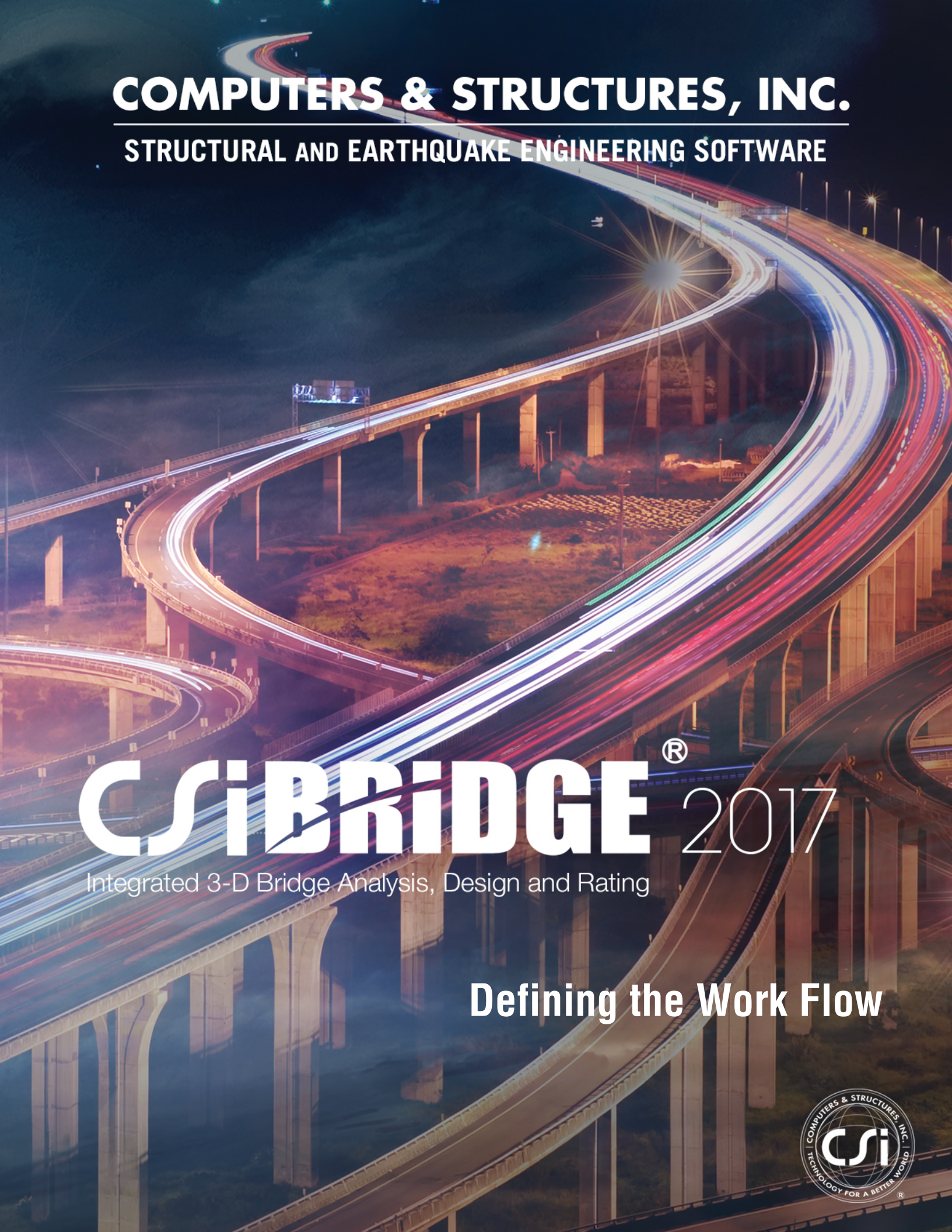

COMPUTERS & STRUCTURES, INC.
STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE
CSiBRIDGE® 2017
Integrated 3-D Bridge Analysis, Design and Rating
Defining the Work Flow
COMPUTERS & STRUCTURES, INC.
CSi
TECHNOLOGY FOR A BETTER WORLD

CSiBridge®

Defining the Work Flow

ISO BRG102816M2 Rev. 0
Proudly developed in the United States of America
October 2016

Copyright

Further information and copies of this documentation may be obtained from:

Computers & Structures, Inc.
www.csiamerica.com

info@csiamerica.com (for general information)
support@csiamerica.com (for technical support)

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND TESTING OF THIS SOFTWARE. HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THIS PRODUCT.

THIS PRODUCT IS A PRACTICAL AND POWERFUL TOOL FOR STRUCTURAL DESIGN. HOWEVER, THE USER MUST EXPLICITLY UNDERSTAND THE BASIC ASSUMPTIONS OF THE SOFTWARE MODELING, ANALYSIS, AND DESIGN ALGORITHMS AND COMPENSATE FOR THE ASPECTS THAT ARE NOT ADDRESSED.

THE INFORMATION PRODUCED BY THE SOFTWARE MUST BE CHECKED BY A QUALIFIED AND EXPERIENCED ENGINEER. THE ENGINEER MUST INDEPENDENTLY VERIFY THE RESULTS AND TAKE PROFESSIONAL RESPONSIBILITY FOR THE INFORMATION THAT IS USED.

Contents

Chapter 1 Introduction

1.1 Graphical User Interface 1-3

1.2 Organization 1-12

1.3 Recommended Reading/Practice 1-13

Chapter 2 File

2.1 File > New 2-2

2.2 File > Open 2-4

2.3 File > Save and Save As 2-5

2.4 File > Import 2-5

2.5 File > Export 2-7

2.6 File > Batch File 2-10

2.7 File > Print 2-10

2.8 File > Report 2-11

2.9 File > Pictures 2-14

2.10 File > Settings 2-15

2.11 File > Language 2-21

Chapter 3 Home

3.1 Home > Bridge Wizard 3-1

3.1.1 Using the Bridge Wizard 3-5
3.1.2 Steps of the Bridge Wizard 3-6

3.2 Home > View 3-15

3.3 Home > Snap 3-17

3.4 Home > Select 3-18

3.5 Home > Display 3-21

Chapter 4 Layout

4.1 Layout > Layout 4-2

4.1.1 Bridge Layout Preferences Form – Screen Capture 4-5
4.1.2 Bridge Layout Line Data Form – Screen Capture 4-6

4.2 Layout > Lanes 4-10

4.2.1 Bridge Lane Data Form – Screen Capture 4-14
4.2.2 Lane Data Form – Screen Capture 4-15

Chapter 5 Components

5.1 Components > Properties 5-2

5.1.1 Material Properties Forms – Screen Capture 5-7
5.1.2 Frame Properties Form – Screen Capture 5-10
5.1.3 Section Designer – Screen Capture 5-11

5.1.4 Cable Properties Form – Screen Capture 5-13
5.1.5 Tendon Properties Form – Screen Capture 5-14
5.1.6 Link/Support Properties Form – Screen Capture 5-15
5.1.7 Rebar Properties Form – Screen Capture 5-16

5.2 Components > Superstructure 5-16

5.2.1 Bridge Deck Section Form – Screen Capture 5-20
5.2.2 Bridge Diaphragm Form – Screen Capture 5-22
5.2.3 Parametric Variation Forms – Screen Capture 5-24

5.3 Components > Substructure 5-25

5.3.1 Bridge Bearing Data Form – Screen Capture 5-30
5.3.2 Bridge Restrainer Data Form – Screen Capture 5-31
5.3.3 Foundation Spring Data Form – Screen Capture 5-32
5.3.4 Bridge Abutment Data Form – Screen Capture 5-33
5.3.5 Bridge Bent Forms – Screen Captures 5-34

Chapter 6 Loads

6.1 Loads > Vehicles 6-2

6.1.1 Vehicle Data Forms – Screen Captures 6-4
6.1.2 Vehicle Classes Data Forms – Screen Capture 6-6

6.2 Loads > Load Patterns 6-7

6.3 Loads > Functions 6-9

6.3.1 Response Spectrum Forms – Example Screen Capture 6-14
6.3.2 Time History Form – Example Screen Capture 6-15

6.4 Loads > Loads 6-16

6.4.1 Point Load Form – Screen Capture 6-20
6.4.2 Line Load Form – Screen Capture 6-20
6.4.3 Area Load Form – Screen Capture 6-21
6.4.4 Temperature Gradient Form – Screen Capture 6-21

Chapter 7 Bridge

7.1 Bridge > Bridge Objects 7-2

7.1.1 Bridge Object > Span – Screen Capture 7-8
7.1.2 Bridge Object > Span Items – Screen Captures 7-10
7.1.3 Bridge Object > Supports 7-13
7.1.4 Bridge Object > Superelevation – Screen Capture 7-14
7.1.5 Bridge Object > Prestress Tendons – Screen Captures 7-15
7.1.6 Bridge Object > Girder Rebar – Screen Capture 7-16
7.1.7 Bridge Object > Loads – Screen Capture 7-16
7.1.8 Bridge Object > Groups – Screen Captures 7-19

7.2 Update 7-20

7.2.1 Update > Update 7-21
7.2.2 Update > Auto Update 7-22

Chapter 8 Analysis

8.1 Analysis > Load Cases 8-2

8.1.1 Analysis > Load Cases – Type 8-9

8.1.2 Analysis > Load Cases > Schedule Stages – Screen Capture 8-16
8.1.3 Analysis > Load Cases > Convert Combos – Screen Captures 8-17
8.1.4 Analysis > Load Cases > Show Tree 8-18

8.2 Analysis > Bridge 8-18

8.3 Analysis > Model Lock 8-20

8.4 Analysis > Analyze 8-20

8.4.1 Analysis > Analysis Options – Screen Captures 8-22
8.4.2 Analysis > Run Analysis – Screen Capture 8-22
8.4.3 Analysis > Last Run Details 8-23
8.5 Analysis > Shape Finding 8-23

Chapter 9 Design/Rating

9.1 Design/Rating > Load Combinations 9-2

9.2 Design/Rating > Superstructure Design 9-7

9.2.1 Superstructure Design > Preferences – Screen Capture 9-9
9.2.2 Superstructure Design > Design Requests – Screen Capture 9-10
9.2.3 Superstructure Design > Run Super – Screen Capture 9-11
9.2.4 Superstructure Design > Optimize – Screen Capture 9-11

9.3 Design/Rating > Seismic Design 9-12

9.3.1 Seismic Design > Preferences – Screen Capture 9-14
9.3.2 Seismic Design > Design Requests – Screen Capture 9-15
9.3.3 Seismic Design > Run Seismic – Screen

Capture 9-16

9.4 Design/Rating > Load Rating 9-16

9.4.1 Load Rating > Preferences – Screen Capture 9-18
9.4.2 Load Rating > Rating Requests – Screen Capture 9-19
9.4.3 Load Rating > Run Rating – Screen Capture 9-20
9.4.4 Load Rating > Optimize – Screen Capture 9-21

Chapter 10 Advanced

10.1 Advanced > Edit 10-1

10.2 Advanced > Define 10-4

10.3 Advanced > Draw 10-8

10.4 Advanced > Assign 10-10

10.5 Advanced > Assign Loads 10-22

10.6 Advanced > Analyze 10-27

10.7 Advanced > Tools 10-33

Bibliography

CHAPTER 1 Introduction

CSiBridge is the most productive bridge design package in the industry because it integrates modeling, analysis, and design of bridge structures into a versatile and easy-to-use computerized tool. Terms familiar to bridge engineers are used to define bridge models parametrically: layout lines, spans, bearings, abutments, bents, hinges, and post-tensioning. Spine, shell, or solid object models can be created and update automatically as the bridge definition parameters are changed. Simple or complex bridge models can be built and changes can be made efficiently while maintaining total control over the design process. Lanes and vehicles can be defined quickly and can include width effects. Simple and practical Gantt charts can be created to simulate modeling of construction sequences and scheduling.

This manual provides a quick reference to the features and commands available in CSiBridge. Figure 1-1 identifies a general work flow and the related graphical user interface (GUI) components used to accomplish the tasks shown. This chapter describes the GUI, the organization of subsequent chapters, and suggested additional reading material.

Note: When the program first launches, a Welcome form will display. Click **Continue** in the lower right-hand corner to move past the form. Click the *Do not show this Welcome Screen again* check box to permanently close the form.

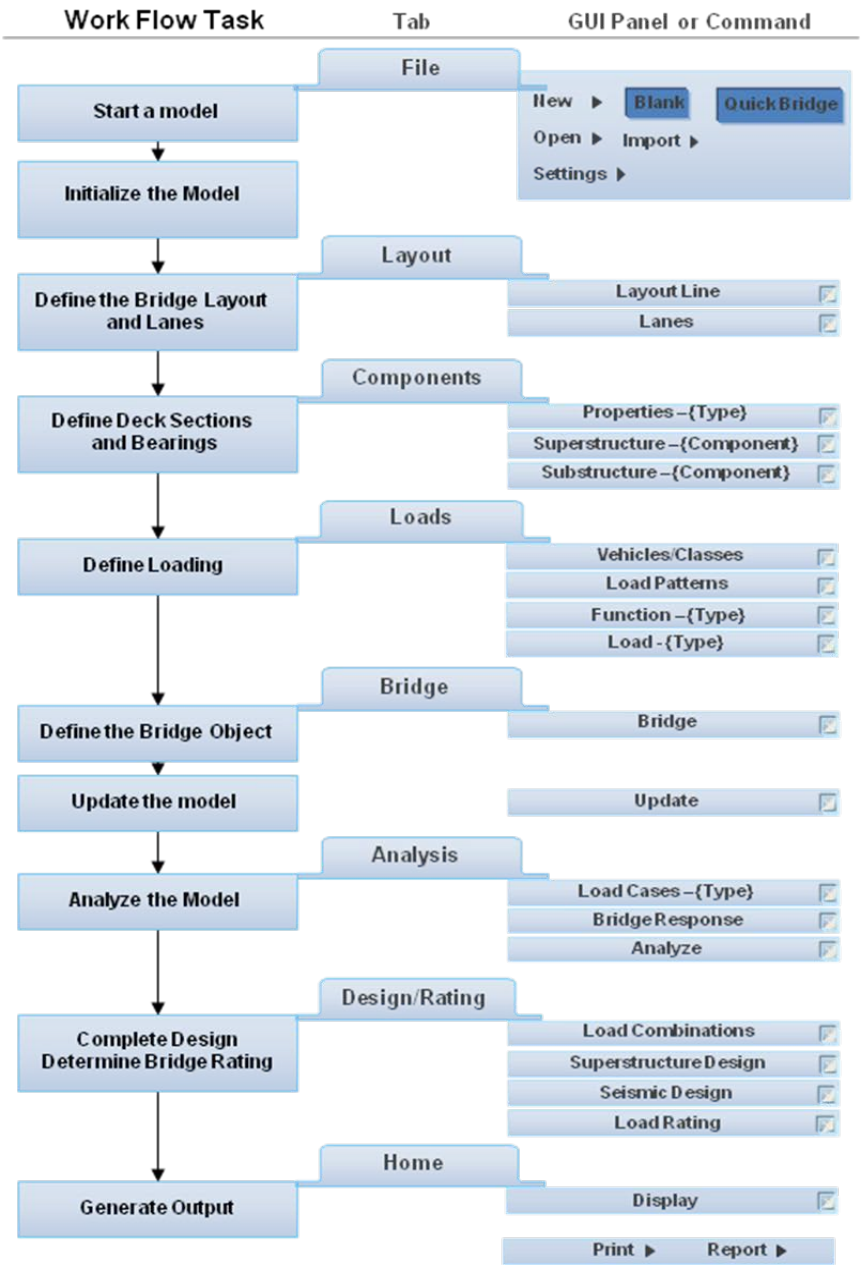


Figure 1-1 Basic Work Flow and Related Graphical User Interface Tabs and Panels or Commands

The ***Bridge Wizard*** is a key feature in the program; it is a step-wise guide through the model creation and analysis. The ***Wizard*** is an asset to both the beginner and more advanced user because it works seamlessly with other program functions, meaning that users are not "locked in" to the ***Wizard***. A model can be initiated using the ***Wizard*** and then individual commands "outside" the ***Wizard*** can be used to adjust the model geometry, components, analysis and design parameters, and so on. Similarly, the ***Wizard*** can be used at any point to "pick up" the modeling process that was not initiated using the ***Wizard*** by selecting the appropriate "Bridge Object." Details about the ***Bridge Wizard*** are provided in Chapter 3.

1.1 Graphical User Interface

The GUI consists of some elements familiar to Microsoft Windows users as well as command functions familiar to users of other CSi programs. The components of the interface and the logic behind their arrangement is explained in this section. More detailed descriptions of commands can be found in subsequent chapters.

The title bar, display window(s), and status bar are common elements in Windows-based programs.

- As is typical, the **title bar** displays the name of the program (i.e., CSiBridge) and the name of the model file. The far right-hand side of that bar includes the Windows minimize, maximize, and close buttons.

- When a model file is opened, it is shown in a **display window**. Click in the window to activate it; actions related to the model (e.g., draw, select, and so on) are carried out in an "active window." Click the "expand arrow" on the far upper right-hand corner of the display window to display the "+ Add New Window" option and open an additional window. The tab on the left-hand side of the display window identifies the type of view (e.g., 3D, X-Y Plane @ Z=0). Click the "x" close button in the upper right-hand corner of the tab to close a display window. At least one display window must remain open.

