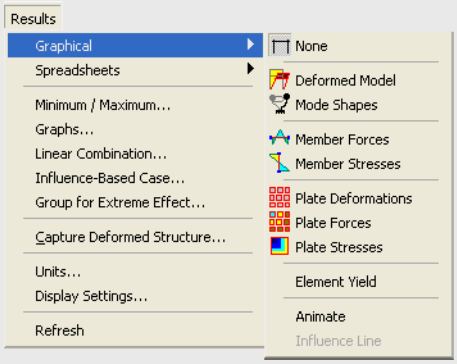
See Analysis Results Reference [in *LARSA 2000 Reference*] for a detailed description of each result type available in LARSA.

For More Information

- Analysis Results Reference [in *LARSA 2000 Reference*].

Viewing Results Graphically

Most of the results computed by the analysis engine can be viewed graphically, including deformed structure, mode shapes, member forces/stress, and plate forces/stresses.



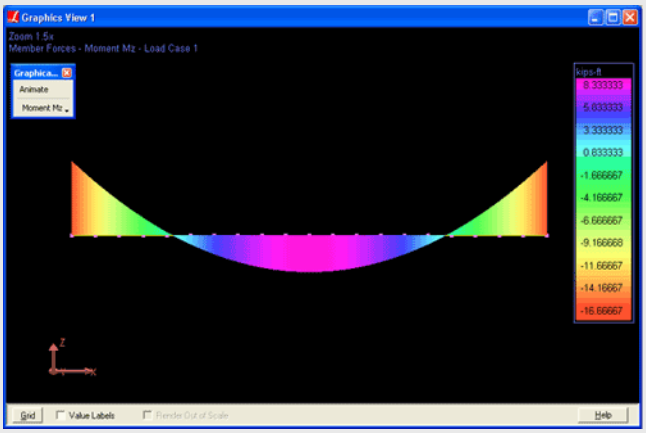
Choosing Graphical Results

Results can be viewed graphically by choosing an item from the graphical results menu or by clicking a graphical result icon in the toolbar. The icons on the toolbar are the same as those on the menu shown here.

This tool can be found on the Results menu under "Graphical."

Graphical results are shown for the active result case or are an envelope of the selected result cases. To activate and select result cases, use the Analysis Results Explorer [p66]. If the Explorer is not visible, activate it by selecting **View > Analysis Results**.

Select the analysis result cases from the Explorer. You can click on each result case to select them individually. However, if you would like to view the envelope of multiple result cases, you can select multiple result cases by holding the `CTRL` key, or you can select a range of result cases by holding the `SHIFT` key. Clicking on a result group while holding the `CTRL` key will select or unselect all the result cases in that group.



Member Forces Shown Graphically

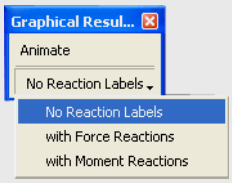
Animating Results, Value Labels

Animating Results: All result types can be animated using the *Animate* button on the *Graphics Results Options* toolbox or using the **Results** > **Graphical** > **Animate** menu. Animation works by viewing each result case back to back in a specific result group. Please note that this does not apply to mode shapes, as each mode shape is animated individually.

Value Labels: To label points on member forces/stresses diagrams, check the *Value Labels* checkbox at the bottom of the graphics window.

Types of Graphical Results

Deformed Model



Deformed Model Options

To view the deformed model, select **Results > Graphical > Deformed Model**.

The *Graphics Results Options* toolbox will pop up (shown to the right). You can view model deformation for the selected result case with *Force Reactions*, *Moment Reactions* or without any reaction labels.

The scale factor of the deformed model is set in the Analysis Results Explorer [p66] or in Graphical Results Options [p143]. In Graphical Results Options [p143], you can also choose to display the undeformed structure in a ghosted mode and whether to display member span deflections (which can be very slow for large models).

See: Joint Displacements [in *LARSA 2000 Reference*], Joint Reactions [in *LARSA 2000 Reference*] (for reaction labels), Member Displacements [in *LARSA 2000 Reference*] (for span deflection)

Mode Shapes

To view mode shapes, select **Results > Graphical > Mode Shapes**.

Setting Scale Factor

The scale factor of the mode shapes is set in the Analysis Results Explorer [p66] or in Graphical Results Options [p143]. In Graphical Results Options [p143], you can also

choose to display the undeformed structure in a ghosted mode.

See: Mode Shapes [in *LARSA 2000 Reference*]

Member Forces

To view member forces, select **Results > Graphical > Member Forces**.

The *Graphics Results Options* toolbox will allow you to select which direction of forces to view: Axial Fx, Shear Fy, Shear Fz, Torsion Mx, Moment My, Moment Mz.

The diagram size is set in the Analysis Results Explorer [p66] or in Graphical Results Options [p143]. Also in Graphical Results Options [p143], you can choose whether to display the diagram as a line diagram or with solid shading, whether to invert the diagrams, and the number of stations along the members to report forces at.

See: Member Sectional Forces [in *LARSA 2000 Reference*]

Member Stresses

To view member stresses, select **Results > Graphical > Member Stresses**.

The *Graphics Results Options* toolbox will allow you to select the location of the stress to view: at the centroid (P/A), at Stress Point A, at Stress Point B, at Stress Point C, at Stress Point D, the Shear Stress in Y, the Shear Stress in Z, the Maximum Absolute Stress, the Maximum Tensile Stress, and the Maximum Compressive Stress.

The diagram size is set in the Analysis Results Explorer [p66] or in Graphical Results Options [p143]. Also in Graphical Results Options [p143], you can choose whether to display the diagram as a line diagram or with solid shading, whether to invert the diagrams, and the number of stations along the members to report forces at.

See: Member Stresses [in *LARSA 2000 Reference*]

Plate Deformation

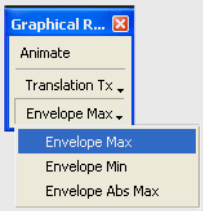


Plate Deformation Options

To view plate deformations, select **Results > Graphical > Plate Deformations**.

The *Graphics Results Options* toolbox (shown to the right) will allow you to select the direction and the envelope type from the option lists.

Plate deformations are displayed as a contour diagram and are based only on the displacements of the joints.

Contour diagram must be selected in Graphical Results Options [p143]. There, you can choose whether to display the contour diagram on the deformed position of the plates and how the diagram should be displayed (lines only, solid shading, or smooth shading).

See: Joint Displacements [in *LARSA 2000 Reference*]

Plate Forces

To view plate forces, select **Results > Graphical > Plate Forces**.

The *Graphics Results Options* toolbox will allow you to select the direction and the envelope type from the option lists.

Choose the display mode in Graphical Results Options [p143]. There, you can choose whether to display forces as a contour diagram or by shading each plate with a color representing the forces on the plate. You can also choose whether to display the diagram on the deformed position of the plates and how a contour diagram should be displayed (lines only, solid shading, or smooth shading).

See: Plate Forces at Joints [in *LARSA 2000 Reference*]

Plate Stresses

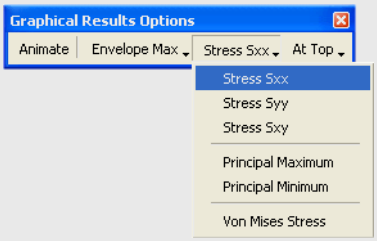


Plate Stresses Options

To view plate stresses, select **Results > Graphical > Plate Stresses**.

The *Graphics Results Options* toolbox will allow you to select the direction, the envelope type and the stress location.

Stress trajectories can also be viewed on each plate. Each stress trajectory arrow indicates the direction of the maximum principal stress in each plate. A small perpendicular line is also drawn on each plate to show the direction of the minimum principal stress. Stress trajectory icons can be turned on and off in Graphical Results Options [p143].

Choose the display mode in Graphical Results Options [p143]. There, you can choose whether to display forces as a contour diagram or by shading each plate with a color representing the forces on the plate. You can also choose whether to display the diagram on the deformed position of the plates and how a contour diagram should be displayed (lines only, solid shading, or smooth shading).

See: Plate Stresses [in *LARSA 2000 Reference*]

Element Yield

To view element yield, choose **Results > Graphical > Deformed Model**.

The *Graphics Results Options* toolbox will allow you to select the whether the result will be viewed with icons for plastic hinges.

See: Member Yield and Strains [in *LARSA 2000 Reference*]

For More Information

- Graphical Results Options [p143].
- Results Spreadsheets [p127].
- Analysis Results Reference [in *LARSA 2000 Reference*].

Results Spreadsheets

All analytical results can be viewed in numerical form through LARSA's results spreadsheets.

This tool can be found on the Results menu under "Spreadsheets."

A toolbar for quick access to the results spreadsheet can be activated from **View** > **Toolbars** > **Numerical Results** menu.

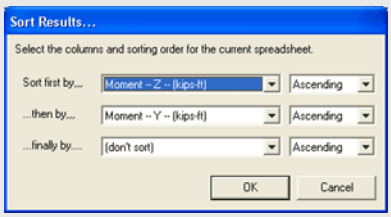
Results Data Toolbar

Selecting Result Cases

Results are reported for **selected** result cases. Result cases are selected in the Analysis Results Explorer [p66].

Click on a result case to select it individually. However, if you would like to view the envelope of multiple result cases, select multiple result cases by holding the `CTRL` key, or you can select a range of result cases by holding the `SHIFT` key. Clicking on the result group while holding the `CTRL` key will select/unselect all the result cases under that group.

Sorting



Mode Shape Options

Result spreadsheets can be sorted according to the values in one, two, or three columns.

This tool can be found on the Edit menu under "Sort."

Select columns on which to sort and the sorting direction. Then click OK.

If *Torsion X*, *Moment Y*, and *Moment Z* are chosen as the sorting columns (in that order), then:

- The rows in the spreadsheet are first sorted according to their torsion values.
- If two or more rows in the spreadsheet have the same torsion values, then those rows are sorted according to their moment y values.
- If in those rows two or more rows have the same moment y values, then those rows are sorted according to their moment z values.

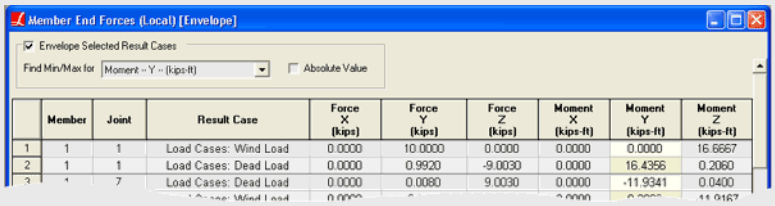
Enveloping

All result spreadsheets can be enveloped by checking the *Envelope Selected Result Cases* check box above the spreadsheet. A select box will appear to let you specify the column to envelope for. Select the column from the select box.

The spreadsheet will show the result data enveloped over that column. The enveloped spreadsheet rows are colored to differentiate between minimum and maximum. Gray rows always represents the maximum.

Normally, LARSA will present the result case with the most negative value in the minimum rows and the result case with the most positive value in the maximum rows. If no result case had a negative value in the enveloped column, then *no data* will be displayed in the minimum row. Similarly, if no result case had a positive value in the enveloped column, then *no data* will be displayed in the maximum row.

LARSA can also envelope using absolute values. In this case, LARSA will report the result case with the value smallest in magnitude (ignoring sign) in the minimum row and the case with the value largest in magnitude in the maximum row.

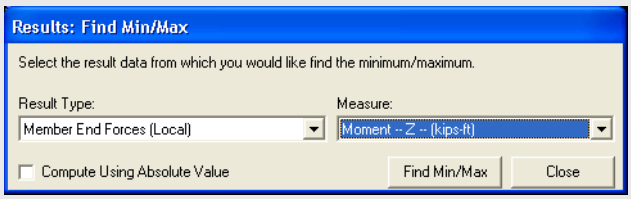


	Member	Joint	Result Case	Force X (kips)	Force Y (kips)	Force Z (kips)	Moment X (kips-ft)	Moment Y (kips-ft)	Moment Z (kips-ft)
1	1	1	Load Cases: Wind Load	0.0000	10.0000	0.0000	0.0000	0.0000	16.6667
2	1	1	Load Cases: Dead Load	0.0000	0.9920	-9.0030	0.0000	16.4356	0.2060
3	1	7	Load Cases: Dead Load	0.0000	0.0080	9.0030	0.0000	-11.9341	0.0400

Enveloping Spreadsheet Results

Finding Minimums and Maximums

Select **Minimum/Maximum** under **Results** menu. Select the result type and the column to find the minimum and maximum values. Users also have the option to compute using the absolute value. Click Find Min/Max to get the minimum and maximum values for the specified column. The dialog box will expand to show the results in spreadsheet format.



Mode Shape Options

Incremental and Class-Based Results

In Staged Construction Analysis, result cases always report the cumulative effects of loads and other construction activities on the structure. Check the *Incremental* check box at the top of the results spreadsheets to see just the incremental effects of the selected construction step. For most results, incremental effects are computed by subtracting the results of the previous construction step from the results of the selected construction step.

Sometimes it is important to see the cumulative effects of just post-tensioning or other types of loads. Even if a structure has been subjected to different types of loads, partial cumulative effects can be reported for particular load classes. If construction steps have load classes

associated with them, then a drop-down list of used load classes appears at the top of results spreadsheets. Choose a load class to see the partial cumulative effects of those loads.

Load classes are assigned to load cases [see "Static Load Cases" in *LARSA 2000 Reference*] through the Load Cases Explorer [p58], and they are automatically assigned by the analysis engine to generated result cases when there are time-dependent material effects.

Other Options

Some features associated with result spreadsheets can be accessed under **Edit** menu or by right-clicking the spreadsheet.

- **Sum Selected Cells:** Show the summation of the values of the selected cells.
- **Create Input Load Case:** Creates a input load combination based on result case that resulted in the row under the cursor.
- **Create Result Case:** Creates a linear result combination based on the result case that resulted in the row under the cursor.