- The **status bar** at the very bottom of the program window shows the x, y, and z coordinates of the mouse cursor in the active display window, the coordinate system being used by the display, and the units being used in the model.

Figure 1-2 shows a ribbon of the user interface, annotated with the terminology used in this manual.

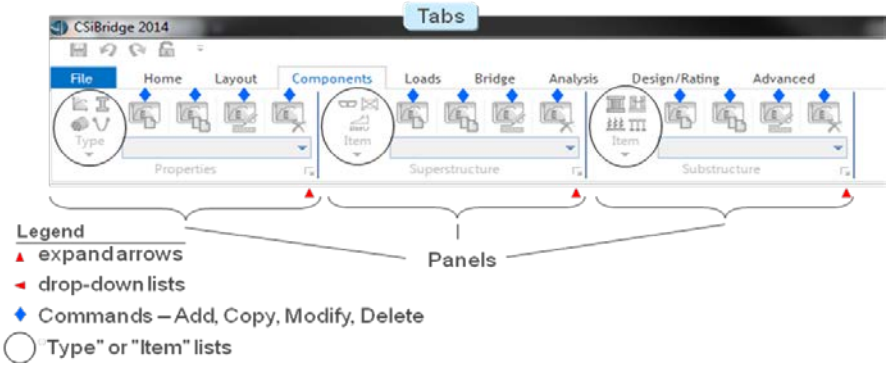


Figure 1-2 A ribbon of the Graphical User Interface annotated with the terminology used in this manual

When the **File** tab is clicked, a drop-down menu of commands displays. Table 1-1 identifies the features available on the File menu. More information about the File is provided in Chapter 2.

Table 1-1 File Commands and Features – See Chapter 2 for more information

Command	Features
New	▪ Initialize the model ▪ Set the base units ▪ Record project information (client name, and so on) ▪ Select a start option: Blank or Quick Bridge ▪ A display area on the right-hand side of the menu shows the recently stored model files
Open	Open an existing model file
Save / Save As	Save a bridge model / Save the model using a new name

Table 1-1 File Commands and Features – See Chapter 2 for more information

Command	Features
Import	Import files in these formats: text stored in ASCII format; Excel; Access; SAP2000; CIS/2; SDNF; AutoCAD; IFC; IGES; Nastran; STAAD/ GTSTRUDL; StruCAD*3D; LandXML
Export	Export files in these formats: text into ASCII format; Excel; Access; CIS/2; SDNF; AutoCAD; IFC; IGES; Perform 3D (text file), Perform 3D Structure
Batch File	▪ Run the analysis and manage the analysis files for a list of CSiBridge model files with no additional action required by the user; useful for running multiple models when the computer is unattended (e.g., overnight)
Print	▪ Print Graphics ▪ Print Tables ▪ Print Setup
Report	▪ Create Report ▪ Report Setup ▪ Advanced Report Writer
Pictures	▪ Create files in bitmap format of the entire screen, the main program window, the current window including the title bar; the current window without the title bar, or a user specified region. ▪ Create a metafile of the current display window. ▪ Create a multi-step animation video or a cyclic animation video of the model showing the current analysis results.
Settings	▪ Units – Set the number formatting to be followed by any database generated by the program. ▪ Tolerance – Auto merge tolerance, 2D view cutting plane, plan fine grid spacing, plan nudge value, screen selection tolerance, screen snap to tolerance, screen line thickness, printer line thickness, maximum graphic font size, minimum graphic font size, auto zoom step, shrink factor, maximum line length in text file ▪ Database table utilities and settings -- Set Current Format File Source, Edit Format File, Set Current Table Name Source, Write Default Tables Names to XML, Documentation to Word, Auto Regenerate Hinges after Import ▪ Colors – Change default color settings for on-screen display and printed output. ▪ Other settings – Graphics mode, auto save, auto refresh, show bounding plane, moment diagrams on tension side, sound, show result values while scrolling ▪ Project Information – Company name, client name, project name, project number, model name, model description, revision

Table 1-1 File Commands and Features – See Chapter 2 for more information

Command	Features
	number, frame type, engineer, checker, supervisor, issue code, design code ▪ Comments and Log – Track the status of the model, keep a "to-do" list, and retain key results that can be used to monitor the effects of changes to the model.
Languages	English and Chinese
Resources	Help, Documentation, CSI on the Web, CSiBridge News, About CSiBridge
Exit	Closes the program

When a form is displayed in the program, clicking the F1 key will display a context-sensitive help topic.

Near the top of the program window, but beneath the title bar, is a short **menu bar of icons** that can be clicked to perform frequently required tasks, such as Save or Lock and Unlock a model. In addition, a right click will display the Customized Quick Access Toolbar. Click a command on the list of commands to add that command to the menu bar. A check mark preceding a command indicates that the command has been added. To remove a command, click the command to uncheck it. After a command has been added to the menu bar, clicking it will immediately execute the command. An option is also available to change the color – blue, silver, black – used to display the ribbon.

Below that menu bar of icons and to the right File tab is a series of eight other **tabs**: **Home**, **Layout**, **Components**, **Loads**, **Bridge**, **Analysis**, **Design/Rating**, and **Advanced**. When read from left to right, the names of the tabs generally reflect the sequence of actions required to generate a model. Click any tab at any time to display its contents, which consists of panels.

Panels are grouped on a particular tab because of the generally common nature of their function. For example, the **Home** tab includes the *View*, *Snap*, *Select* and *Display* panels. The names of those panels reflect the functions related to working with the *active view*. The *View* panel in-

cludes commands to set a 3D, XY, or XZ view, to access zoom features, to Set Display Options, and many other view-related commands.

The *Snap* panel tools are used to increase accuracy and speed when drawing and editing objects in the *active view*. The *Select* panel includes the commands used to select and deselect objects in the *active view*. The *Display* panel includes the commands to specify what is shown on the model in the *active view*.

The composition of the panels varies somewhat, depending on their general function. For example, the *View*, *Snap*, *Select*, and *Display* panels on the **Home** tab have icons and drop-down lists of commands that generally immediately execute actions or display forms with options to filter actions, such as selecting material properties. Alternatively, the **Components** tab, for example, includes panels of commands that are used to add, modify, or delete bridge component definitions (e.g., material, frame, or cable properties; deck sections or diaphragms; bearings, restrainers, or foundation springs). Thus, the panels on the **Components** tab have **Type** and **Item** commands to select the type of bridge component and associated commands and expand arrows that when clicked, display {component type} **definition** forms that are used to name a definition and that have buttons that can be used to display {component type} **data** forms that are used to specify parameters for the named definition.

Note: Hover text displays the functions of icons and drop-down lists when the mouse cursor is moved over them.

- Clicking the small expand arrow in the lower right-hand corner of a panel displays the form used to add, modify, or delete *definitions*. (That definition form may have buttons that subsequently can be used to display the data form referenced in the next bullet item.)

- In most cases, clicking the first of the four commands above the drop-down list on a panel is a "shortcut" to the form used to define *data* for a new definition. (The hover text for the first of the four command icons should read something similar to "New, Add a new {*component*}," while hover text that displays when the cursor is placed over

the down arrow of the drop-down list should read something similar to "Current {*Property* or *Item*}.") Note that clicking the third command icon (hover text may read "Modify, Modify/Show the specified {*component* or *item type*}") will display the data form for the definition selected in the drop-down list.

Some panels include "*More*" buttons. Clicking those buttons displays drop-down menus of additional commands.

IMPORTANT NOTE: For the sake of brevity, the use of the words tab, panel, and icon in command names has been eliminated. For example, the command used to access the Display Options for Active Window form is the ***Home > View > Set Display Options*** command, which means: click the Home tab, then on the View panel, click the Set Display Options icon.

The program actions/options on each of the tabs are identified briefly in Table 1-2. The table also identifies the chapters in this manual devoted to each of the tabs.

Table 1-2 CSiBridge Tabs, Panels, Actions/Options and Associated Chapters

Tab	Panel	Actions/Options
Home (Chapter 3)	Bridge Wizard	Step-wise guide through the creation, analysis, and design processes
	View	Zoom features, pan, set views, rotate a view or perspective toggle, refresh window, shrink objects, set display options, set limits, show grids, show axes, invert view selection, remove and restore selection from view, show all, refresh view
	Snap	Snap to points; snap perpendicular; snap to ends and midpoints; snap to lines and edges; snap to intersections; snap to fine grids
	Select	Pointer/window, poly, intersecting poly, intersecting line, coordinate specification (3D box, specified coordinate range, click joint in XY plane, XZ plane, YZ plane), select lines parallel to (click straight line object, coordinate axes or plane), properties (materials, frame sections, cables, tendons, area sections, solids, links, frequency dependent link), assignments (joint supports,

Table 1-2 CSiBridge Tabs, Panels, Actions/Options and Associated Chapters

Tab	Panel	Actions/Options
		joint constraints), groups, labels, all; deselect; select using tables, invert selection, get previous selection, select using intersecting line, clear selection
Home (Chapter 3) (continued)	Display	Show undeformed shape, show bridge superstructure forces/stresses, show deformed shape, show shell force/stress plots, show bridge loads, show bridge superstructure design results, show joint reaction forces, show solid stress plots, show tables, show influence lines/surfaces, show frame/cable/tendon force diagrams, show link force diagrams; save named display, show named display, show load assignments, show miscellaneous assignments, show lanes, show plot functions, show static pushover curve, show hinge results, show response spectrum curves, show virtual work diagrams, show plane stress plots, show asolid stress points, show input/log files
Layout (Chapter 4)	Layout Line	Preferences; initial and end stations; bearing; initial grade, vertical or horizontal layout variations
	Lanes	Selection of the layout line and station for specification of centerline offset and lane width; lane edge type (interior, exterior); object loading by group or program determined; load discretization lengths along and across lanes
Components (Chapter 5)	Properties	Materials, frames, cables, tendons and links, rebar sizes
	Superstructure	Decks, diaphragms, parametric variations
	Substructure	Bearings, restrainers, foundation springs, abutments, bents
Loads (Chapter 6)	Vehicles	Vehicles, vehicle classes
	Load patterns	Dead, vehicle live, wind, temperature, quake, more…
	Function	Response spectrum, time history
	Loads	Point, line, area, temperature gradient
Bridge (Chapter 7)	Bridge Objects	Bridge object, spans, span items (diaphragms, hinges, user points), supports (abutments, bents), superelevation, prestress tendons, girder rebar, loads (point, line, area, temperature gradient), groups

Table 1-2 CSiBridge Tabs, Panels, Actions/Options and Associated Chapters

Tab	Panel	Actions/Options
	Update	Update and auto update
Analysis (Chapter 8)	Load Cases	All, static, nonlinear stage construction, multistep static, modal, response spectrum, time history, moving load, buckling, steady state, power spectral density, hyperstatic; schedule stages (construction schedules), convert combinations, show tree
	Bridge	Bridge response
	Lock	Model lock and unlock
	Analyze	Analysis options, run analysis, last run (show results of last analysis run)
	Shape Finding	Model geometry, reset geometry
Design/Rating (Chapter 9)	Load Combinations	Combination type (Linear Add, Envelope, Absolute Add, SSRS, Range Add, and load case with applicable scale factor), add defaults (code-generated combos)
	Superstructure design	Preferences (code) and design request -- check type, station ranges (i.e., where in the structure the design applies), design parameters (e.g., flexure or stress factors), and demand sets (load combinations to be considered in the design), run superstructure design, optimize design
	Seismic design	Preferences (code) and design request -- an extensive array of parameters (e.g., response spectrum function, seismic design category, P-Delta analysis and so on), run seismic, report
	Load Rating	Preferences (code), rating request, including Rating Type (e.g., flexure, shear, minimum rebar), Station Ranges (i.e., where in the structure the rating applies), Rating Parameters (depends on the Rating Type), Demand Sets (load combinations to be considered in the rating) and if applicable, Live Load Distribution Factors (see Chapter 3 of the *Bridge Superstructure Design* manual), run rating, optimize rating
Advanced (Chapter 10)	Edit	Points, lines, areas, undo/redo, cut/copy/paste, delete, add to model from template, interactive database editing, replicate, extrude, move, divide solids, show duplicates, merge duplicates, change labels

Table 1-2 CSiBridge Tabs, Panels, Actions/Options and Associated Chapters

Tab	Panel	Actions/Options
	Define	Section properties, mass source, coordinate systems/ grids, joint constraints, joint patterns, groups, section cuts, generalized displacements, functions, named property sets (frame and area modifiers, frame releases), pushover parameter sets (force v displacement, ATC 40 capacity spectrum, FEMA 356 coefficient method, FEMA 440 equivalent linearization, FEMA 440 displacement modification), named sets (tables, virtual work, pushover named sets, joint TH response spectra, plot function traces)
	Draw	Set select mode, set reshape object mode, draw one joint link, draw two joint link, draw frame/cable/ tendon, quick draw/frame/cable/tendon, quick draw braces, quick draw secondary beams, draw poly area, draw rectangular area, quick draw areas, draw special joint, quick draw link, draw section cut, draw developed elevation definition, draw reference point, draw/edit general reference line, new labels
	Assign	Joints (restraints, constraints, springs, panel zones, masses, local axes, merge number, joint patterns), frames (sections, property modifiers, material property overwrites, releases/partial fixity, local axes, reverse connectivity, end length offsets, insertion point, end skews, fireproofing, output stations, P-Delta force, lane, tension/compression limits, hinges, hinge overwrites, line springs, line mass, material temperature, automatic frame mesh), areas (section, stiffness modifiers, material property overwrites, thickness overwrites, local axes, reverse local 3 axis direction, area springs, area mass, material temperature, automatic area mesh, general edge constraints), cable (section, property modifiers, material property, output stations, insertion point, line mass, reverse connectivity, material temperature), tendon (properties, local axes, material temperature, tension/compression limits), solid (properties, surface spring, local axes, edge constraints, material temperature, automatic solid mesh, switch faces), link/support (properties, local axes, connectivity), assign to group, update all generated hinge properties, clear dis-

Table 1-2 CSiBridge Tabs, Panels, Actions/Options and Associated Chapters

Tab	Panel	Actions/Options
		play of assigns, copy assigns, paste assigns
Advanced (Chapter 10) (continued)	Assign Loads	Joints (forces, displacements, vehicle response components), frames (gravity, point, distributed, temperature, strain, deformation, target force, auto wave loading parameters, open structure wind parameters, vehicle response components), areas (gravity, uniform, uniform to frame, surface pressures, pore pressure, temperature, strain, rotate, wind pressure coefficient, vehicle response components), cables (gravity, distributed, deformation, strain, target force, temperature, vehicle response components), tendons (gravity, deformation, strain, target force, temperature, tendon force, vehicle response components) solids (gravity, strain, pore pressure, surface pressure, temperature, vehicle response components) link/support (gravity, deformation, target force, response components)
	Analyze	Create analysis model, Model-Alive™
	Frame Design	Steel, concrete, overwrite frame design procedure, lateral bracing
	Tools	Add/show plug ins, CSi load optimizer

1.2 Organization

Chapter 1 of this manual describes the user interface. Chapter 2 explains the function of the File tab. Chapters 3 through 10 describe the various other tabs in the program, using similar content structure. That is, each chapter begins by identifying the general features provided by the tab. An explanation is then provided correlating the tabs to the default definitions created when the Quick Bridge template is used to start the model, and when the ***Bridge Wizard*** or the Blank option (i.e., import model data) is used to work with a model. An annotated graphic of each panel is provided, followed by a table that briefly explains the function of the commands on the panel. When applicable, screen captures follow the table.

1.3 Recommended Reading/Practice

Review of "Watch & Learn" Series™ tutorials, which are found at http://www.csiamerica.com, is strongly recommended before attempting to design a bridge using CSiBridge. Additional information can be found in the on-line Help facility available from the **File > Resources** command.

Also, other bridge related manuals include the following:

- *Introduction to CSiBridge* – Introduces CSiBridge design when modeling concrete box girder bridges and precast concrete girder bridges. The basic steps involved in creating a bridge model are described. Then an explanation of how loads are applied is provided, including the importance of lanes, vehicle definitions, vehicle classes, and load cases. The *Introduction* concludes with an overview of the analysis and display of design output.

- *Bridge Superstructure Design/Rating* – Describes using CSiBridge to complete (1) bridge design in accordance with the AASHTO STD 2002 or AASHTO LRFD 2012 code, the CAN/CSA-S6-06 code and the EUROCODE for concrete box girder bridges, or the AASHTO 2012 LRFD code, the CAN/CSA-S6-06 code and the EUROCODE for bridges when the superstructure includes precast concrete girder or steel I girder bridges with a composite slab, and (2) bridge rating in accordance with the 2011 AASHTO Rating code for concrete box girder bridges and precast concrete girder or steel I girder bridges with a composite slab. Loading and load combinations as well as Live Load Distribution Factors are described. The manual explains how to define and run a design request and provides the algorithms used by CSiBridge in completing concrete box girder, cast-in-place multi-cell concrete box, and precast concrete bridge design in accordance with the AASHTO code. The manual concludes with a description of design output, which can be presented graphically as plots, in data tables, and in reports generated using the Advanced Report Writer feature.

- *Seismic Analysis and Design* – Describes the eight simple steps needed to complete response spectrum and pushover analyses, determine the demand and capacity displacements, and report the demand/capacity ratios for an Earthquake Resisting System (ERS).

CHAPTER 2 File

Clicking the **File** tab displays a drop-down menu of commands related to maintaining the model file (create a new file, open an existing file, save a file), importing data into and exporting data from a model file, setting up a batch of files upon which to run analysis without further user input, producing output (graphics, reports, bitmaps, metafile, animation video), and setting a range of parameters used within the program (units, tolerances, display color, sound, project information, comments and log, and the like). The **Recent Model**s display area shows the recently stored models. **Resources** and **Exit** buttons are along the very bottom of that display area. Use the **Resources** button to access help-type resources, including About CSiBridge and Documentation. When a form is displayed in the program, clicking the F1 key will display a context-sensitive help topic.

This chapter describes the commands found in the File tab.

IMPORTANT NOTE: This manual addresses work flow for models created using the Blank option or the Quick Bridge template. The other templates, which can be used, are not the focus of this manual.

Figure 2-1 illustrates the work flow when starting a model using the Quick Bridge template.

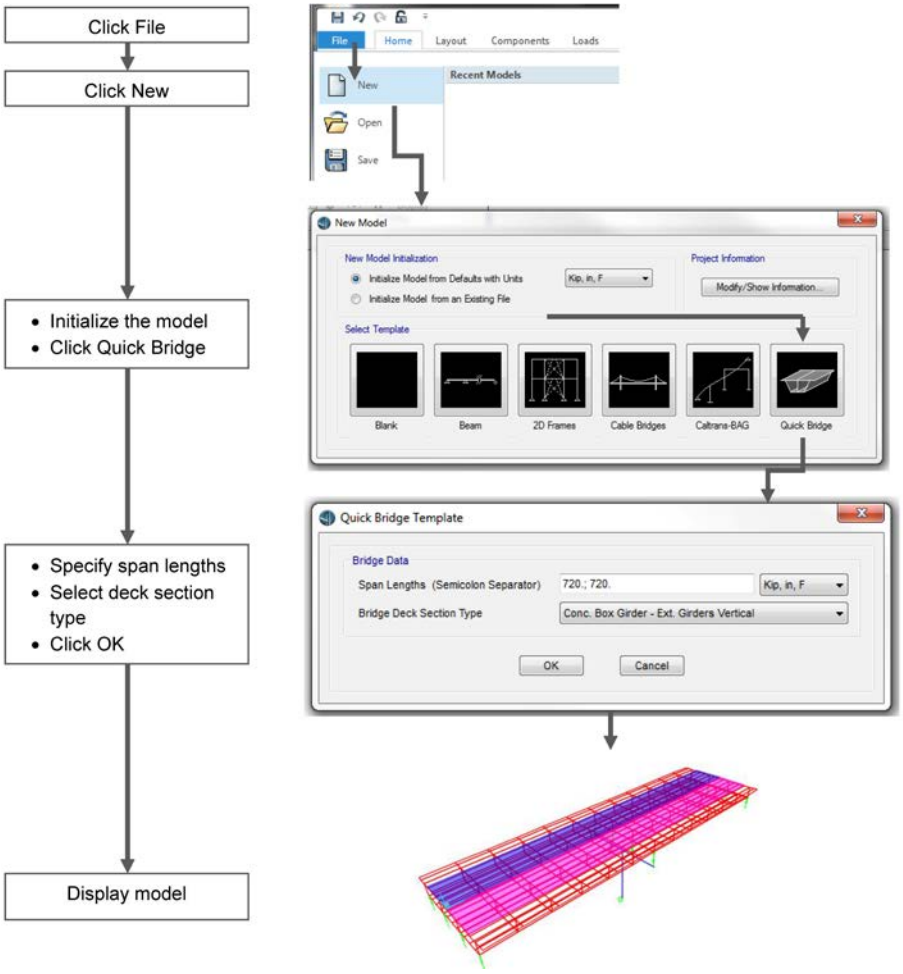


Figure 2-1 Work flow for starting a new model using Quick Bridge

2.1 File > New

Start a new model file by clicking the **File > New** command, which displays the New Model form shown in Figure 2-1. The form has options to initialize the model and to start the model from templates.

Initializing the model determines the units to be used and the default definitions of all properties, components, loading definitions, design settings, and other defined items. Bridge objects and other physical objects (lines, areas, links, and the like), and assignments to these objects, are not included in the initialization process.

When the **Initialize Model for Defaults with Units** option is selected, CSiBridge will use the default program definitions. The default definitions are typical for the type of bridge selected. Use the drop-down list to specify the units to be used in the model.

Note: The units used to start a model become the base units for that model. If different units are used in the model, they are always converted to and from the base units. The model will always open in the base units, so choose the units carefully.

When the **Initialize Model from an Existing File** option is selected, CSiBridge picks up the initial definitions from a previous model. This option generally is preferred if common sets of properties and definitions are used for a number of models of the same project or across projects that benefit from consistency (e.g., for the same client). If this option is selected, click one of the template buttons and, when the program displays the Open Model File form, select the "previous model."

The Quick Bridge option typically produces structures with uniform spacing, unless the spacing is modified using the form that displays after Quick Bridge template has been selected. Table 2-1 identifies the templates.

IMPORTANT NOTE: This manual addresses work flow for models created using the Blank option or the Quick Bridge template. Thus, the work flow in this manual describes using the Bridge Wizard and the various tabs of the graphical user interface: Layout, Components, Loads, Bridge, Analysis, Design/Rating, and Advanced.

Table 2-1 File > New > {Template}

Templates	Description
Blank	Opens the program without any template being loaded. This option can be helpful when a **File > Import** command will be used to initiate a model. Use this option to build the model, analyze it, and design it using the commands on various tabs of the CSiBridge ribbon. The **Bridge Wizard**, a step-wise guide though the bridge modeling and analysis processes, can also be used for model creation and analysis. The **Bridge Wizard** is described in more detail in Chapter 3.
Beam	Opens a single beam bridge model based on a user-specified number of spans and the lengths of the spans, and selection or specification of a section property.
2D Frame	Opens a 2D Frame model based on user-specified parameters. Select from three frame types: Portal Frame, Concentrically Braced Frame and Eccentrically Braced Frame.
Cable Bridge	Creates a cable suspension bridge model based on specification of the deck width, minimum middle sag, and the number of divisions on the left, right, and middle spans.
Caltrans BAG	Opens a model using the Bridge Analysis Generator, which generates a model to perform response spectrum dynamic analysis and static analysis for a concrete bridge structure. The template is most suited for use by the California Department of Transportation in the USA.
Quick Bridge	Opens a typical bridge model based on initial, specified span lengths and selection of a deck section type from a drop-down list of common bridge construction configurations. This template works well with the **Bridge Wizard** or with the commands on the program tabs.

2.2 File > Open

After a CSiBridge model has been created and saved, it may be opened using the **File** > **Open** command. The CSiBridge file may be selected by browsing to locate the appropriate file folder.

2.3 File > Save and File > Save As

The **File** > **Save** command opens a standard Microsoft Windows-type save window. Use the form to specify the name and path for storing the file. The file will have a .BDB extension.

The **File** > **Save As** command can be used to save the file using a new filename.

2.4 File > Import

Clicking the **File** > **Import** command displays a list of subcommands that can be used to import model data in a variety of formats. Various forms will display depending on the type of import. As an example, Figure 2-2 shows the form that displays after the **File** > **Import** > **Text**, **File** > **Import** > **Excel,** or the **File** > **Import Access** commands are used. The options on the form can be used to start a new model with the imported data or add the imported data to an existing model.

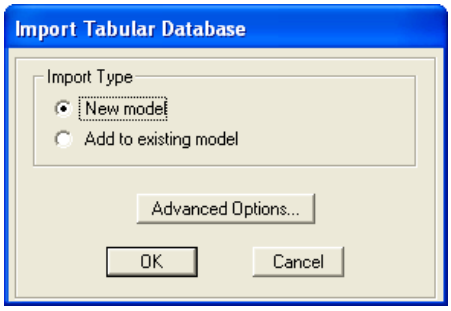


Figure 2-2 Import Tabular Database form

If the *Add to existing model* option is selected, clicking the **Advanced Options** button will open a form that can be used to resolve conflicts between the data in the existing model and the data being imported. Conflicts could be items with the same name or items in the same location, for example.

Table 2-2 identifies the subcommands and the types of files that can be imported.

Table 2-2 File > Import > {Command}

Command	File Extension	Description
Excel	.xls	Imports model definition data that has been stored in Microsoft Excel spreadsheet format as a tabular database, usually with an .xls extension.
Access	.mdb	Imports model definition data that has been stored in a Microsoft Access database format as a tabular database, usually with an .mdb extension.
Text	.$br, .b2k	Imports SAP2000/Bridge or CSiBridge model data that has been stored in plain (ASCII) text format. Each time a model is saved, CSiBridge automatically writes the complete model definition as a tabular database in a text file with a .$br extension. The .$br file is intended for recovering the model in emergency crash situations. Thus, if for some reason the .BDB file will not open, import the corresponding .$br file.
SAP2000	.sdb	Import a SAP2000 model, (i.e., .sdb).
CIS/2	.STP	Imports an .stp file created using the CIMsteel Integration Standard (CIS), which is a set of formal computing specifications used in the steel industry to make software applications mutually compatible.
SDNF	.sdnf, .dat	Imports an .sdnf file created using a Steel Detailing Neutral File. This file format contains steel fabrication and shop drawing information, and can be imported to the model to compare or sync steel members. Used primarily in the USA.
AutoCAD	.dxf	Imports a .dxf file, an AutoCAD Drawing Interchange file. This feature is intended to facilitate importing model geometry from AutoCAD, including AutoCAD r14, AutoCAD 2000 and AutoCAD2002.
IFC	.ifc	Imports Industry Foundation Classes (IFC) model data. IFC is an object-oriented file developed to facilitate interoperability in the building industry.
IGES	.igs	Imports Initial Graphics Exchange Specification (IGES) data, which allows digital exchange of information among computer-aided design systems.
Nastran	.dat	Imports structural analysis models created using NASTRAN; includes geometry, connectivity, material and section properties, loads, and constraint conditions. Assumes that the NASTRAN files are compatible with MSC/NASTRAN Version 68.

Table 2-2 File > Import > {Command}

Command	File Extension	Description
STAAD/ GTSTRU-DL	.std/.gti	Imports structural analysis models created using GTSTRUDL/STAAD; includes geometry, connectivity, material and section properties, loads, and constraint conditions. Because GTSTRUDL/STAAD and CSiBridge have different FEM libraries and different analytical capabilities, not all GTSTRUDL/ STAAD data can be imported.
StruCAD *3D		Imports StruCAD*3D model data. StruCAD*3D is a 3D Finite Element Method software program used in the structural analysis and design of steel and concrete structures.
LandXML	.xml	Import a text-based file to allow project data to be exchanged across different software packages, including points, point groups, description keys, surfaces, parcels, horizontal alignments, profiles, cross-sections.

Note: The Bridge Object, which is generated using the options on the Bridge tab and described in Chapter 7, is the backbone of the modeling process. The Bridge Object definition includes the layout line, the spans, span items (diaphragms, hinges, user points), supports (abutments, bents), superelevation, prestress tendons, girder rebar, and loads. Therefore, CSiBridge cannot automatically incorporate imported data into a Bridge Object definition unless the data previously was defined as part of a SAP2000/Bridge or CSiBridge model.

2.5 File > Export

Clicking the **File** > **Export** command displays a list of subcommands that can be used to export model data in a variety of formats. Exporting to a text file or to Microsoft Excel and Microsoft Access files is among the more common uses of the Export command. When the subcommands for these types of exports are used, a figure similar to that shown in Figure 2-3 is displayed. Use the options on the form to select the specific

data to be exported. If necessary, with the form displayed, depress the F1 key to access a context-sensitive help topic.

Table 2-3 identifies the Export subcommands and the types of data that can be exported. As many files as necessary can be exported from a given CSiBridge model. Each file may contain different tables or may apply to different parts of the model. The files may be used for processing by other programs, for modification before re-importing into CSiBridge, or for any other purpose. However, if the exported file is to contain a complete description of the model, be sure to export all importable model-definition data for the entire structure.

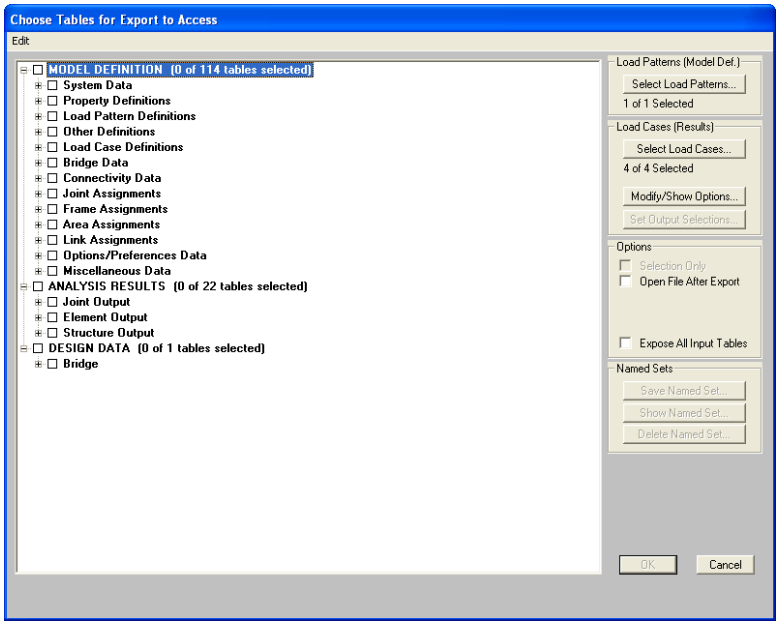


Figure 2-3 Export Tables form

Table 2-3 File > Export > {Command}

Com-mand	File Extension	Description
Text	.$br, .b2k	Exports user-selected data to a user-specified file-name that will have a .b2k extension. The .$br file is intended for recovering the model in emergency crash situations. The .$br file is not a substitute for the data-base file, but it does contain all of the information nec-essary to recreate the model.
Excel	.xls	Exports user-selected data to a user-specified file-name that will have a .xls extension. Depending on the selection made on the *Choose Tables for Export to Excel* form, Microsoft Excel may launch and open the newly created .xls file.
Access	.mdb	Exports user-selected data to a user-specified file-name that will have a .mdb extension. Depending on the selection made on the *Choose Tables for Export to Access* form, Microsoft Access may launch and open the newly created .mdb file.
CIS/2	.STP	Exports a file using the CIMSteel Integration Stand-ards (CIS). CIS is a set of formal computing specifica-tions used in the steel industry to make software appli-cations mutually compatible. This file type is often used by steel fabricators outside the USA.
SDNF	.sdnf, .dat	Exports a file using a Steel Detailing Neutral File (SDNF). This file format is used by steel fabricators to translate steel members from models to shop draw-ings. Used primarily in the USA.
AutoCAD	.dxf	Exports a dxf file. This file format can be read by many graphics programs and is commonly used to exchange drawings between programs. The .dxf export feature is intended to facilitate exporting geometry data into Au-toCAD format compatible with AutoCAD r14, AutoCAD 2000 and AutoCAD 2002, and other .dxf compatible programs
IFC	.ifc	Exports Industry Foundation Classes (IFC) model da-ta. IFC is an object-oriented file developed to facilitate interoperability in the building industry.
IGES	.igs	Exports Initial Graphics Exchange Specification (IGES) data, which allows digital exchange of information among computer-aided design systems. IGES is a neutral exchange format for 2D and 3D models, draw-ings, and graphics.

Table 2-3 File > Export > {Command}

Command	File Extension	Description
Peform3D		Exports a text file of analysis results in a format compatible with Perform-3D, a highly focused nonlinear software tool for earthquake resistant design.
Perform3D Structure		Export a model file in a binary format compatible with Perform-3D Structure, a highly focused nonlinear software tool for earthquake resistant design

2.6 File > Batch File

A batch file is a list of CSiBridge model files. When a batch file is run, CSiBridge will open the listed model files in succession, run their analyses, and manage the analysis files (save all, save some files, or delete all files) with no action required by the user. Thus, the **File > Batch File** command is useful for running multiple models when the computer is unattended (e.g., overnight).

First use the A**nalyze > Analysis Options** command to specify that model definition and analysis results tables be automatically saved after an analysis has been run. Then, use the **File > Batch File** command to generate the analysis results of multiple model files (i.e., output tables) as well as the analysis files for those models (i.e., binary files).

2.7 File > Print

The **File > Print** command has three subcommands. Table 2-4 briefly describes these subcommands.

Table 2-4 File > Print > {Command}

Command	Description
Print Graphics	Prints the graphic displayed in the active CSiBridge window. The displayed print is sent *immediately* to the default or last used printing device (plotter, printer, and so on). It may be prudent to use the Print Setup command (see below) before using this command.

Print Tables	Displays a form similar to that shown in Figure 2-3. Use the form to specify the data to be printed and the format to be used (e.g., rich text format, text, hypertext markup language, and so on).
Print Setup	Use this command to select a default printer that will be used when the print command is activated as well as to set the size and orientation of the paper.