For More Information

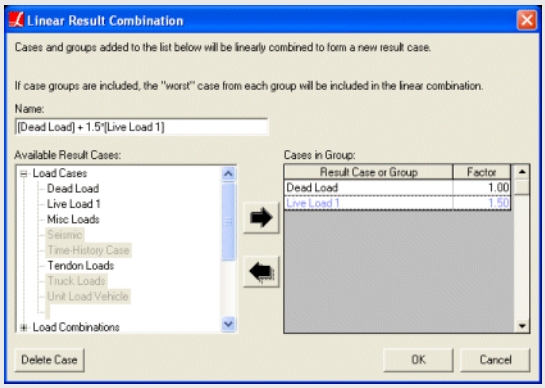
- Viewing Results Graphically [p120].
- Analysis Results Reference [in *LARSA 2000 Reference*].

Linear Result Combinations

Linear Result Combinations provide the ability to dynamically combine multiple result cases. Whereas load combinations combine load cases defined before an analysis, linear result combinations combine result cases after an analysis without the need to reanalyze the structure.

Results for these combinations are generated only when they are needed, so creating and editing linear results combinations are fast operations. In addition, results for linear result combinations are discarded immediately after they are used, so linear combinations will perform slower than pre-analyzed results, but no extra analysis time or hard drive space is needed to form the combinations.

This tool can be found on the Results menu under "Linear Combinations."



Linear Result Combinations

How They Work

Linear result combinations operate by summing together the results of multiple cases. Factors may be applied to the cases to achieve a weighted linear combination.

Any type of result case may be used in a linear result combination. Two dead load cases may be combined together, for instance. Or, one step in a time history analysis can be combined with one position of a load pattern from a moving load analysis.

Linear result combinations have several additional uses for influence-based results. Because an influence-based result case has the effect of only one lane's loading, linear result combinations can be used to combine multiple influence-based cases. The result - a linear combination of linear combinations of influence line data - represents the combined worst-case effect of multiple lane loadings. Furthermore, influence-based cases can be combined with static or live load cases for use with design codes.

Using Linear Result Combinations

To create a linear result combination, select the result cases to be combined from the list on the left and click the right-directed arrow to add them into the list on the right. Enter any factors, if needed, in the list on the right.

Combine for Extreme Effects

> Check the box *Combine for Extreme Effects* to linearly combine the result cases for extreme effects. When this box is checked **and** you are viewing enveloped results, then only cases which contribute negatively to a result are included in the combination when reporting minimum value of the result, and only cases which contribute positively to a result are included in the combination when reporting maximum value of the result.
>
> This option has the effect of reporting the **most extreme** combination of any of the result cases being linearly combined, which may not involve all of the cases in the combination. If this option is not checked, then all cases in the combination are always added together.

Click OK to create the new case. The new case will add together each of the result cases in the list on the right. The new case will appear in the Analysis Results Explorer [p66].

A name for new linear result combination case will be supplied automatically, but the name may be changed if needed.

If a result group (e.g. "Load Combinations", "Moving Load 1") is added into the list on the right, then the worst case in that group will be used in the linear combination. The worst case is chosen each time a result is requested such that LARSA always reports the most extreme

value.

To edit a linear result combination after it has been created, find the combination in the Analysis Results Explorer, right-click it, and then click *Properties*.

To delete a linear result combination, find the combination in the Analysis Results Explorer, right-click it, and then click *Delete*.

For More Information

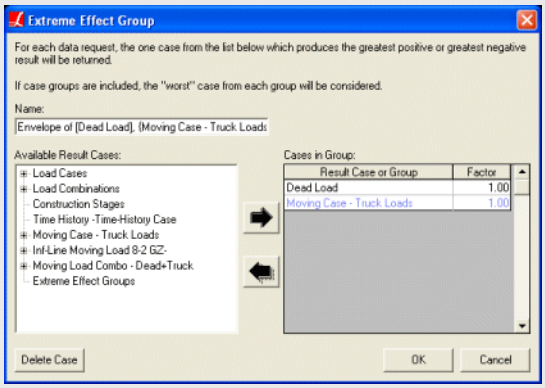
- Extreme Effect Groups [p134].

Extreme Effect Groups

Extreme effect groups represent the worst-case scenario from a set of result cases, similar to envelopes.

For each result data request, an extreme effect group returns the data from the one case in the group that had the most negative (when a minimum is being reported) or positive (when a maximum is being reported) value. Extreme effect groups can only be used when enveloping results.

This tool can be found on the Results menu under "Group for Extreme Effects."



Extreme Effect Groups

How They Work

Extreme effect groups can also be thought of as a "pick one," because only the data from the case with the most extreme values are shown for any given result value.

The most extreme case returned for joint 1's x-reaction may not be the same as the extreme case returned for joint 2's x-reaction. Similarly, each station along a member may come from a different result case. Each row in the spreadsheet comes from only one result case. If an

envelope is requested for y-moment, then the results will come from the result case with the minimum and maximum extreme values in y-moment. The other columns in the spreadsheet, such as axial force, will not be extremes -- instead they will correspond to the result cases that had the min/max y-moment for that row.

Extreme effect groups will never return a maximum that is negative or a minimum that is positive. This follows the definition of extreme effects used in influence-based results and differentiates extreme effect groups from simple envelopes.

Extreme effect groups can only be used when enveloping result data. As with the influence-based results, not all graphical results are enveloped and so they cannot be used to display extreme effect group results.

Although envelopes of result data can be quickly found by selecting multiple cases in the Analysis Results Explorer [p66], the list of cases which make up these on-the-fly envelopes cannot be saved for later use. Furthermore, the data from those envelopes cannot be used in linear result combinations. Extreme effect groups achieve a similar result but do not have these limitations.

Using Extreme Effect Groups

Creating Extreme Effect Groups is done similarly to creating Linear Result Combinations [p131].

To edit an Extreme Effect Group after it has been created, find the combination in the Analysis Results Explorer, right-click it, and then click *Properties*.

To delete an Extreme Effect Group, find the combination in the Analysis Results Explorer, right-click it, and then click *Delete*.

For More Information

- Linear Result Combinations [p131].

Graphing Results

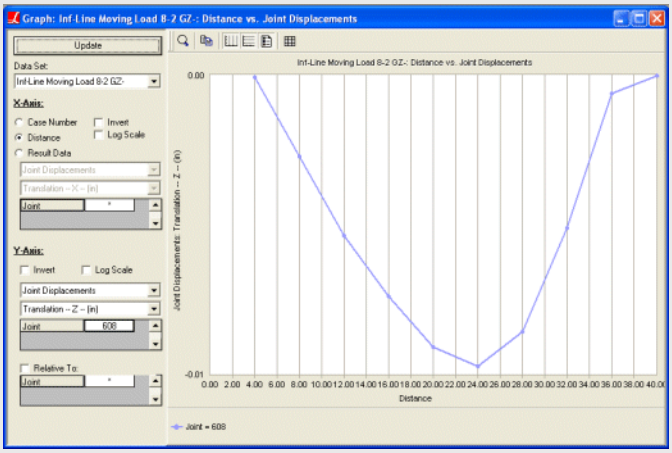
Analysis results can be graphed using LARSA's graph feature. Result versus case, result versus time, and result versus result are all supported.