2.8 File > Report

Clicking the **File** > **Report** command displays a menu of subcommands: **Create Report**, **Report Setup** and **Advanced Report Writer**. The **Create Report** and **Report Setup** commands would generally be used in conjunction. That is, the **Create Report** command prints a report using the settings specified using the **Report Setup** command, including the data source, the output format, and the data types. Alternatively, the **Advanced Report Writer** command "starts from scratch," allowing the user to specify both the content and the format of a report in a more detailed process. Table 2-5 provides a brief description of the features of each command. Figure 2-4 shows the Report Setup Data form that displays when the **File** > **Report** > **Report Setup** command is used. Figure 2-5 shows the Create Custom Report form that displays when the **File** > **Report** > **Advanced Report Writer** command is used. Recall that depressing the F1 key will display context sensitive help when a form is shown in the active window.

Table 2-5 File > Report > {Command}

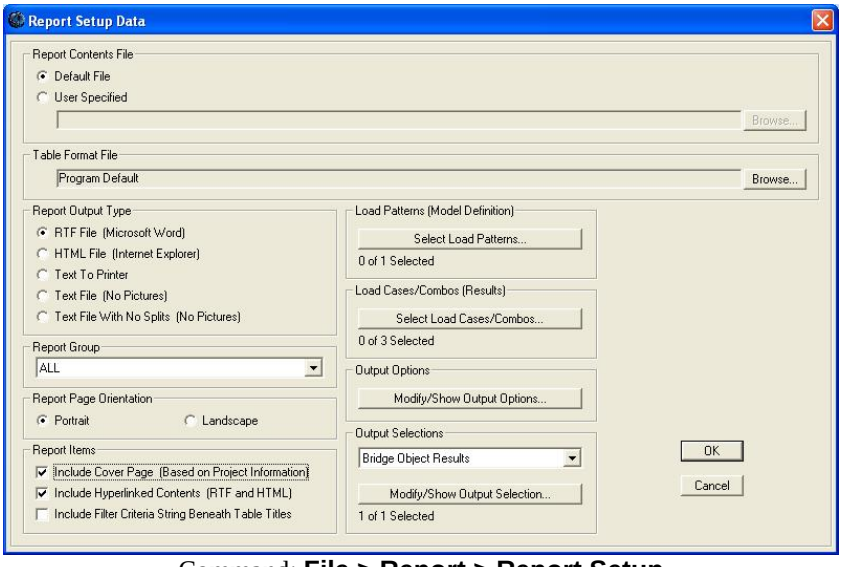
Command	Description
Create Report	Generates a report for the open file using the data source, output format, and data types selected using the **Report Setup** command.
Report Setup	Does not generate a report; opens the Report Setup form (Figure 2-4). Use that form to specify the following: ▪Report contents, as specified in an .xml file, including instruction to include a user-supplied company logo and company name on the cover page as lifted from the Project Information form (see Project Information in Table 2-7). ▪Table format file (.fmt), which specifies the database fields to be used, the width of data columns, number format (zero tolerance, number of decimal places and so on), any data filtering, data

Table 2-5 File > Report > {Command}

Command	Description
	sorting order. ▪Output type, including .rft, .html, text to printer, text without pictures, text without splits and no pictures. ▪Group(s) for which data will be included in the report (helpful in focusing the report on key components in a model) ▪Portrait or landscape page orientation.
Report Setup (continued)	▪Report components to be included: cover page using information from the Project Information form (see Project Information in Table 2-7); hyperlinked table of contents, in RTF and HTML formats only; printing of filter criteria (as specified in the table format file-see above) beneath the table title(s). ▪Data to be included: load patterns; results of selected load cases/load combos. ▪Output parameters for each load case type. ▪Name(s) of the Bridge Object for which results are to be included.
Advanced Report Writer	Displays the Create Custom Report form (Figure 2-5). Allows the user to select the content and format for the report and then creates the report in accordance with user specifications. Use the form to select the following: ▪Source file(s) for the data to be included in the report. This feature can be used to combine data from multiple sources, including the database of the open model file, data from an Excel file or a text file, and data exported from the model file into a new Access .mdb file. As each data source is selected, the other options on the form can be used to specify how the data will be presented (e.g., with or without section headings, with or without page breaks before or after). ▪Report output type, including .rtf, .txt, or .html. The user can opt to open the generated report using the appropriate program (e.g., Microsoft Word for the .rft format; default text editor for .txt) ▪Data to be included on a database-table-by-database-table basis. That is, a display area shows the data tables available from the selected source; highlight a table name and click the **Add Selected DB Table(s) to Report** button to add it to the *Items Included in Report* list. Note that after at least one item has been added to the list, the **Change Source DB** button becomes available. This button can be used to switch to another .mdb file from a different model file.

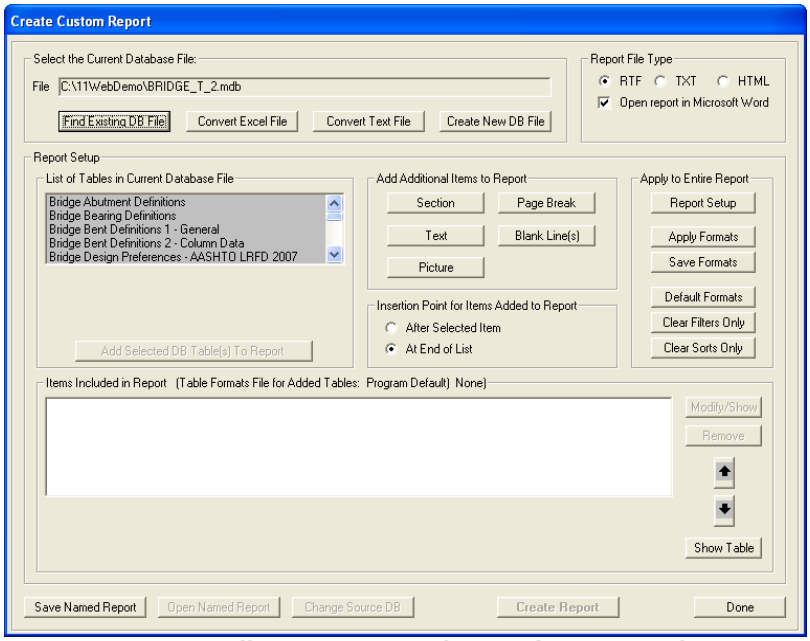
Table 2-5 File > Report > {Command}

Command	Description
Advanced Report Writer (continued)	▪Layout of the report, including levels of heads (1, 2, 3) and alignment for section headings; text of section headings; pictures/graphics and their alignment, caption, dimensions and so on; insertion of page breaks; insertion of blank lines. These items are added to the list of data items to be included in the report (see previous bullet) after a selected item or at the end of the list, depending on user selection. ▪Report setup items similar to those achieved using the **File > Report > Report Setup** command, including table formatting, filtering criteria, sorting order, hyperlinked table of contents and page orientation. Also includes page setup with respect to margins; specification of fonts for table titles, field headings, data in the tables, headings, text, figure captions, specification of the individual items to be included on a cover page. ▪Saving of the format specified using the form and also applying a previously saved format. Note that after a format file (.fmt) has been generated using this command, the .fmt file can also be used with the **Report Setup** command. ▪Removing any filter criteria that has been applied. ▪Removing any sort order that has been applied.



Command: **File > Report > Report Setup**

Figure 2-4 Report Setup Data



Command: **File > Report > Advanced Report Writer**

Figure 2-5 Create Custom Report form

2.9 File > Pictures

The **File** > **Pictures** command displays a menu of subcommands that enables capturing of screen images of the active window as bitmaps and metafiles, as well as creating multi-step videos or cyclic animations. Table 2-6 identifies the subcommands and briefly describes them.

Table 2-6 File > Pictures > Subcommands

Subcommand	Description
Bitmap - Entire Screen	Creates a bitmap (.bmp) of the entire Windows screen, including any exposed Windows wallpaper and the entire CSiBridge window, including the File, the menu bar of icons, the tabs, the display window(s) that shows the current model, and the status bars for both CSiBridge and Windows (e.g., the start button).

Table 2-6 File > Pictures > Subcommands

Subcommand	Description
Bitmap - Main Window	Creates a .bmp of the CSiBridge window, including the File, the menu bar of icons, the tabs, the display window(s) that shows the current model, and the CSiBridge status bar (e.g., XYZ coordinates, coordinate system, units).
Bitmap - Current Window with Title bar	Creates a .bmp of the active display window (where the model is shown) and the title bar along the top of the window.
Bitmap - Current Window without Title bar	Creates a .bmp of the active display window (where the model is shown) but does not include the title bar along the top of the window.
Create Multi-Step Animation Video	Saves an .avi file of the movement of the model structure after a time history analysis has been run. The saved .avi can be played using the media player supplied with the Windows program.
Create Cyclic Animation Video	Saves an .avi file of the animated mode shapes and other deformed shape plots of the model structure. The saved .avi can be played using the media player supplied with the Windows program.

2.10 File > Settings

The **File** > **Settings** command displays a menu of subcommands that can be used to set the display and output units, the tolerances, the database table utilities/settings, the display and output color settings, and other miscellaneous setting, as well as to record project information and comments and review the program-generated information log. Table 2-7 identifies the subcommands and describes them briefly. Figures 2-6 through 2-8 illustrate some of the forms that display when the subcommands of the **File** > **Settings** command are used.

Table 2-7 File > Settings > {Command}

Command	Description
Units	Specifies the number formatting to be applied in any of the databases generated by the program. This feature allows different units to be set for a given item. For example, the base units set when the model file was initialized could be Kip, in, F, which means all dimensions throughout the program are converted to Kip, in, F whenever the file is saved. This feature could be used to change the units used for lengths for one item, such as Section Dimensions, to Kip, ft., F. The Format option allows users to convert all of the units to English, Metric or Current Consistent units. Some caution is warranted here to ensure that errors related to variation in units are not made.
Tolerances	Used to set parameters applied to various program features involving proximity considerations (e.g., minimum distance for spacing fine grids; minimum distances allowed when working with the Snap To feature) and model display (e.g., minimum pixel size for line thickness, minimum point size for fonts). Please consult the context sensitive help topic for further details concerning each dimension or tolerance item (depress the F1 key when the form shown in Figure 2-7 is displayed).
Database Table Utilities and Settings	Displays a form with multiple buttons that when clicked display the forms used to manage the program database files, including: > **Set Current Format File Source** – Allows selection of the database table format file (.fmt) to be used as the basis for formatting tabular output. Options include the programmed format, an .fmt file that ships with the program, and a user specified file in the appropriate format. > **Edit Format File** – Use to make changes to the tables included a format file (.fmt). > **Set Current Table Name Source** – Use to alter database table names. > **Write Default Table Names to XML** – Use to select data tables for inclusion in a saved .xml file that subsequently can be used in generating reports. > **Documentation to Word** – Use to create a file(s) in Microsoft Word that identifies the types of data in the database, including a brief description of the function of the data, the units, the format, and so on. Caution, the All Tables file is large.

Table 2-7 File > Settings > {Command}

Command	Description
Database Table Utilities and Settings (continued)	> The *Auto Regenerate Hinges After Import* check box on this form is a toggle that when enabled (a check mark precedes the name) instructs the program to automatically regenerate any hinges in the model after data has been imported into the model from an external source.
Colors	Change the default settings for display and output colors.
Other Settings	Displays a form with options that control graphical display and some operational features of the program. > **Graphics Mode** – Choose the mode for display: GDI Plus or Direct X. GDI Plus makes two-dimensional drawing easier. Direct X is better suited for displaying full color graphics and 3D animation. > **Auto Save** – Click the **Modify/Show** button to display a form with options to specify that the model be saved automatically at specific intervals and that the emergency backup file (i.e., the .$2k text file) always be saved each time the auto save occurs. > **Auto Refresh** – Toggle to indicate if the program should refresh the model view after changes have been made to the model data. > **Show Bounding Plane** – Toggle to turn off or on the cyan-colored line that shows the location of the active plan or elevation view. For example, if a plan view is active and a 3D view is also showing, the bounding plane appears in the 3D view around the level associated with the plan view. > **Moment Diagram on Tension Side** – Toggle to plot the moment diagrams for frame elements with the positive values on the tension side of the member or on the compression side of the member. > **Sound** – Toggle to turn sound off or on when viewing animation of deformed shapes and mode shapes. > **Show Result Values While Scrolling** – Toggle to turn off or on the display of a small text box when the mouse cursor is moved over a deformed shape.
Project Information	Use to record project data that subsequently could be included in printed output tables or reports or in an exported file or an on-screen display. Data includes company name, client name, project name, project number, model name, model description, revision number, frame type, engineer, checker, supervisor, issue code, design code.

Table 2-7 File > Settings > {Command}

Command	Description
Comments and Log	Displays an up-to-date record of when and where the file was stored. The comment log may also be used to track the status of the model, to keep a "to-do" list for the model, and to retain key results that can be used to monitor the effects of changes to the model. Those notations can be deleted or modified and comments may be typed directly into the comment log at any time.

This command (see Table 2-7) displays this form.

File > Settings > Units.

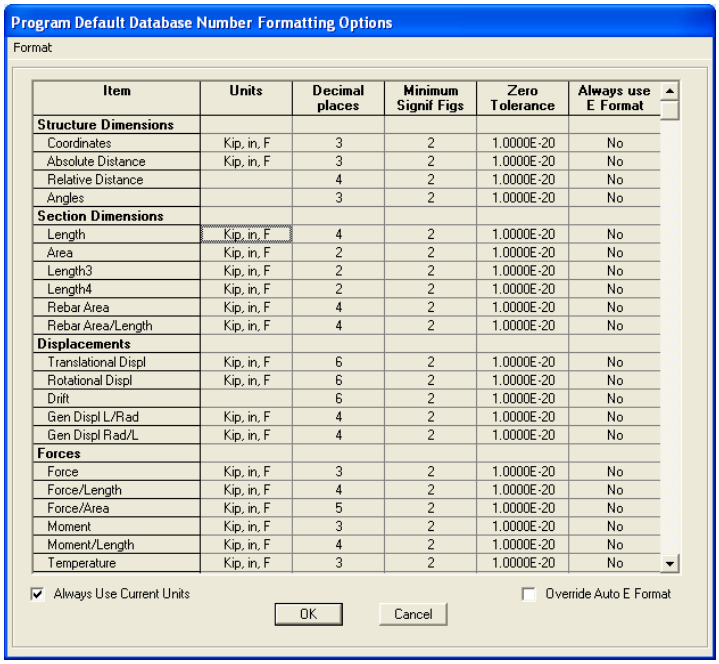


Figure 2-6 Program Default Database Number Formatting Options form

This command (see Table 2-7) displays this form.

File > Settings > Tolerances.

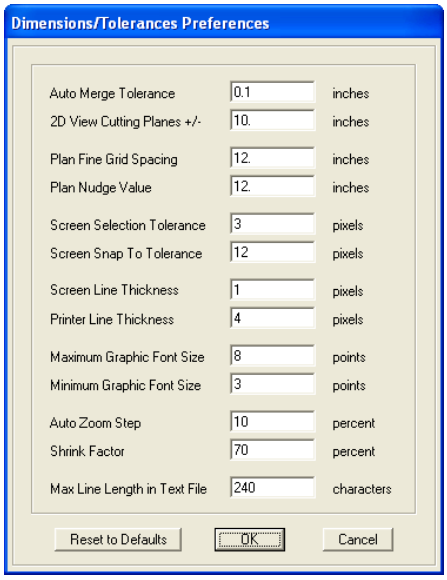


Figure 2-7 Dimensions/Tolerances Preferences

This command (see Table 2-7) displays this form.

File > Settings > Other Settings.

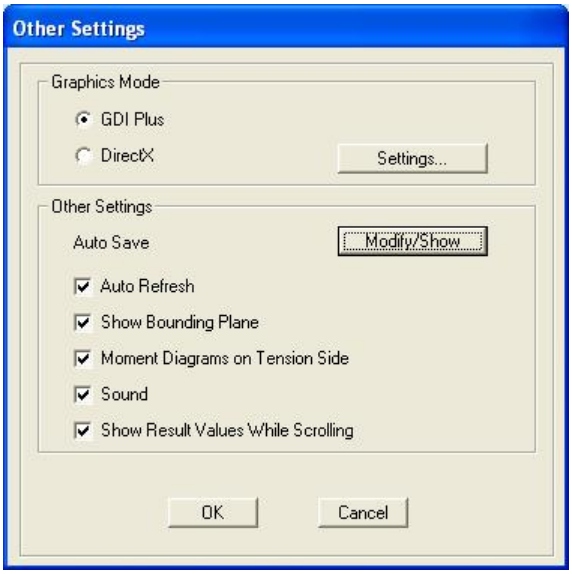


Figure 2-8 Other Settings

File > Settings > Other Settings > Settings button

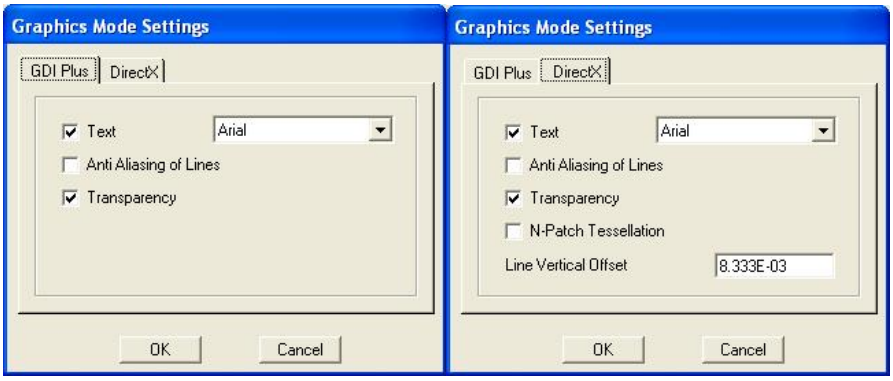


Figure 2-9 Graphics Mode Settings

File > Settings > Other Settings > Modify/Show button

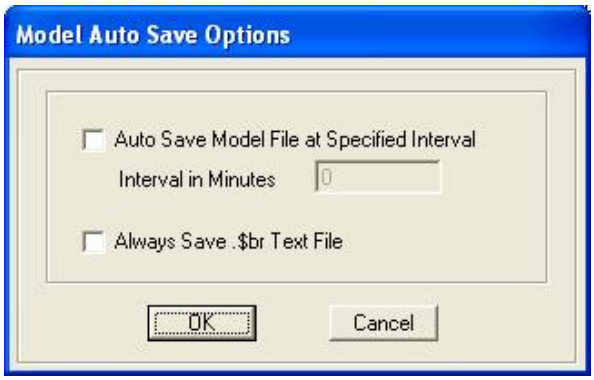


Figure 2-10 Model Auto Save Options

This command (see Table 2-7) displays this form.

File > Settings > Project Information.

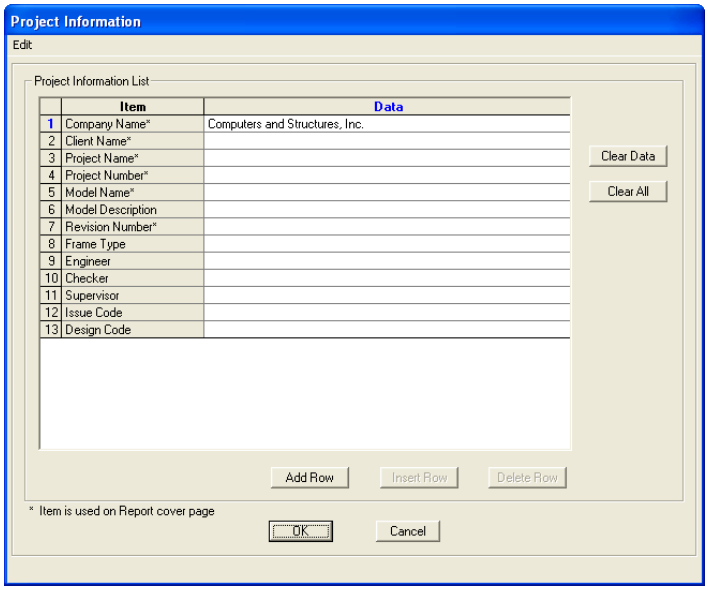


Figure 2-11 Project Information

2.11 File > Language

CSiBridge is currently available in English and Chinese. Use the **File** > **Language** command to change languages.

CHAPTER 3 Home

The **Home** tab consists of the *Wizard*, *View*, *Snap*, *Select*, and *Display* panels. The ***Bridge Wizard*** can be used to step through the modeling and analysis processes when the Quick Bridge template or the Blank option is used to start the model (see Chapter 2).

The commands on the *View*, *Snap*, *Select*, and *Display* panels can be used to manage the active view (e.g., zoom features, set 3D, XY, XZ, ZY views. and so on), improve the accuracy of operations in the active view (e.g., apply Snap tools to ensure that the end of a drawn line object connects exactly to an existing point object or grid coordinate), assist operations in the active view through targeted selection (e.g., select objects based on their material property assignment), and determine the results to be shown in the active view. Thus, the **Home** tab has the commands needed to adjust the active view and to work in it efficiently.

Each of these features and their associated commands are described briefly in this chapter.

3.1 Home > Bridge Wizard

The ***Bridge Wizard*** provides a simple and easy way to navigate through the bridge modeling and analysis processes. Unlike other program "wiz-

ards," it is possible to "pick up" and "leave" the **Bridge Wizard** at any time. Figure 3-1 shows the **Home > Bridge Wizard** command and the Bridge Modeler Wizard form that displays when this command is used. Note that the commands on the other panels have been blocked from this illustration.

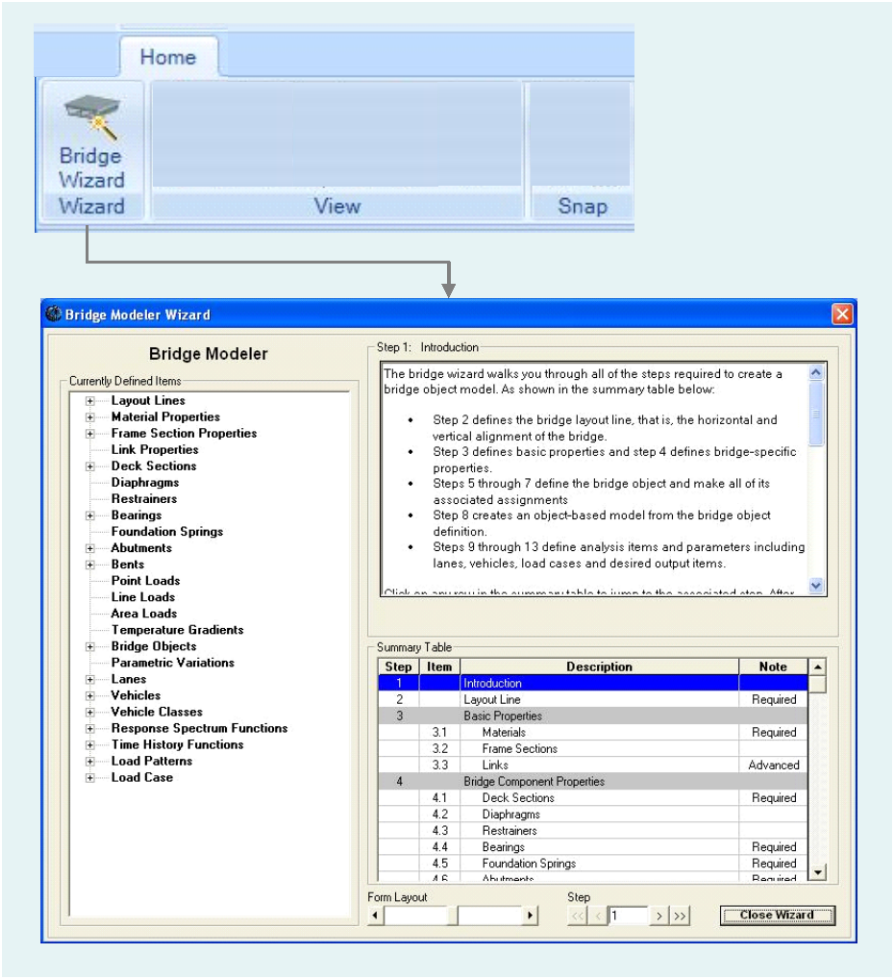


Figure 3-1 Home > Bridge Wizard command and Bridge Modeler Wizard form

Note that the tree structure on the left-hand side of the form keeps a current record of the components that have been defined for the bridge model. The informational display area in the upper right-hand side of the

form changes depending on the Step/Item/Description selected from the Summary Table in the lower right-hand side of the form. That is, the information displayed briefly explains the selected Step/Item. Clicking on an item in the tree view "jumps" the informational display and the Summary Table to the Step/Item associated with the selected tree view item.

Note the Form Layout slide bar near the center at the bottom of the form. Use that slide bar to reveal more of the information display area (slide the bar to the left) or more of the Summary Table (slide the bar to the right). Figure 3-2 shows the complete set of Steps/Items in the Summary Table area.

It is possible to move around the Summary Table area as follows:

- Click on any row to jump to that Step.
- Depress the up and down arrow keys of the keyboard to move up or down one Step at a time.
- Type a Step number into the Step control near the bottom of the form and depress the Enter key on the keyboard to jump directly to the specified Step.
- Use the Step control arrows to move to the first Step (<<), previous Step (<), next Step (>) or last Step (>>).
- The slide bar on the right-hand side of the Summary Table can be used to move down and up along the display to expose areas not shown.

The "Note" column in the Summary Table identifies some Steps as *Required* and others as *Advanced*. The Steps identified as Required must be completed to create a bridge model. The Steps designated as Advanced are those that generally are not used in a typical model. Those Steps with no designation should be used in a model at the user's discretion.

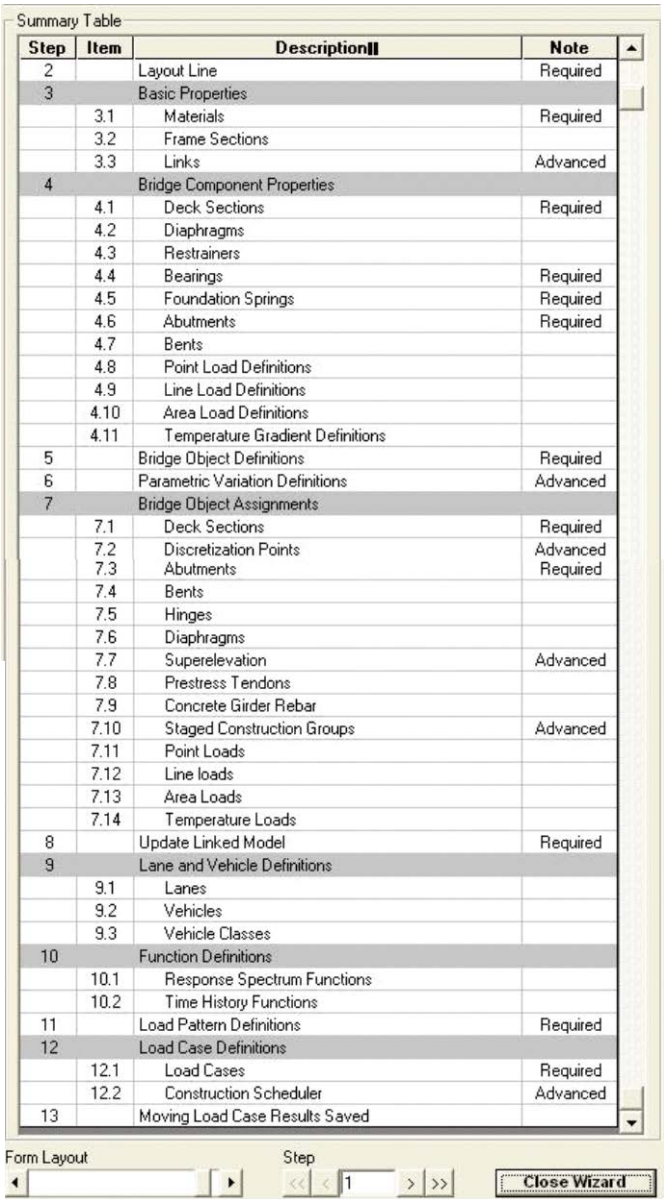


Figure 3-2 Bridge Wizard Summary Table

3.1.1 Using the Bridge Wizard

Recall from Chapter 2 that the **Bridge Wizard** is designed to be used when a model is started by clicking the Blank option or the Quick Bridge template.

- When the Blank option is used, it is possible to immediately open the **Bridge Wizard** by clicking on it on the Home tab. In that case, the **Bridge Wizard** is used to create the entire model. Thus, the first step is to define the layout line, and then continue following the Steps as outlined in the Summary Table and as explained in the information display area.

- When the Quick Bridge template is used, the span lengths and the deck section type are initially defined, before the **Bridge Wizard** becomes accessible on the Home tab. When the span lengths and deck section are specified, the program defines the layout line as well as default material property and frame section property definitions suitable for the selected deck section type. The program also defines bearings, abutments, and bents, and generates a Bridge Object, which is the backbone of the model. In generating the Bridge Object, the various definitions are assigned to the span length(s). The program also adds default definitions for lanes, vehicles, response spectrum functions, time history functions, load patterns and load cases. In this case, the **Bridge Wizard** can be used to review the default definitions, and where necessary adjust them.

In either case (i.e., starting from the Blank option or Quick Bridge template), it is possible to use the various tabs of the graphical user interface to add, modify, and delete the initial default definitions and to add further definitions, for example: link properties; diaphragms; restrainers; foundation springs; point, line, and area loads and temperature gradients.

More importantly, after the Bridge Object has been generated, the commands on the **Analysis** and **Design/Rating** tabs can be used to define the load combinations used in the analysis; complete the Design Request for superstructure and seismic design; and complete the rating request. Re-

ports also can be generated using commands on the **Design/Rating** tab, or using the **File > Report** commands.

3.1.2 Steps of the Bridge Wizard

A general overview of the Steps on the **Bridge Wizard** (see Figure 3-2) is as follows:

- Step 2 defines the bridge layout line; that is, the horizontal and vertical alignment of the bridge.
- Step 3 defines basic properties for materials, frame sections, and links (where applicable).
- Step 4 defines bridge-specific properties (deck sections, diaphragms, restrainers, bearings, foundation springs, and so on).
- Steps 5 through 7 define the bridge object and make all of its associated assignments. That is, after the geometry has been defined (i.e., the layout line definition) and the bridge components have been defined, these steps assign the definitions to the span lengths.
- Step 8 creates an object-based model from the bridge object definition.
- Steps 9 through 13 define analysis items and parameters, including lanes, vehicles, load cases, and desired output items.

For each step in the **Bridge Wizard** (except Step 1, the Introduction) a button appears immediately below the informational display area. Clicking the button opens the form associated with the Step. In a few cases the button may be disabled. This occurs when prerequisite Steps have not been completed, such as:

- A layout line and a deck section property are required before a bridge object definition.
- A bridge object definition is required before any bridge object assignments can be made.

- A layout line definition or frame objects must exist in the model before lanes can be defined.

For the Bridge Object Assignments, a Bridge Object drop-down list also displays immediately below the informational text. The assignments made using the listed Steps will be applied to the Bridge Object selected from that drop-down list.

Table 3-1 briefly describes the Steps/features of the **Bridge Wizard**.

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Layout Line	Defines the layout line, which is used for defining the horizontal and vertical alignment of the bridge and the vehicle lanes. Layout line definitions are based on stations for linear dimensions, bearings for horizontal alignment, and grades for vertical alignment. Layout lines may be straight, bent, or curved both horizontally and vertically. Horizontal curves are circular with spirals, if necessary. Vertical curves are parabolic or circular. The forms used to define the layout line are identified in *Chapter 4 Layout*.
Basic Properties	▪ **Material Properties** – Defines the material properties used in the frame section property definition and the deck section property definition. ▪ **Frame Sections** – Defines the frame section properties used in the cap beams and columns in bent property definitions, girder sections in some deck section property definitions, continuous beam sections in some abutment property definitions, and frame sections in some diaphragm property definitions. ▪ **Links** – Defines link properties that are used in restrainer property definitions, bearing property definitions, and foundation spring property definitions. In each of those property definitions, a user method of specifying the desired property, without reference to a link property, is available. In general, we recommend that you use the user method rather than specifying your own link properties. If you do use link properties, take special care to make sure the local axes are defined correctly. The forms used to define the material properties, frame sections and link properties are identified in *Chapter 5 Components.*
Bridge Component Properties	▪ **Deck Sections** – Used to define the bridge superstructure, select from various parametric deck sections, including concrete box girder, concrete flat slab, precast concrete girder and steel girder deck sections.

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Bridge Component Properties (continued)	▪ **Diaphragms** – Specify data for vertical diaphragms that span across the bridge. A diaphragm property can be solid concrete; steel X, V or K bracing; a single steel beam or steel plate. Solid concrete diaphragms are applicable only at locations where concrete superstructure deck sections exist. Steel diaphragms are applicable only at locations where steel girder superstructure deck sections exist. In area object and solid object bridge models, the diaphragms are modeled using area and solid objects, respectively. In spine models an automatically generated link object is added at each diaphragm location to represent the diaphragm mass and weight. ▪ **Restrainers** – Specify data for restrainer cables, which are used as tension ties across superstructure discontinuities. Restrainers may be assigned at abutments, hinges and at bents where the superstructure is discontinuous over the bent. When specified, the program assumes that a restrainer cable exists at each girder location. A restrainer property can be specified as a Link/Support property or it can be user defined. The user defined restrainer is recommended. The user defined restrainer is specified by a length, area and modulus of elasticity. ▪ **Bearings** – Specify data for bridge bearings, which are used in abutment, bent, and hinge assignments to the bridge object. At abutments, bearings are used in the connection between the girders and the substructure. At bents, bearings are used in the connection between the girders and the bent cap beam. At hinges, bearings are used in the connection between the girders on the two sides of the hinge. A bearing property can be specified as a Link/Support property or it can be user defined. The user defined bearing is recommended and allows each of the six degrees of freedom to be specified as fixed, free or partially restrained with a specified spring constant.