Graphs are an easy way to see result data graphically. Graphs can be used to show

- The displacement of a joint over time.
- The moment on a member as a truck passes over it.
- The reaction of a joint versus its rotation, over time.
- The stress in a member versus the displacement of its i-joint, over time.
- The difference between joint A's displacement and joint B's displacement versus the forces on joint A, over time.

But LARSA 2000's graphs are not limited to only these few possibilities.

This tool can be found on the Results menu under "Graph."



Graphing Results

Overview

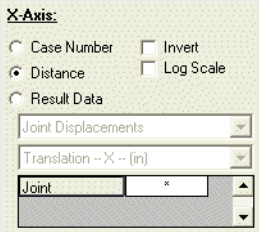
Data Set

Select from what result data to plot the graph. Graphs can be plotted over the steps in moving-load and influence-line cases, time-history cases, and construction stages. A data point will be plotted for each case in the data set.

The x-axis and y-axis of your graph are set up independently. Each axis's parameters are explained below.

Click the *Update* button at the top of the window to update the graph based on your choices.

X-Axis



X-Axis Parameters

Choose what will be plotted on the x-axis of the graph.

Case Number

The ordinal number of the result case will be plotted on the x-axis. This is generally acceptable when cases are evenly spaced, such as in a moving-load or time-history analysis.

Distance

The distance of the load patterns applied to the structure from the start of the lane is plotted on the x-axis. This is only available if the dataset is a moving-load or influence-line case [see "Moving Load Analysis" in *LARSA 2000 Reference*].

Time

The time of the case is plotted on the x-axis. This is only available if the dataset is a time-history case [see "Linear Time History Analysis" in *LARSA 2000 Reference*].

Day

The day number of the case is plotted on the x-axis. This is only available if the dataset is a staged-construction case.

Result Data

A particular result data from the case will be plotted on the x-axis.

Result Type

Select the type of result from the first selection box, such as *Member End Forces*, and then select the particular result in the second selection box, such as *Force -- Z -- At Start Joint*.

Data Selection

Then, fill in all of the fields in the next box. If you chose *Joint Displacements / Translation -- X*, then the next box will display *Joint*, indicating you need to enter a joint number. The x-translation displacements of that joint will be plotted on the x-axis for each case in the data set.

All fields must be filled in.

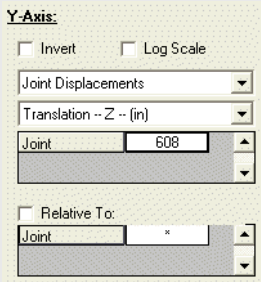
Invert

Displays the graph mirrored horizontally.

Log Scale

Displays the x-axis on a logarithmic scale.

Y-Axis



Y-Axis Parameters

Choose what will be plotted on the y-axis of the graph. The y-axis is set up similarly to the x-axis, except that only result data can be plotted on the y-axis.

Invert

Displays the graph flipped vertically.

Log Scale

Displays the y-axis on a logarithmic scale.

Result Type

Select the type of result from the first selection box, such as *Member End Forces*, and then select the particular result in the second selection box, such as *Force -- Z -- At Start Joint*.

Data Selection

Then, fill in the fields in the next box. If you chose *Joint Displacements* / *Translation -- X*, then the next box will display *Joint*, indicating you need to enter a joint number. The x-translation displacements of that joint will be plotted on the y-axis for each case in the data set.

Unlike for the x-axis, you do not need to fill in all of the fields in this box. If a field is left with an asterisk, then multiple series will be plotted on the graph, one for each possible value of the field. For instance, if you are plotting member sectional forces, if you enter a member number but leave out a station number, then one series will be

plotted for each station on the member. (The number of computed stations is set up in Graphical Results Options [p143].)

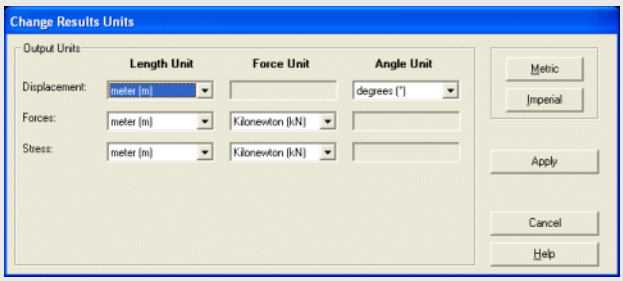
Relative To

Check this box to plot values on the y-axis relative to another value. For instance, you may want to plot the x-translation displacement of joint 1 relative to the same type of displacement of joint 2. (That is, the first value minus the second value.) Fill in all of the fields in this box to choose what to plot the values relative to. The same result type as selected above applies to these values.

Results Units

Results in LARSA are displayed according to the results units selected for the project. The results units used by the open project can be edited using the Results Units dialog.

This tool can be found on the Results menu under "Units."



Changing Results Units

The Results Units dialog only applies to numerical results. Model units are selected in the model units dialog [p86].

The Results Units dialog is arranged into a simple matrix. The rows represent categories of units:

Displacement
Used for joint displacements and related units.

Forces
Used for units that are related to forces.

Stress
Used for units that are related to stresses.

The units for each category can be independently changed so that, for instance, joint displacements need not be in the same length units as member section stresses.

The columns of the matrix represent unit types: Length, Force, and Angle. The unit in the cell in the Force column and Stress row will be used in units that indicate stress.

A unit can be selected for most cells in the matrix. Some cells are gray, such as Displacement/Force, for which unit selection is not applicable.

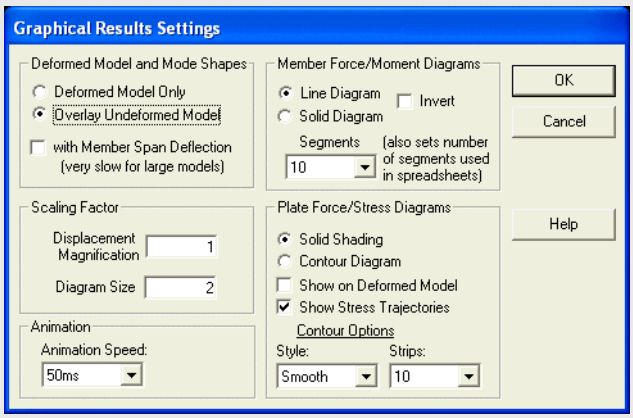
Click the *Metric* or *Imperial* buttons to immediately change all units to the standard metric or imperial units.

Click *Apply* to update the units used to display results. No immediate unit conversion is performed.

Graphical Results Options

The graphical results display settings control how graphical results are displayed. Display settings can be access by right-clicking the graphics window.

This tool can be found on the Results menu under "Display Settings."



Graphical Results Settings

Deformed Model and Mode Shapes

Deformed Model Only

Indicates only the deformed position of the structure will be displayed.

Overlay Undeformed Model

Indicates that in addition to the deformed position of the structure, the original position will be displayed ghosted.

with Member Span Deflections

Indicates that the span deflection of each member will be displayed. For large models, this process can be very slow. It is recommended that if the graphics drawing time is too

much, you should uncheck this option. With this option unchecked, members are drawn as straight lines between the deformed positions of the joints.

Scaling Factor

Displacement Magnification

Specifies the magnification factor of the joint displacements used to display deformed model graphically.

Diagram Size

A number between 1 and 20, specifies the size of moment and stress diagrams on the screen. A value of 1 indicates diagrams should be drawn very small, and 20 indicates diagrams should be drawn very large.

Animations

Animation Speed

Determines the rate at which animations proceed. A value of 500ms indicates animations will update every 500 milliseconds, or twice every second.

Member Force/Moment Diagrams

Line Diagram

indicates force/moment diagrams are displayed as line diagrams, which sometimes make it easier to read the value labels.

Solid Diagram

Indicates force/moment diagrams are to be filled in according to the color legend on the right-hand side of the graphics window.

Segments

Indicates how many segments to divide each member into to display the forces/moments. A value of 10 is usually sufficient, for which 11 data points are drawn. To see a smoother diagram, increase the number of stations. A greater number of stations means the results will take longer to draw. This option also controls the number of segments displayed in the spreadsheets.

Plate Force/Stress Diagrams

Solid Shading

Indicates each plate will be filled with the color that corresponds to the average force or stress acting on it.

Contour Diagram

Indicates a contour diagram will be drawn over the surface of the plates, showing the contours of the forces or stresses on the plates.

Show on Deformed Model

Indicates the diagram will be displayed on plates in their deformed position. Otherwise, the plates are drawn in their original position.

Show Stress Trajectories

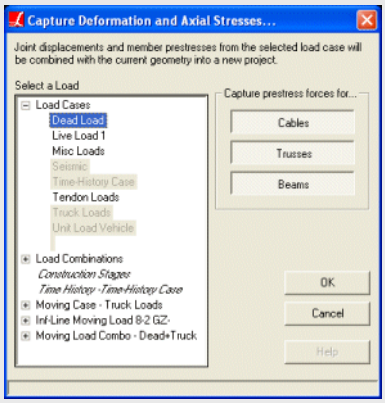
Indicates stress trajectories will be drawn over each plate. Each stress trajectory arrow indicates the direction of the maximum principal stress in each plate. A small perpendicular line is also drawn on each plate to show the direction of the minimum principal stress.

Contour Options

Specifies how a contour diagram of plate forces or stresses is to be displayed. For *Style*: *Lines* indicate only contour lines will be drawn, *Solid* indicates each contour "bar" will be solid-shaded, and *Smooth* indicates full-color shading will be used to show the forces or stresses. *Strips* specifies the number of contour lines/strips to draw. More lines means the diagram will be drawn slower, but more precisely.

Capture Deformed Structure

Creates a new project from the deformed geometry of the open project.



Capture Deformed Structure

The Capture Deformed Geometry tool is used to take the deformation and prestress results of an analysis and create a new project with joints initially at those deformed coordinates and members initially with those prestresses.

To create a new project based on the deformed model, open the Capture Deformed Structure tool.

This tool can be found on the Results menu under "Capture Deformed Structure."

Result Case

Select the result case from the list on the left from which you want to take the deformations and prestresses.

Member Types

Select which types of members for which prestresses should be captures: Cables,

Trusses, and Beams. All three member types are on by default. Click a member type to not capture those prestresses.

Click OK to create the new project. The open project will be closed. You will be prompted to save your changes if the project has been modified since it was last saved.

Automatic Code-Based Load Combinations

The Auto Load Combinations tool is used to create input load combinations [p58] and linear result combinations [p131] according to standard design codes and user defined codes.

This tool can be found on the Input Data/Results menu under "Auto Load Combinations."

Five standard design codes are included with the Auto Load Combinations tool:

- AASHTO-LFD Load Factor Design: TABLE 3.22.1A from Standard Specifications For Highway Bridges, Sixteenth Edition.
- AASHTO-LFD Service Load: TABLE 3.22.1A from Standard Specifications For Highway Bridges, Sixteenth Edition.
- AASHTO-LRFD: TABLE 3.4.1-1 - Load Combinations and Load Factors from AASHTO LRFD Bridge Design Specificatios, Second Edition 1998.
- LRFD 2001: ASCE 7 from Load and Resistance Factor Design, Third Edition.
- Canadian Code: Clause 7, Load and Safety Criterion, Handbook of Steel Construction.

Users can also enter and save their own codes.

A code consists of a set of combinations. Each combination is of a form similar to:

```
1.2 * (1*Dead + 1.5*Live + 2*Seismic + . . .)
```

The Auto Load Combinations tool selects existing load cases or result cases in the open project and combines them based on their Load Class, according to the chosen code.

Some codes have factors that are chosen by the engineer. The Auto Load Combinations tool calls these factors Code Variables, and they can be set by the user.

Example Use

Take the following setup as an exmaple:

A code specifies these combinations:

Code Combination 1: Dead Load + Live Load

Code Combination 2: Dead Load + Wind Load

Code Combination 3: Live Load + Wind Load

And the user selects to combine these load cases:

Load Case 1 (Load Class = Dead Load)

Load Case 2 (Load Class = Live Load)

Load Case 3 (Load Class = Wind Load)

The Auto Load Combination tool will produce these load combinations:

Load Combination 1: Load Case 1 + Load Case 2

Load Combination 2: Load Case 1 + Load Case 3

Load Combination 3: Load Case 2 + Load Case 3

Simple Combinations versus Permutations

If in the above example, the user selected two load cases both with Load Class set to Dead Load, the Auto Load Combinations tool can take one of two routes.

In the simple combination method, code combinations involving dead load will include both dead load cases. For the above example, if the second dead load case is Load Case 4, then the Auto Load Combinations tool will create these combinations:

Load Combination 1: Load Case 1 + Load Case 4 + Load Case 2

Load Combination 2: Load Case 1 + Load Case 4 + Load Case 3

Load Combination 3: Load Case 2 + Load Case 3

However, the user might instead want only one dead load case to be in each combination. Using the permutation method, the Auto Load Combinations tool will create load combinations for all of the possible choices of load cases to use for each combination. For the above example, it will produce:

Load Combination 1A: Load Case 1 + Load Case 2

Load Combination 1B: Load Case 4 + Load Case 2

Load Combination 2A: Load Case 1 + Load Case 3

Load Combination 2B: Load Case 4 + Load Case 3

Load Combination 3: Load Case 2 + Load Case 3

Similarly, if there is additionally a second live load case Load Case 5, then these load combinations will be produced:

Load Combination 1A: Load Case 1 + Load Case 2

Load Combination 1B: Load Case 4 + Load Case 2

Load Combination 1C: Load Case 1 + Load Case 5

Load Combination 1D: Load Case 4 + Load Case 5

Load Combination 2A: Load Case 1 + Load Case 3

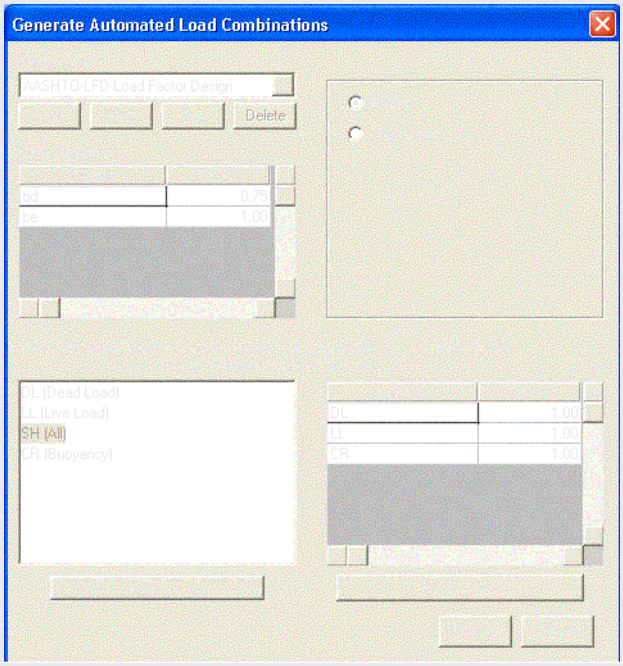
Load Combination 2B: Load Case 4 + Load Case 3

Load Combination 3A: Load Case 2 + Load Case 3

Load Combination 3B: Load Case 5 + Load Case 3

The number of load cases selected per class is not limited to two. Any selection of load cases can be used.

Using Auto Load Combinations



Auto Load Combiantions

The *Design Codes* list display the standard design codes and any user-created codes previously entered. Select a code from the list.

☞The code's combinations can be viewed by clicking the *View* button.

Some codes have factors that can be set by the engineer. The *Code Variables* spreadsheet contains these factors to be set. Variables have default values when a code specifies them. Enter values for these variables where needed.

The list on the lower left displays load cases from the Load Cases Explorer or result cases from the Analysis Results explorer, depending on whether the tool was opened from the **Input Data** or **Results** menu.

☞Choose which load cases/result cases to combine by selecting them from the list on the left and clicking the *Add* button.

☞On top of the factors specified by the design code, factors may be applied to individual load cases as they are used by the code. A factor of 1.00 indicates not additional user factor is to be applied on top of the code.

☞If multiple load cases with the same load class have been selected, choose whether simple combinations or permutations should be used. The difference is explained above.

☞Click *Generate* to create the new load combinations or linear result combinations. If this tool is run more than once, clicking *Generate* will not delete load cases or result combinations previously created by the tool.

Creating New Codes

User-defined codes can be created and used in the Auto Load Combinations tool.

☞*New* is used to create a new user defined design code.

☞Or, use *Copy* to create a new user code based on an existing code.

☞Click *Edit* to edit a user-defined code.

A window with a spreadsheet will appear. The rows of the spreadsheet are code-based combinations, and the columns are load classes. For each combination, enter the factor to be used for each load class in the appropriate column. For load classes not included in the combination, leave the factor as zero.

A factor can be applied to all load classes, on top of any factors specified per class, by entering a factor in the first factor column.

Factors that must be set by the engineer at the time of generating combinations (code variables) can be entered using a variable name instead of a numeric factor. Variable names must start with a letter.

User codes are stored in `My Documents\LARSA Projects\User Codes` with the file extension `code`.

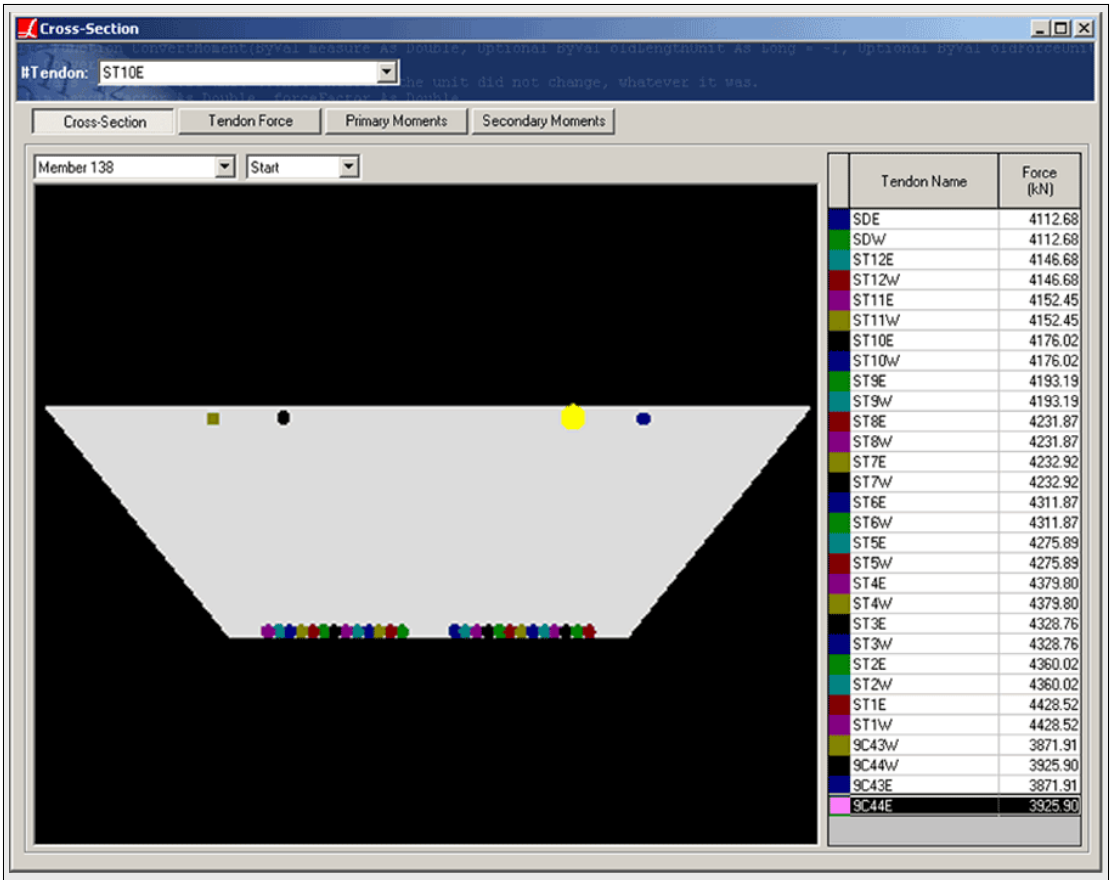
Tendon Results Tools

The tendon results tool visualizes tendon results, including geometry, losses and primary/secondary moments.

This tool can be found on the Results menu under "Tendon Results."

Tendons at Cross-Section

This tool shows all the tendons passing through at the given location of a member on its cross-section. Each tendon shown inside the cross-section is color coded and matched with the spreadsheet showing tendon forces at this location.