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Bridge Component Properties (continued)	▪ **Foundation Springs** – Specify data for the connection of the substructure to the ground. Foundation spring properties are used in abutment and bent property definitions. At bents, foundation springs may be used at the base of each column. In this case the foundation springs are used as point springs. At abutments, foundation springs are used as point springs for a foundation spring-type substructure, and they are used as spring properties per unit length for a continuous beam-type substructure. A foundation spring property can be specified as a Link/Support property or it can be user defined. The user defined spring is recommended. The user defined foundation spring allows each of the six degrees of freedom to be specified as fixed, free or partially restrained with a specified spring constant. For cases where the spring property is used for a continuous beam support, a factor is specified indicating the length over which the specified properties apply. ▪ **Abutments** – Specify the support conditions at the ends of the bridge. Abutment properties are used in abutment assignments to the bridge object. The abutment property allows specification of the connection between the abutment and the girders as either integral or connected to the bottom of the girders only. The abutment property also allows the abutment substructure to be specified as a series of point springs (one for each girder) or a continuously supported beam. ▪ **Bents** – Specify the geometry and section properties of the bent cap and the bent columns. They also specify the base support condition of the bent columns. Bent properties are used in abutment assignments to the bridge object. The bent property allows specification of the connection between the abutment and the girders as either integral or connected to the bottom of the girders only. The bent property also allows specification of a single bearing line (continuous superstructure) or a double bearing line (discontinuous superstructure). When double bearing lines are used, the distance from the bent location (that is specified in the bridge object definition) to each bearing line is included in the bent property.

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Bridge Component Properties (continued)	▪ **Point, Line, Area Load Definitions** – Allows definition of unique point, line, and area loads that have a user defined direction, value and location. The loads may be defined in Force or Moment. An example of a point load might be signage on the bridge structure. These loads are assigned to the bridge model using the Bridge Object (see Chapter 7). ▪ **Temperature Gradient Definitions** – Defines temperature gradient patterns over the height of the bridge superstructure for later use in bridge object temperature load assignments. Several code-specified temperature gradient definitions are available as well as user temperature gradient definitions. The forms used to define bridge component properties are identified in *Chapter 5 Components.* The forms used to define the point, line, and area loads are identified in *Chapter 6 Loads*.
Bridge Object Definition	The main component of CSiBridge, the bridge object definition includes definition of bridge spans and the following assignments: deck sections to each span; additional discretization points, including their skews, along each span; abutments, including their skews, at each end of the bridge; bents, including their skews, at each bent location; hinges, including their skews, along each span; diaphragms, including their skews, along each span; superelevations; prestress tendons; girder rebar; bridge construction groups; point, line and area loads; and temperature loads. The forms used to define bridge spans and make these assignments are identified in *Chapter 7 Bridge*. Recall that the forms used in creating most of these definitions (see previous Bridge Component Properties step) are identified in *Chapter 5 Components*, with forms associated with point, line, and area loads identified in *Chapter 6 Loads*.
Parametric Variations	Parametric Variation Definitions – Can be used to define variations in the deck section along the length of the bridge. Almost all parameters used in the parametric definition of a deck section can be specified to vary. More than one parameter can vary at the same time, if necessary. Each varying parameter can have its own unique variation. Example uses of parametric variations include varying the bridge depth and the thickness of girders and slabs along the length of the bridge. The variations may be linear, parabolic, or circular. After a variation has been defined, it can be assigned as part of the deck section assignment to bridge objects. The forms used to define a parametric variation are identified in *Chapter 5 Components.*

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Bridge Object Assignments	▪ **Deck Sections** – Allows a deck section property to be specified for each span, and variation of the superstructure along the length of the span can be assigned. ▪ **Discretization Points** – Allows users to specify points among the span where the bridge object will be discretized. Also a skew associated with a discretization point can be specified. User discretization points supplement the discretization specified when the linked model is updated. In most models, creating user specified discretization points is unnecessary. The discretization specified when the linked model is updated typically is sufficient. ▪ **Abutments** – Allows users to specify, at each end of the bridge, end skews; end diaphragm properties, if any; substructure assignment for the abutment which may be None, an abutment property, or a bent property; vertical elevation and horizontal location of the substructure; bearing property, elevation, and rotation angle of the bridge default. Note that the elevations specified for the substructure and the bearings are Global Z coordinates. ▪ **Bents** – Allows users to specify the superstructure assignment, including a diaphragm property that, for bents at superstructure discontinuities, can be specified on each side of the discontinuity along with a restrainer property, restrainer vertical elevation, and initial gap openings at the top and bottom of the superstructure; bent property orientation; vertical elevation and horizontal location of the bent; and bearing property, elevation and rotation angle from the bridge default – note that for bents at superstructure discontinuities bearings are separately specified on each side of the discontinuity. Note also that the elevations specified for the restrainer, bent, and the bearings are Global Z directions. Typically, along each bearing line there is one bearing for each girder. ▪ **Hinges** –Allow users to specify, for each hinge, the location and orientation, the bearing property, elevation, and rotation angle from the bridge default, the restrainer property and elevation, diaphragm properties before and after the hinge, initial gap openings at the top and bottom of the superstructure. Note that the elevations specified fro the bearing and restrainer are Global Z coordinates. Typically there is one bearing and one restrainer for each girder. ▪ **Diaphragms** – A diaphragm assignment includes a diaphragm location, property, and orientation. The diaphragms assigned here are in-span diaphragms. Diaphragms that occur at abutments, bents, and hinges are assigned as part of the bridge object abutment, bent and hinge assignments, respectively. Although any diaphragm property can be assigned within a span, a

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
	concrete diaphragm will be used by the program only if it occurs within a span with a concrete deck section, and a steel diaphragm will be used by the program only if it occurs within a span with a steel deck section. ▪ **Superelevation** – A superelevation assignment for a bridge object is referenced to the layout line. The superelevation is specified in percent and it indicates the rotation of the superstructure about its longitudinal axis. The superelevataion may be constant or it may vary along the bridge. In most bridge model including superelevation is probably an unnecessary refinement. ▪ **Prestress Tendons** –Tendon assignments include the location of the start and end of the tendon, the vertical and horizontal layout of the tendon, tendon section properties, loss parameters and jacking options, tendon load specified as a force or a stress, the tendon modeling options as loads or as elements. Several quick start options are available to assist in defining the layout of parabolic tendons. ▪ **Concrete Girder Rebar** – Allow users to specify rebar in the girders of concrete deck sections. The rebar is used by the program when designing the superstructure. Both transverse (shear) and longitudinal rebar can be assigned. ▪ **Staged Construction Groups** – Allow users to specify data so that the program can automatically create groups that can be used in staged construction load cases. In the assignment, a group is specified to contain certain elements of the bridge structure. such as girders between two sections along the bridge. When the linked bridge object is updated, the program automatically fills the group with the appropriate objects. ▪ **Point, Line** and **Area Loads** – Allows users to assign predefined point, line, and area loads to the bridge superstructure. ▪ **Temperature Loads** – Apply pre-defined temperature gradient loads to the superstructure. Loads may be constant temperature changes or temperature gradient changes over the height of the superstructure.
Update Linked Model	Creates the object-based model from the Bridge Object definition. Spine, area, and solid models can be created when the model is updated. The model must be updated each time the definitions are changed for the changes to take effect. The type of model can be changed at any time. This command also accesses options that allow the user to specify discretization of the object-based model. The form used to update the bridge model is identified in *Chapter 7 Bridge.*

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Lane and Vehicle Definitions	▪ **Lanes** – Must be defined to analyze a bridge model for vehicle live loads. Lanes are used in the definition of moving load type load cases and in the definition of bridge live type load patterns, which are used in static and dynamic multi-step load cases. Lanes can be defined with reference to layout lines or existing frame objects. Typically, when using CSiBridge, lanes should be defined from layout lines. Lanes can be defined with width, if desired. ▪ **Vehicles** – Must be defined to analyze a bridge model for vehicle live loads. Vehicle loads are applied to the structure through lanes. Numerous standard vehicles are built into the program. In addition, the general vehicle feature allows creation of customized vehicle definitions. Each vehicle definition consists of one or more concentrated or uniform loads. ▪ **Vehicle Classes** – Must be defined to analyze a bridge model for vehicle live loads using a moving load load case. A vehicle class is simply a group of one or more vehicles for which a moving load analysis is performed (one vehicle at a time). The forms used to define lanes, vehicles, and vehicle classes are identified in *Chapter 4 Layout*.
Function Definitions	▪ **Response Spectrum Functions** – Required for creating response spectrum load cases. If a response spectrum analysis is to be performed for a bridge model, use this step to define the function. Many standard response spectrum functions are built into the program. In addition, the user function feature creates user-defined functions, and the function from file feature obtains a function definition from an external file. ▪ **Time History Functions** – Required for creating time history load cases. If a time history analysis is to be performed for a bridge model, use this step to define the required functions. Some generic time history functions are built into the program. In addition, the user function feature can be used to create user-defined functions, and the function from file feature obtains a function definition from an external file. The forms used to create these function definitions are identified in *Chapter 6 Load*.

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Load Pattern Definitions	A load pattern is a specified spatial distribution of forces, displacements, temperatures, and other effects that act upon the structure. A load pattern by itself does not cause any response in the structure. Load patterns must be applied in load cases in order to produce results. One special type of load pattern available in CSiBridge is the Bridge Live load pattern. In this type of load pattern, one or more vehicles that move across the bridge are specified. For each vehicle, a time is specified for the vehicle to start loading the bridge, the initial vehicle location, the direction of travel and the speed. When used in a multi-step static or multi-step dynamic (direct integration time history) load case, this type of load pattern is useful in evaluating special vehicle loads. The forms used to define a load pattern are identified in *Chapter 6 Loads*.
Load Case Definitions	▪ **Load Cases** – Defines how loads are to be applied to the structure (e.g., statically or dynamically), how the structure responds (e.g., linearly or nonlinearly), and how the analysis is to be performed (e.g., modally or by direct-integration). Any load case type can be used when analyzing a bridge model. For seismic analysis, static, response spectrum and time history load case types are useful. Pushover analysis can be performed using a nonlinear static load case. Staged construction analysis is also performed using nonlinear static load cases. Several analysis options are available that are specialized for analysis of vehicle live loads. Moving load cases compute influence lines for various quantities and solve all permutations of lane loading to obtain the maximum and minimum response quantities. Multi-step static and multi-step dynamic (direct integration time history) load cases can be used to analyze one or more vehicles moving across the bridge at a specified speed. These multi-step load cases are defined using special bridge live load patterns that define the direction, starting time, and speed of vehicles moving along lanes (see previous Load Pattern Definition item). ▪ **Construction Scheduler** – Useful in performing staged construction analysis, the Construction Scheduler automatically creates the required stage construction load cases to analyze the bridge based on the specified schedule. Use the schedule to identify tasks, their durations, tasks that must be completed before others (i.e., predecessors), and “Operations” that specify the effects of the tasks on structure development with respect to staged construction analysis. The forms used to define the load case and to schedule construction are identified in *Chapter 8 Analysis*.

Table 3-1 Home > Bridge Wizard > {Step}

Step	Description
Moving Load Case Results Saved	Allows explicit specification of the analysis results to be produced for a moving load load case. This feature reduces the computationally intensive and time consuming nature of the analysis of moving load load cases, particularly for larger models. The form used to specify the results to be saved is identified in *Chapter 8*.

Note that *Chapter 8 Analysis* and *Chapter 9 Design/Rating* describe the additional tasks/steps required to complete definition of the load combinations to be used in the design; the design request for superstructure design and seismic design using the identified load combinations; and the rating request required to obtain the rating for the bridge model. Report generation is also covered in *Chapter 9 Design/Rating* and in *Chapter 2 File*.

3.2 Home > View

Viewing a model is controlled using **Home > View > {Command}**. Figure 3-3 shows the **Home > View** commands; the Wizard and Snap commands have been blocked from this image. Table 3-2 provides a brief explanation of the action of each command. Most of the view commands provide an immediate response when clicked. In some cases an additional form appears.

Figure 3-3 Home > View commands

Table 3-2 Home > View > {Command}

Command	Action
Rubber Band Zoom	Immediately zooms in on the portion of the model that lies within a drawn rubber band window.
Restore Full View	Immediately restores the view such that the entire model fills the window.
Restore Previous Zoom	Immediately returns the view one step back to the previous zoom setting.
Zoom In One Step	Immediately zooms the view in one step.
Zoom Out One Step	Immediately zooms the view out one step.
Pan	Immediately allows the model to be moved around in the active window.
Set Default 3D View	Immediately sets the view to the default 3-D View.
Set XY View	Immediately sets the view to the default XY View.
Set XZ View	Immediately sets the view to the default XZ View.
Set Plan View	Immediately sets the view to the default YZ View.
Rotate 3D View	Only active when a 3D view of the model is being displayed, immediately allows the model to be rotated in any direction. The model is rotated about a point defined by clicking on the screen to begin the model rotation. After the mouse button has been released, the command must be reused to enable further rotation.
Perspective Toggle	Immediately turns the perspective view effects on and off. If a model is viewed in elevation or plan and the Perspective Toggled is activated, the view will change from a planer view to a 3D perspective view.
Refresh Window	Immediately redraws the window without rescaling.
Object Shrink	Graphically toggles the size of object between a smaller size and the original size. The shrink factor is specified using the **File > Settings > Tolerances** command. This is a graphical change only. No changes in member connectivity or size are made to the analytical model. This feature is particularly useful to see how members are divided or subdivided.
Set Display Options	Display a form that can be used to control display, such as which objects, labels, and property identifiers are displayed, along with how they are displayed, e.g., show extrusions.

Table 3-2 Home > View > {Command}

Command	Action
Set Limits	Displays a form that can be used to define the portion of a model to be displayed. This is especially useful with large models. The view limits are defined by specifying X, Y and Z coordinate ranges. Portions of a model located outside of the specified X, Y and Z range limits are not displayed.
More > Set 3D View	Can be used to define the view angles for the plan, elevation and aperture.
More > Set 2DView	Can be used to define the View Plane as YZ, XZ or XY.
More > Show Grid	Toggles the display of grids in view on and off.
More > Show Axes	Toggles the display of the global axes in a view on and off.
More > Show Selection Only	Use to view the features of a model that have been selected.
More > Invert View Selection	Deselects all of the selected items and selects all of the previously unselected items.
More > Remove Selection from View	Use to view the features of a model that are not selected.
More > Show All	Restores the view to include all features of the model set to be viewed as defined using the **Set Display Options** command.
More > Refresh View	Reactivates the display of the current model view.

3.3 Home > Snap

Various snap commands allow the user to position the select arrow on the desired point, line, area, or solid object. Figure 3-4 shows the **Home > Snap** commands; the **Wizard** and **View** commands have been blocked from this image. Table 3-3 provides a brief explanation of the actions of each command.

Figure 3-4 Home > Snap commands

Table 3-3 Home > Snap > {Command}

Command	Action
Snap to points	Finds and snaps to the points closest to the mouse pointer.
Snap to Perpendicular	Snaps to the intersection point of a line drawn from the last entered point perpendicular to the frame objects or area edge closest to the mouse pointer. This is a helpful way to make sure that lines are perpendicular to each other.
Snap to Ends and Midpoints	Finds and snaps to the closest midpoint or end of frames and shells. It also will snap to the end points of NL Link elements.
Snap to Lines and Edges	Finds and snaps or "hugs" the closest frame object, grid line or edge of the closest area object. Even though this does not provide the same level of accuracy as the other snap options, it is a good way to make sure that the object being drawn is located on the object being "hugged."
Snap to Intersections	Finds and snaps to the intersection of two frame objects and a frame object with an area object, regardless of whether there is a joint at the intersection location.
Snap to Fine Grid	Finds and snaps to the intersections of the fine grids in a planar view. The Fine Grid spacing may be controlled using the **File > Settings > Tolerances** command.

3.4 Home > Select

With the program in Select mode, objects can be selected graphically in the active display window using the mouse cursor. Selected objects are indicated by dashed lines. Figure 3-5 shows the **Home** > **Select** commands; the **Home** tab has been moved so that it would be included in the graphic and the **Display** commands have been blocked from this image. Table 3-4 provides a brief explanation of the action of the **Select** commands.

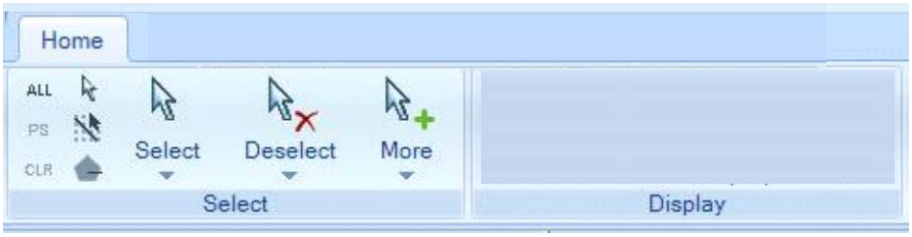


Figure 3-5 Home > Select commands

Table 3-4 Home > Select > {Command}

Command/Button	Action
Select All ALL	Selects all of the objects in the model. The total number of objects selected is displayed on the status bar. Be careful using this command. The command does not select only the objects shown in a particular window, but rather it literally selects all objects in the model.
Select Pointer/ Window	**By Pointer** – Selects/deselects objects with each mouse click on the object. Only a single object at a time can be selected/ deselected. When in Select mode, selection/deselection By Pointer is always available. An object remains selected until it is deselected or the **Clear Selection** command (see below) is used. Use the Selection List when multiple objects are present within the screen selection tolerance of the mouse click. **By Window** - Draw a window around one or more objects to select them. Drag the mouse over the object(s) to be selected.
Get Previous Selection PS	Use to reselect the previously selected objects.
Select Using Intersecting Lines	Click this command and draw a line through one or more objects to select them.
Clear Selection CLR	Clears the selection of all currently selected objects.
Select Using Polygon	Use the Select Using Polygon command to select objects by enclosing them within a polygon shape.
Select	> **Pointer/Window** – See previous description for Select Pointer/Window.