Tendon Cross-Sectional View

☞Select the tendon from the list box located at the top left side of the window. This will fill in the list of members located on the path of the selected tendon.

☞Select the member and the location on that member (start/end) to view.

Tendon Forces

This tool graphs short-term and long-term losses along the PT path. The vertical axis shows the force in the tendon. The horizontal axis shows the distance from the start of the tendon path. Short-term losses are represented in dark red, and long-term losses are shown in blue.

Primary Moments

This tool shows the primary moments at the end of each selected member graphed according to the members' x-coordinates. The horizontal axis is the member end x-coordinate, and the vertical axis is the primary moment at that location.

Secondary Moments

This tool shows the secondary moments at the end of each selected members, graphed according to the members' x-coordinates, as with the primary moments graph.

For More Information

- The Tendon Pseudoelement [in *LARSA 2000 Reference*].
- Tendon Results [in *LARSA 2000 Reference*].

Steel Design

The Steel Design Plugin performs code checks and design for a variety of codes.

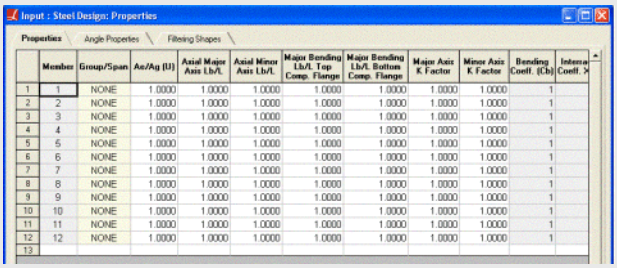
Steel design has been designed as a plugin to LARSA 2000. It is not included in LARSA 2000 installation. You will have to download the separate *Steel Design Plugin* installation file from our LARSA website. For more information on installing plugins please refer to Installing Plugins [p163].

All Steel Design commands are on the **Design** menu.

Preparing Input

Steel Design will only code check or design members that have been marks as to be code checked or designed. To mark members for Steel Design, open the **Design** and click **Add Selected Members**. This tool will mark all selected members to be analyzed for steel design.

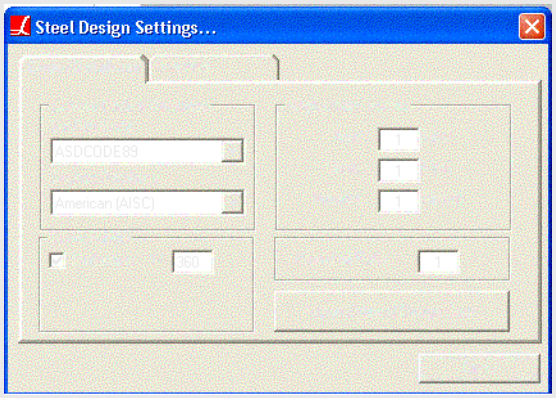
Each member to be code checked or designed needs to be given design parameters, such as unbraced length and K factors. Steel Design parameters can be edited by choosing **Design Parameters** from the menu.



	Member	Group/Span	Ae/Ag (U)	Axial Major Axis Lb/L	Axial Minor Axis Lb/L	Major Bending Lb/L Top Comp. Flange	Major Bending Lb/L Bottom Comp. Flange	Major Axis K Factor	Minor Axis K Factor	Bending Coeff. (Cb)	Interaction Coeff. K
1	1	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
2	2	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
3	3	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
4	4	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
5	5	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
6	6	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
7	7	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
8	8	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
9	9	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
10	10	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
11	11	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
12	12	NONE	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1.0000	1	
13											

Steel Design Parameters

Options



Code/Database

Other options need to be set before running a code check or design.

Select **Options** from the menu.

Section Code and Database

Choose a design code and section database that will be used in the code check.

Use Structure Groups in Design

Turn on/off the option to use structure groups during the code check / design. The use of structure groups to code check or design a group of members together is explained below.

Design Failing Members Only

Steel Design will first investigate the members and design only the ones that are failing.

Neglectable Forces

All forces that are below the specified values here will be ignored during the code check/design.

Pass/Fail Ratio

Code check and design uses this ratio to identify the failing members.

Import External Design Code

Users can write their own design codes as plugins to be used directly from LARSA. If you have a user-written design plugin, click *Import External Design Code* to load the design plugin. For more information, refer to Creating a Steel Design Code [in *LARSA 2000 Developer's Guide*].

Using Structure Groups and Spans

All members that are in the same structure group [p60] will be designed with the same section.

All members that are in the same span [see "Spans" in *LARSA 2000 Reference*] will be code checked/designed as one member. The code will use the length of the span instead of the length of the individual members during both code check and design. Steel Design will choose one section for the whole span. The results can be viewed for each member separately.

If a member is both in a structure group and a span, assignment to a structure group is ignored.

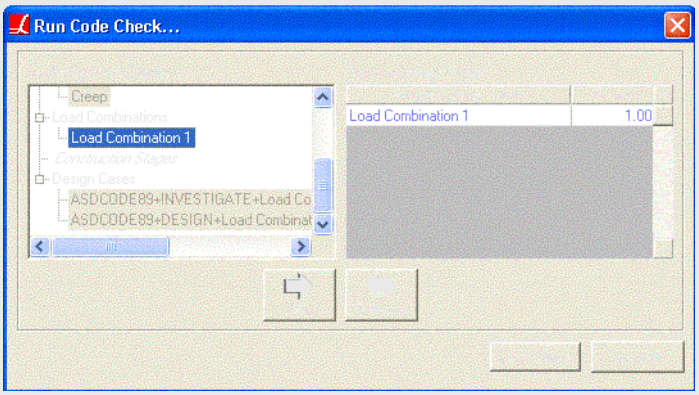
You can turn these options off from the *Steel Design Plugin Options* dialog box.

Code Check

Before the code check make sure the model has been analyzed.

In the **Design** menu choose **Code Check**.

From this window you can select Load Combinations [in *LARSA 2000 Reference*] to be code checked.



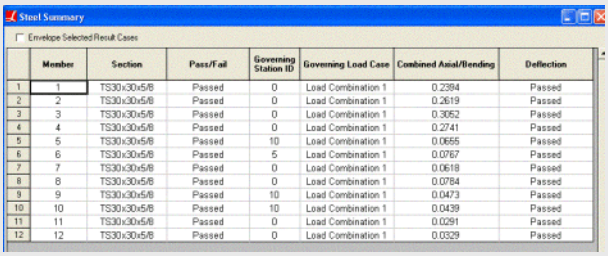
Run Code Check

To select a load combination to be code checked, double-click the load combination or click it once and then click the *Add* button. It will be added to the list on the right, which shows which combinations will be code checked.

Once you add the load combination, you can edit or remove it from the spreadsheet. To remove a load combination, select the combination and click *Remove*.

After you select the load combinations, run the code check by clicking the *Code Check* button.

When the program finishes the code check, you can see the results. Open the **Design** menu and select **Results : Summary**, **Results : Axial/Bending**, or **Results : Graphics**.



Steel Summary

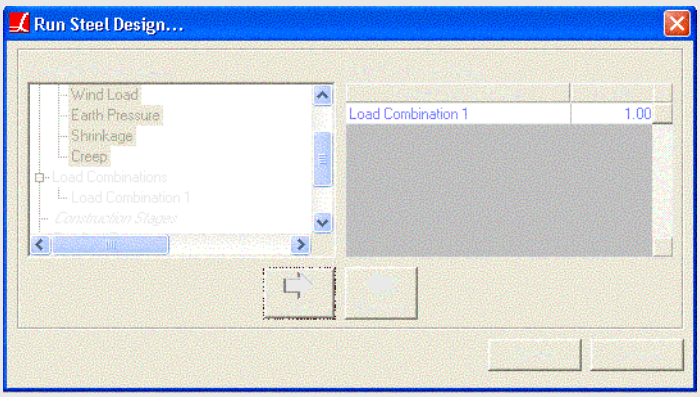
Envelope Selected Result Cases

	Member	Section	Pass/Fail	Governing Station ID	Governing Load Case	Combined Axial/Bending	Deflection
1	1	TS30x30x5/8	Passed	0	Load Combination 1	0.2394	Passed
2	2	TS30x30x5/8	Passed	0	Load Combination 1	0.2619	Passed
3	3	TS30x30x5/8	Passed	0	Load Combination 1	0.3052	Passed
4	4	TS30x30x5/8	Passed	0	Load Combination 1	0.2741	Passed
5	5	TS30x30x5/8	Passed	10	Load Combination 1	0.0655	Passed
6	6	TS30x30x5/8	Passed	5	Load Combination 1	0.0767	Passed
7	7	TS30x30x5/8	Passed	0	Load Combination 1	0.0618	Passed
8	8	TS30x30x5/8	Passed	0	Load Combination 1	0.0784	Passed
9	9	TS30x30x5/8	Passed	10	Load Combination 1	0.0473	Passed
10	10	TS30x30x5/8	Passed	10	Load Combination 1	0.0439	Passed
11	11	TS30x30x5/8	Passed	0	Load Combination 1	0.0291	Passed
12	12	TS30x30x5/8	Passed	0	Load Combination 1	0.0329	Passed

Result Spreadsheet

Design

To perform a design, follow the same steps through Code Check. Then, choose **Design** from the **Design** menu.



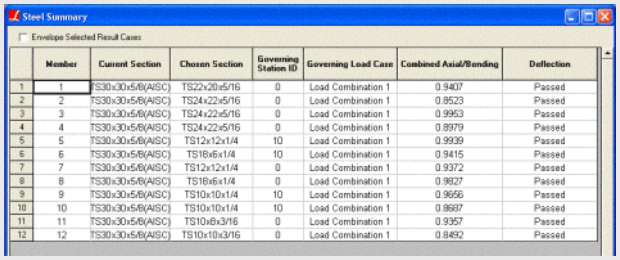
Run Steel Design

To select a load combination to be designed, double-click the load combination or click it once and then click the *Add* button. It will be added to the list on the right, which shows which combinations will be code checked.

Once you add the load combination, you can edit or remove it from the spreadsheet. To remove a load combination, select the combination and click *Remove*.

Click *Design* to perform the design.

Then check the results following the same steps as in Code Check.



Steel Summary

Envelope Selected Result Cases

	Member	Current Section	Chosen Section	Governing Station ID	Governing Load Case	Combined Axial/Bending	Deflection
1	1	TS30x30x5/8(AISC)	TS22x20x5/16	0	Load Combination 1	0.9407	Passed
2	2	TS30x30x5/8(AISC)	TS24x22x5/16	0	Load Combination 1	0.8523	Passed
3	3	TS30x30x5/8(AISC)	TS24x22x5/16	0	Load Combination 1	0.9953	Passed
4	4	TS30x30x5/8(AISC)	TS24x22x5/16	0	Load Combination 1	0.8979	Passed
5	5	TS30x30x5/8(AISC)	TS12x12x1/4	10	Load Combination 1	0.9939	Passed
6	6	TS30x30x5/8(AISC)	TS16x6x1/4	10	Load Combination 1	0.9415	Passed
7	7	TS30x30x5/8(AISC)	TS12x12x1/4	0	Load Combination 1	0.9372	Passed
8	8	TS30x30x5/8(AISC)	TS16x6x1/4	0	Load Combination 1	0.9827	Passed
9	9	TS30x30x5/8(AISC)	TS10x10x1/4	10	Load Combination 1	0.9656	Passed
10	10	TS30x30x5/8(AISC)	TS10x10x1/4	10	Load Combination 1	0.8697	Passed
11	11	TS30x30x5/8(AISC)	TS10x8x3/16	0	Load Combination 1	0.9357	Passed
12	12	TS30x30x5/8(AISC)	TS10x10x3/16	0	Load Combination 1	0.8492	Passed

Design Results Spreadsheet

Graphics

You can view the graph of the ratio for each steel design result case by using **Results:Graphics** under **Design** menu, which will activate the steel design result graphics view. Steel design result graphics view graphs the ratio along the members when the rendering mode is off. If the rendering mode is on, steel design graphics view will draw passing members in yellow and the failing ones in red. The label for each member can be modified at the *Graphics* tab of *Steel Design Plugin Options* dialog box.

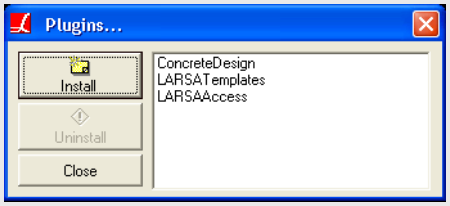
For More Information

- For an overview of installing and uninstalling plugins, see Installing Plugins [p163].

Installing Plugins

Plugins are special programs that attach themselves into LARSA 2000 to expand its functionality. All plugins are stored in files with the ".plugin" extension and they can be downloaded from our website.

Installing Plugins



Plugins Dialog box

Before installing a plugin please make sure that you don't have a previous version of the same plugin already installed. You can check the list of already installed plugins in plugins dialog box.

To install a new plugin:

- Open the **Tools** menu and click **Plugins...**.
- From the plugins dialog box hit *Install*.
- Finally, browse to the plugin file you have downloaded from our website and hit *Open* to activate installation.

The plugin will become activated in LARSA 2000 immediately. For example, you should be able to see the **Design** menu right after the installation of the Steel Design Plugin.

Uninstalling Plugins

With every new version, plugins may loose the compatibility with LARSA 2000, or there may just be a new version of the plugin that you would like to use. For both cases you have to download the compatible version of the plugin from LARSA website. Before installing the new version, you will have to uninstall the existing one.

- Open the **Tools** menu and click **Plugins...**.
- Select the name of the plugin from the list and hit *Uninstall*.

LARSA 2000 will have to close down for the changes to take effect. It will not proceed with uninstallation of the plugin if you have an unsaved active project.

For More Information

- For an overview of plugins, see LARSA 2000 Plugin Technology [in *LARSA 2000 Developer's Guide*].

Index