Table 3-4 Home > Select > {Command}

Command/Button	Action
	> **Poly** - Selects objects using left mouse clicks to define a poly shape around the object to be selected. > **Intersecting Poly** – Similar to poly selection with the additional selection of any objects intersected by the "lines" between mouse clicks that define the poly shape. > **Intersecting Line** – Selects objects by dragging the mouse across objects to "draw" a "selection line." > **Coordinate Specification** ▫ **3D Box** – Works in a 3D View to select objects giving full consideration to connections among points, frames, cables, tendons, areas, or solids. ▫ **Specified Coordinate Range** – Displays a form that can be used to select objects based on specification of X, Y and Z coordinates. ▫ **XY Plane** - Use to select all objects in the XY plane. ▫ **XZ Plane** - Use to select all objects in the XZ plane. ▫ **YZ Plane** - Use to select all objects in the YZ plane. > **Select Lines Parallel To** ▫ **Click Straight Line Object** – Select all line objects parallel to the line object selected after the command is used. ▫ **Coordinate Axes or Plane** – Displays a form that can be used to specify when line objects will be selected on the basis of their orientation relative to specified coordinate axes and planes, or line or area objects. > **Properties** – Material Properties, Frame Sections, Cable Properties, Tendon Properties, Area Sections, Solid Properties, Link Properties, Frequency Dependent Link Properties – Selects objects with the specified property or section.

Table 3-4 Home > Select > {Command}

Command/Button	Action
	> **Assignments** ▫ **Joint Supports** - Selects objects based on joint support assignments. ▫ **Joint Constraints** - Selects objects based on joint constraint assignments. > **Groups** - Selects objects based on group assignments. > **Labels** - Selects objects based on their labels. Can be used to select a single object or multiple objects by specifying a labeling increment or by selecting from a list of object labels. > **All** - See previous description of the **Select All** command.
Deselect	Offers the same options as the Select command (see above), but is used to deselect the specified object(s) using the chosen method.
More	> **Select Using Tables** – Displays a form that has options to select objects by selecting table names. > **Invert Selection** – Selects objects that are not selected and deselects objects that are selected.

3.5 Home > Display

The display command controls what and how various features of a model are to be displayed in the active window. Figure 3-6 shows the **Home > Display** commands; the **Home** tab has been moved so that it would be included in the graphic and the **Select** commands have been blocked from this image. Table 3-5 provides a brief explanation of the actions of the various **Home > Display** commands.

Figure 3-6 Home > Display commands

Table 3-5 Home > Display > {Command}

Command	Action
Show Undeformed Shape	Displays the undeformed shape of the model. Additionally, if a model is being created, it clears the display of any assignments that are showing on the model.
Show Bridge Superstructure Forces/Stress	After analysis, displays the force/stress component selected from a drop-down list. When viewing forces, the entire bridge deck or individual girders may be viewed. Tabular data is also available for viewing or exporting.
Show Deformed Shape	After analysis, use to display the Deformed Shape form, which has options to tailor the display, such as the load case or load combination to be displayed; scaling options; contour options; wire shadow or cubic curve.
Show Shell Force/Stress Plots	After analysis, displays internal area object forces and stresses on shell elements. The internal shell element forces are forces-per-unit-length acting along the mid-surface of the shell element (area object). The internal shell element stresses are stresses acting on the edges (not positive 3-axis face and negative 3-axis face) of the shell element (area object).
Show Bridge Loads	Displays a form with options to select a load pattern; show loads for all Bridge Objects or for a selected Bridge Object; show the loads as force or moment; show point, line, and area loads; and display area loads as pressures or discretized line loads.
Show Bridge Superstructure Design Results	Use to view design results and unity checks (demand/ capacity ratios). Results may also be viewed in Excel spreadsheets or exported.

Table 3-5 Home > Display > {Command}

Command	Action
Show Joint Reaction Forces 	Show support reactions as forces acting on the elements connected to the support. The reaction forces are reported with respect to the global coordinate system. Positive support reaction forces act in the same direction as the positive global axes. The sense of positive moments can be determined using the right-hand rule. As an example, consider a bridge bent column that is supporting gravity load. This gravity load acts in a downward direction. Thus the force imposed on the bottom of the column acts in an upward direction. This is the reaction force reported by CSiBridge. Since the upward force is in the same direction as the positive global Z-axis, the reaction is reported as a positive value acting in the Z direction.
Show Solid Stress Plots 	Displays solid member stresses.
Show tables 	Display the Choose Tables for Display form. Check the check box(es) associated with the item(s) to be displayed.
Show Influence Lines/ Surfaces 	Displays influence lines for the various displacements, reactions, forces, moments, shears, and torsion or axial loads on joints, frames, shells, planes, asolids, solids, and links resulting from a unit load on a defined bridge lane in the structure.
Show Frame/Cable/ Tendon Force Diagram 	Displays column, beam, brace, or cable forces directly on the model.
Show Link Force Diagram 	Displays link force diagrams.
Named Display	> **Save Named Display/View** –Use to save the current display and view settings. Display settings may include, for example, the display of forces or stresses after an analysis. View settings show, for example, a 2D elevation view rotated 90 degrees from default. > **Show Named Display** -- Use to display a named display of a model.

Table 3-5 Home > Display > {Command}

Command	Action
Named Display (continued)	> **Show Named View** – Use to display a named view of a model.
More	> **Show Load Assignments** – Joint, Frame/Cable/ Tendon, Area, Solid, Link – Displays forms to specify the load pattern and other parameters associated with displaying loads for the selected object type. > **Show Miscellaneous Assignments** – Joint, Frame/ Cable/Tendon, Area, Solid, Link – Displays forms to select the assignment to be displayed for the selected object type. > **Show Lanes** – Shows the lane centerline or the lane width. In addition, options may be used to display the lane loading points used by the program in a Moving Load load case and the connections of the lane loading points to the joints in the analysis model. > **Show Plot Functions** – Displays a form that can be used to specify the load case and the function to be displayed. > **Show Static Pushover Curve** – Displays a form that can be used to specify the static nonlinear case for which the pushover curve is being displayed. > **Show Hinge Results** – Displays a plot of the plastic rotation vs. moment, plastic deformation vs. shear force, and plastic moment vs. axial force for a selected hinge. > **Show Response Spectrum Curves** – Displays a response spectrum curve for a selected time history case. > **Show Virtual Work Diagrams** – Displays virtual work diagrams, which show the percentage of virtual work of an element relative to the balance of the structural members. This display can be used to reduce structural deflection by indicating which elements have the highest percentage of energy and thus will most affect the deflection if their stiffnesses were to be modified.

Table 3-5 Home > Display > {Command}

Command	Action
More (continued)	> **Show Plane Stress Plots** – Displays sress contours for planes in the active window using stress averaging. > **Show Asolid Stress Plots** – Displays stress contours for asolids in the active window using stress averaging. > **Show Input/Log Files** – Displays input and output text files.

CHAPTER 4 Layout

The **Layout** tab consists of commands that allow efficient access to the data forms needed to add, copy, or modify layout line and lane definitions. A delete command also is available for deleting a selected definition. A method of displaying the definition form that lists all definitions and that has buttons that perform the same functions as the commands on the **Layout** tab also is available. This chapter describes those data and definition forms.

If the Quick Bridge template was used to start the bridge model, the program will have created a default layout line and some lane definitions. These definitions can be viewed using the commands on the **Layout** tab.

If the ***Bridge Wizard*** is being used, highlighting the Layout Line and Lanes items in the Summary Table and clicking the **Define/Show Layout Lines** and **Define/Show Lanes** buttons will display the Define Bridge Layout Line form and the Define Lanes form that can then be used to access the same data forms as those that can be accessed directly using the **Layout** tab commands**.**

The commands on the **Layout** tab also can be used if the Blank option was used to start the model and the **Bridge Wizard** is not being used (i.e., the model is being built “from scratch” or by importing model data).

4.1 Layout > Layout Lines

Layout lines are reference lines to which bridge span lengths are assigned. Thus, bridge alignment is defined using layout lines. The layout line defines both the vertical and horizontal bridge alignments, which may be defined as straight or curved.

Figure 4-1 shows the commands on the Layout Line panel of the Layout tab.

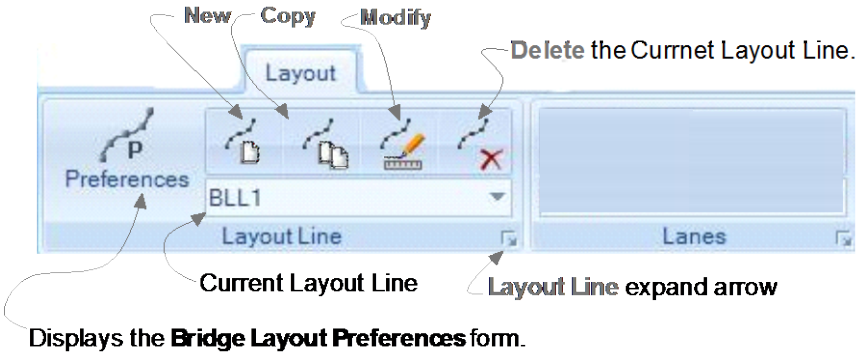


Figure 4-1 Layout tab with annotated Layout Line panel

As indicated in the figure, clicking the expand arrow in the lower right-hand corner of the Layout Line panel will display the Define Bridge Layout Line form shown in Figure 4-2. The previously defined layout lines are shown in the Layout Lines display area on the left-hand side of the form. The buttons in the "Click to:" area on the right-hand side of the form can be used to add, copy, modify, or delete layout line definitions. The buttons in the "Click to:" area display the same forms or function in the same manner as the **New**, **Copy**, **Modify** and **Delete** commands on the Layout Line panel (see Figure 4-1). In essence, the commands on the Layout Line panel by-pass the definition form shown in Figure 4-2 and provide a short cut to the data form for the layout line definition.

IMPORTANT NOTE: The preceding statement has one exception. The Define Bridge Layout Line form has an additional but-

tons, **Add New From Gen Ref Line** that does not have a corresponding command on the Layout panel of the **Layout** tab. This button is available only if a general reference line has been drawn in the model (**Advanced > Draw > More > Draw/Edit General Reference Line** command). When the **Add New From Gen Ref Line** button is clicked, the Bridge Layout Line From General Reference Line form displays (see Section 4.1.2). Use the options on the form to select the general reference line and specify the beginning of the bridge layout line and any x, y, or z offset of the layout line from the general reference line.

Use this command to display this form:

Layout > expand arrow on Layout Line panel

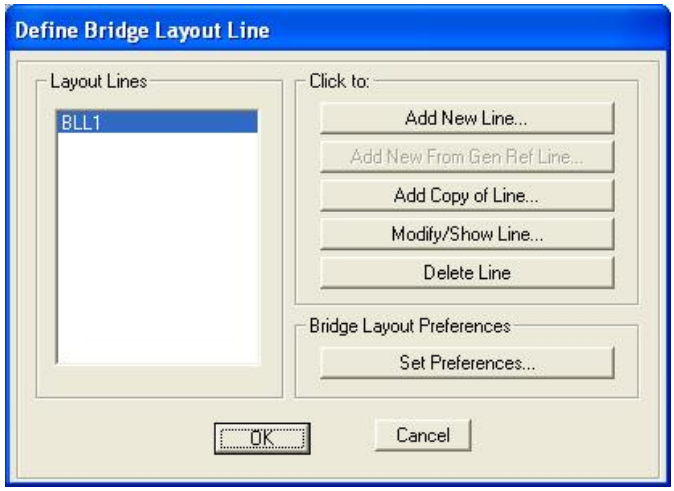


Figure 4-2 Definition form showing previously defined layout lines

Table 4-1 briefly explains the functions of the commands on the *Layout Line* panel. Screen captures of the forms identified in Table 4-1 are provided in the subsection that follows.

Table 4-1 Form Data - Layout > Layout Line >

Command	Data Form Parameters
Preferences	Displays the Bridge Layout Preferences form (see Section 4.1.1). Orient the bridge model by setting the direction of the North Arrow, in degrees, as measured counterclockwise from the positive global X axis. That is, with a typical window setup, the global X axis extends horizontally to the right. A default 90-degree orientation is

Table 4-1 Form Data - Layout > Layout Line >

Command	Data Form Parameters
	offered in the form so that the North Arrow points straight up, orienting the model. The curve discretization also may be set by defining the maximum subtended angle to be used by the program to define a bridge curve. The smaller the curve discretization value the smoother the curve. Multiple layout lines can be defined within the same model file. All layout lines within the same model comply with the North Arrow orientation and curve discretization parameters set on the Bridge Layout Preferences form.
Add	Displays the Bridge Layout Line Data form - The layout lines specify orientation of the bridge relative to the coordinate system. The placement of bridge objects (e.g., abutments, columns, bents, hinges, spans) is accomplished relative to the layout line. The initial and end station locations for the layout line, and thus the Bridge Object, are defined using this form. The form has the following buttons that provide access to additional data forms used to refine the bridge alignment. ▪ **Modify Layout Line Stations** button. Displays a form that can be used to shift the layout line, which will shift all of the stations on the bridge objects and the lanes that reference the layout line. Shifting the layout lines moves the initial station location the specified distance in the global X direction from the coordinate system origin. Note that multiple layout lines can be defined for a single bridge model. Individual layout lines can be shifted independently of each other. Thus, the bridge objects and lanes that reference a specific layout line can be shifted easily using this form. ▪ **Define Horizontal Layout Data** button. Displays a form that can be used to define the horizontal bridge layout. It is possible for a layout line to be defined using combinations of multiple straight and curved segments, including Straight at Previous Bearing to Station, Straight at New Bearing Station, Curved Right at New Bearing Station, or Curved Left at New Bearing Station. The station locations are specified as lengths, measured from left to right, from the initial station location. ▪ **Define Vertical Layout Data** button. Displays a form that can be used to define the vertical bridge layout. It is possible for a layout line to be defined using combinations of multiple grades, including Constant at Previous Grade to Station, Constant at New Grade to Station, Constant Grade to New Elevation at Station, Parabolic to New Grade at Station, and Circular to New Grade at Station. The station locations are specified as lengths,

Table 4-1 Form Data - Layout > Layout Line >

Command	Data Form Parameters
	measured from left to right, from the initial station location. ▪ **Horizontal Layout Line Data – Quick Start** button. Displays a form with a variety of template horizontal layout alignments, such as Straight – Bend Right, Straight – Curve Left, Straight – Curve Left – Straight – Curve Left – Straight, and so on. These template definitions transfer to the horizontal layout data form. As the name suggests, this from is a "quick start" for defining the horizontal layout data.
	▪ **Vertical Layout Line Data – Quick Start** button. Displays a form with a variety of template horizontal layout alignments, such as Straight – Bend Down, Straight – Bend Up, Parabolic Up, and so on. These template definitions transfer to the horizontal layout data form. As the name suggests, this form is a "quick start" for defining the vertical layout data.
Copy	Creates a copy of the layout line definition selected in the Current Layout Line drop-down list. The definition data can be modified as described for the preceding **Add** command.
Modify	Displays the layout line definition selected in the Current Layout Line drop-down list. The definition data can be modified as described for the preceding **Add** command.
Delete	Deletes the layout line definition selected in the Current Layout Line drop-down list unless the definition is being used in a Bridge Object definition (see Chapter 7). A layout line in use in a Bridge Object can not be deleted unless the Bridge Object is first deleted.

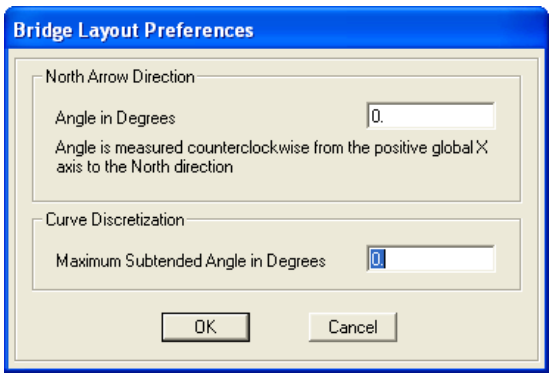
4.1.1 Bridge Layout Preferences Form – Screen Capture

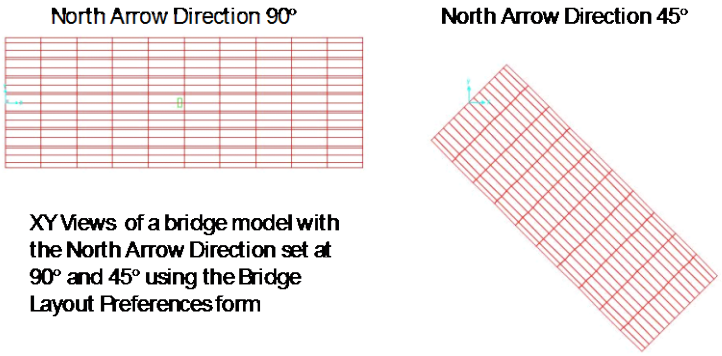
Use these commands (see Table 4-1) to display this form.

Layout > Preferences

or

Layout > expand arrow on the Layout Line panel > **Set Preferences** button



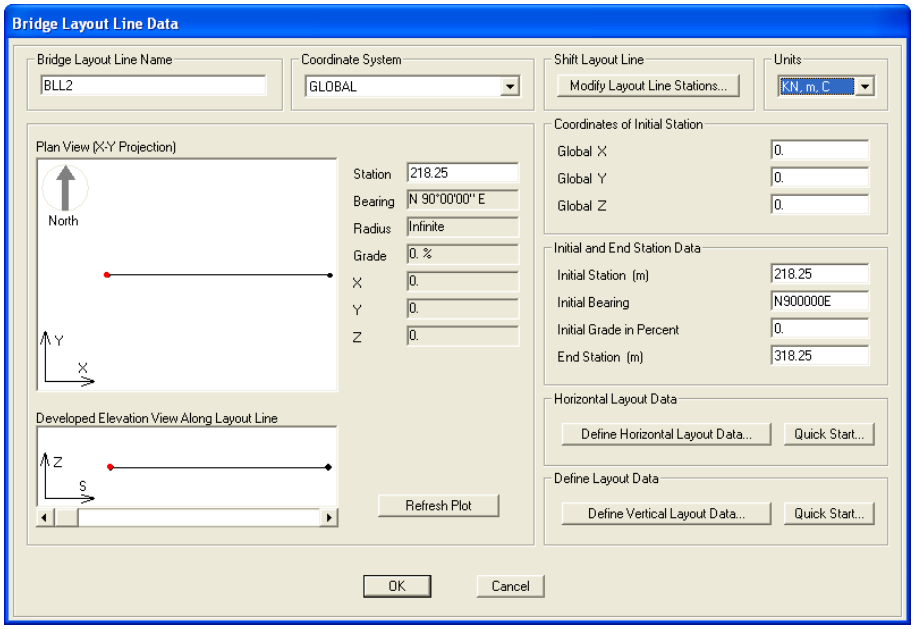


Examples of North Arrow Orientation

4.1.2 Bridge Layout Line Data Form – Screen Captures

Use these commands (see Table 4-1) to display this form:

Layout > Add Or **Layout > expand arrow** on *Layout Line* panel > **Add New Line** button

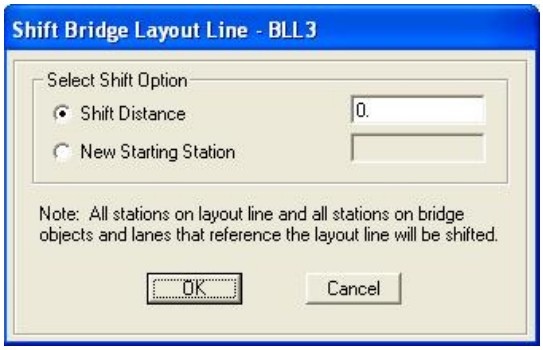


Use these commands (see Table 4-1) to display this form:

Layout > Add > Modify Layout Line Stations button

Or

Layout > expand arrow on Layout Line panel > **Add New Line** button **> Modify Layout Line Stations** button

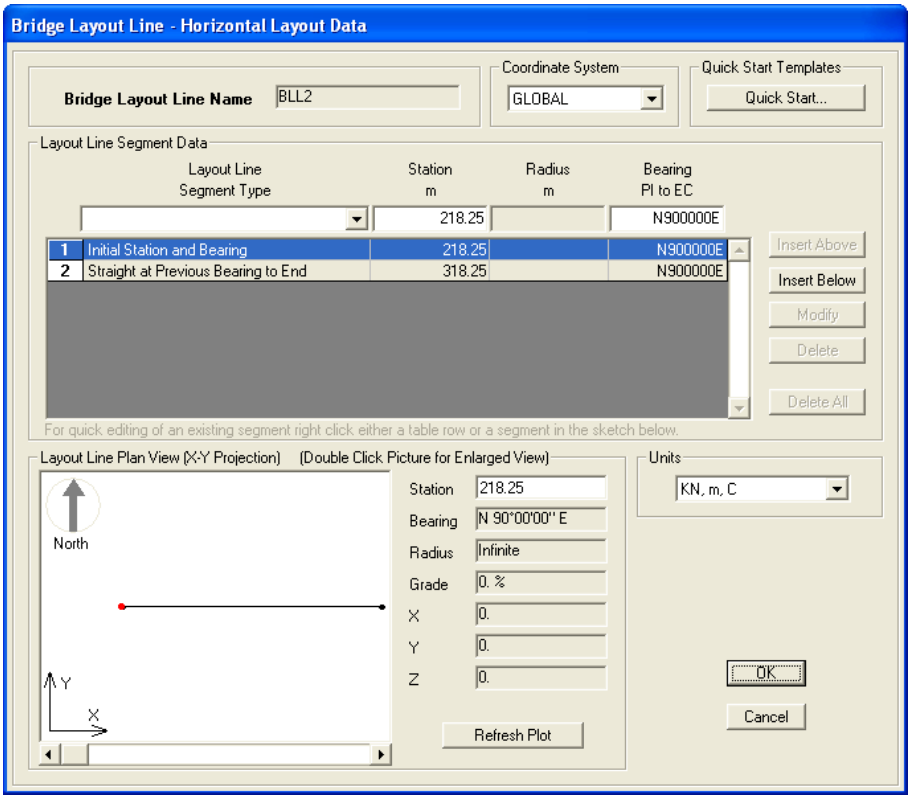


Use these commands (see Table 4-1) to display this form:

Layout > Add > Define Horizontal Layout Data button

Or

Layout > expand arrow on *Layout Line* panel > **Add New Line** button > **Define Horizontal Layout Data** button

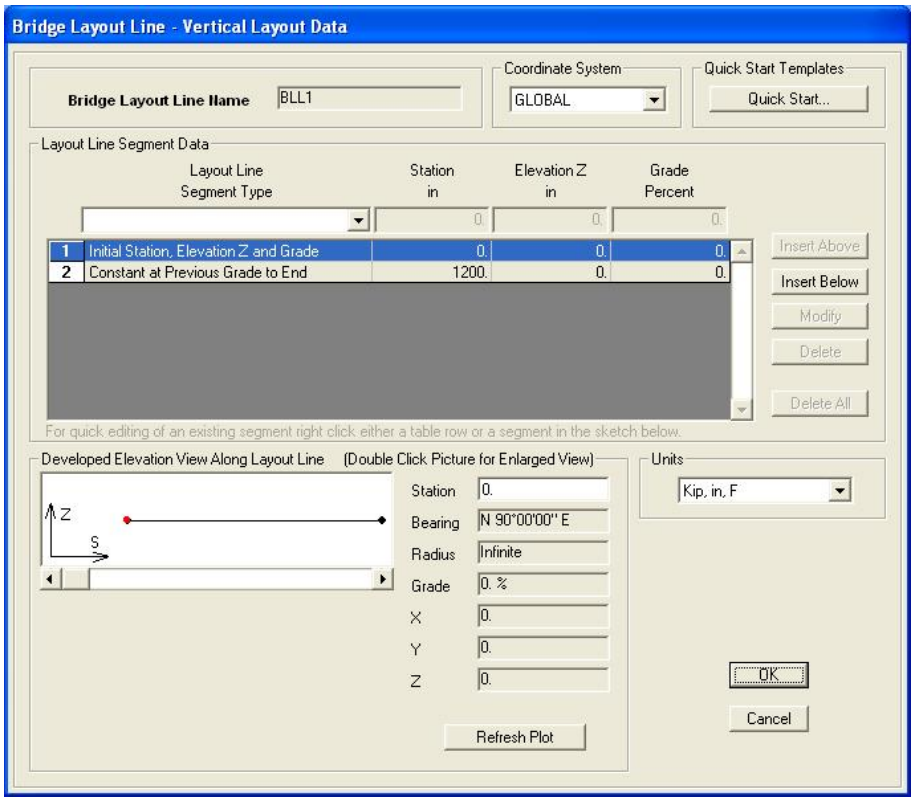


Use these commands (see Table 4-1) to display this form:

Layout > Add > Define Vertical Layout Data button

Or

Layout > expand arrow on *Layout Line* panel > **Add New Line** button > **Define Vertical Layout Data** button

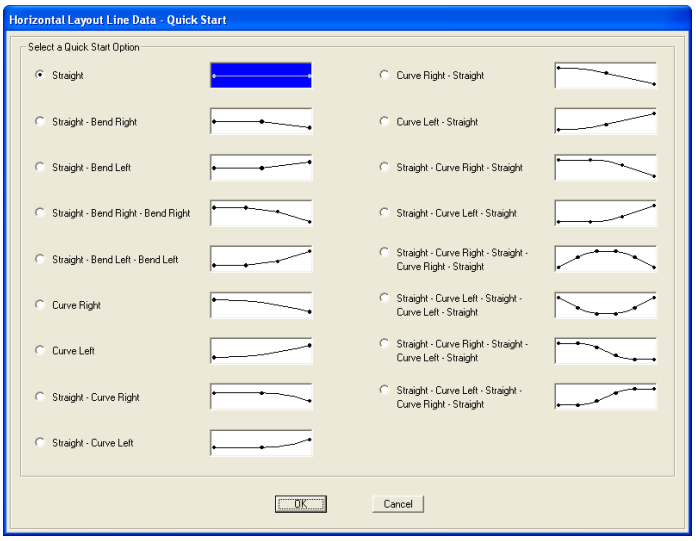


Use these commands (see Table 4-1) to display this form:

Layout > Add > Define Vertical Layout Data – Quick Start button

Or

Layout > expand arrow on *Layout Line* panel > **Add New Line** button > **Define Vertical Layout Data – Quick Start** button

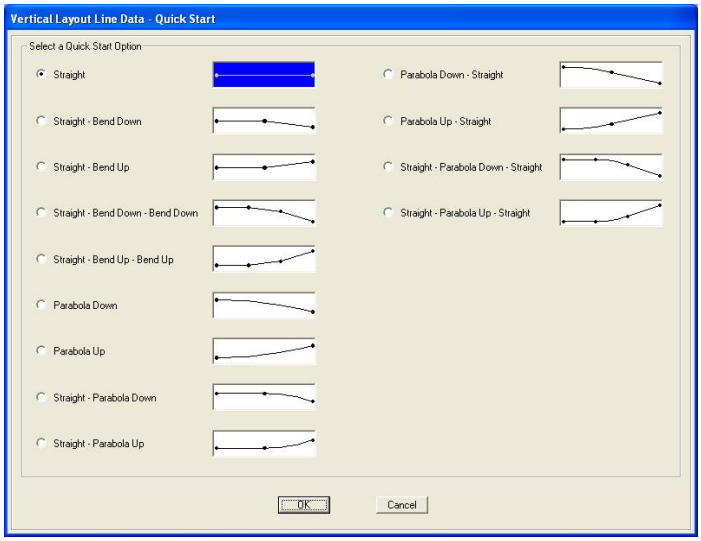


Use these commands (see Table 4-1) to display this form:

Layout > Add > Define Vertical Layout Data – Quick Start button

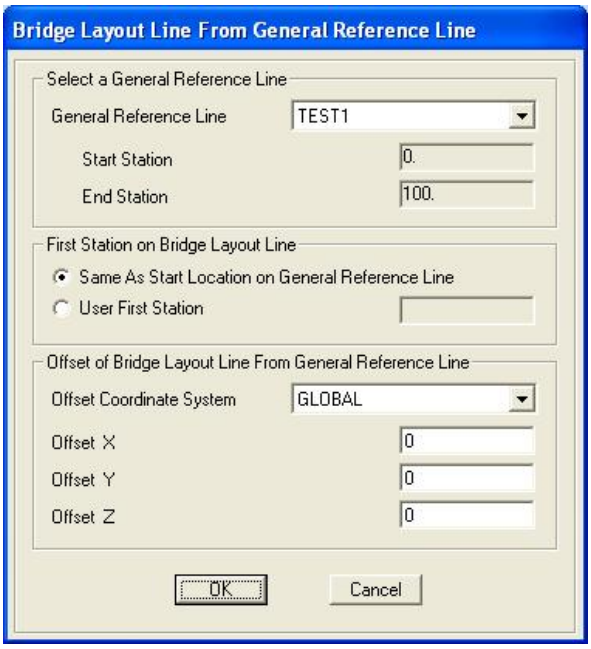
Or

Layout > expand arrow on *Layout Line* panel > **Add New Line** button > **Define Vertical Layout Data – Quick Start** button



Use this commands (see Table 4-1) to display this form:

Add a reference line to the model using the **Advanced > Draw > More > Draw/Edit General Reference Line** command; then click the **Layout > expand arrow > Add New From Gen Ref Line** button.



4.2 Layout > Lanes

Lanes must be defined to analyze a bridge model for live load. They are used in the definition of moving load type load cases (see Chapter 8) and in the definition of bridge live type load patterns (see Chapter 6), which are used in static and dynamic multi-step load cases.

Vehicle live loads are considered to act in traffic lanes transversely spaced across the bridge roadway. The number of lanes and their transverse spacing can be chosen to satisfy the appropriate design code requirements. Lanes need not be parallel or be of the same length. The number of lanes across a roadway may vary along the length to accommodate merges.

For simple bridges with a single roadway, the lanes will usually be parallel and evenly spaced, and will run the full length of the bridge structure.

For complex structures, such as interchanges, multiple roadways may be considered; those roadways can merge and split. Multiple patterns of lanes on the same roadway may be created to examine the effect of different lateral placement of vehicles. For design purposes, a single lane may be defined and loaded. The distribution of live loads to the girders may be defined using the options available within the superstructure design request definition (see Chapter 9).

Figure 4-3 shows the commands on the *Lanes* panel of the **Layout** tab

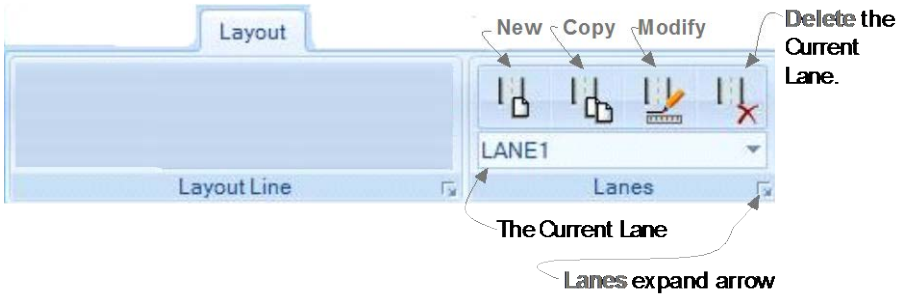


Figure 4-3 Layout tab with annotated Lane panel

Clicking the expand arrow in the lower right-hand corner of the *Lanes* panel will display the Define Lanes form shown in Figure 4-4.

The Define Lanes form shows the previously defined lanes in the Lanes display area on the left-hand side of the form. The buttons in the "Click to" area of the form can be used to add, copy, modify, and delete lane definitions. The buttons in the "Click to:" area display the same forms or function in the same manner as the **New**, **Copy**, **Modify** and **Delete** commands on the Lanes panel (see Figure 4-3). That is, most of the buttons on the Define Lanes form and the commands on the *Lanes* panel display the Bridge Lane Data form (see Section 4.2.1); or in the case of the delete buttons, delete the selected/specified lane definition. In essence, the commands on the *Lanes* panel by-pass the definition form shown in Figure 4-3 and provide a short cut to the data forms used to define a lane.

IMPORTANT NOTE: The preceding statement has one exception. The Define Lanes form has two additional buttons, **Add New Lane Defined From Frames** and **Convert Lane Definition to "From Layout Line,"** that do not have corresponding commands on the *Lanes* panel of the **Layout** tab. When the **Add New Lane Defined From Frames** button is clicked, the Lane Data form displays (see Section 4.2.2). When the **Convert Lane Definition to "From Layout Line"** button is clicked the conversion is immediate and irreversible.

Use this command to display this form:

Layout > expand arrow on *Lanes* panel

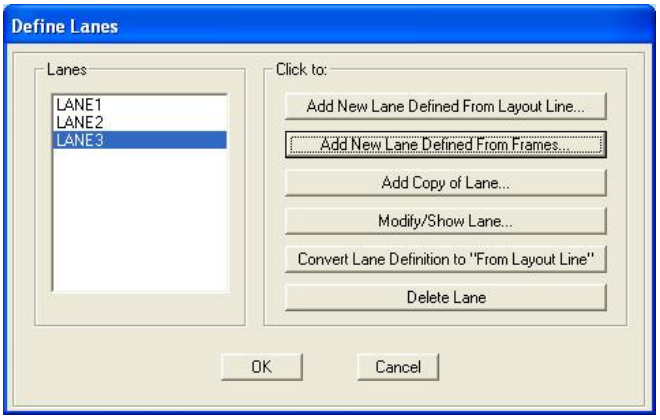


Figure 4-4 Definition form showing previously defined lanes

Table 4-2 briefly explains the **Layout > Lanes** commands.

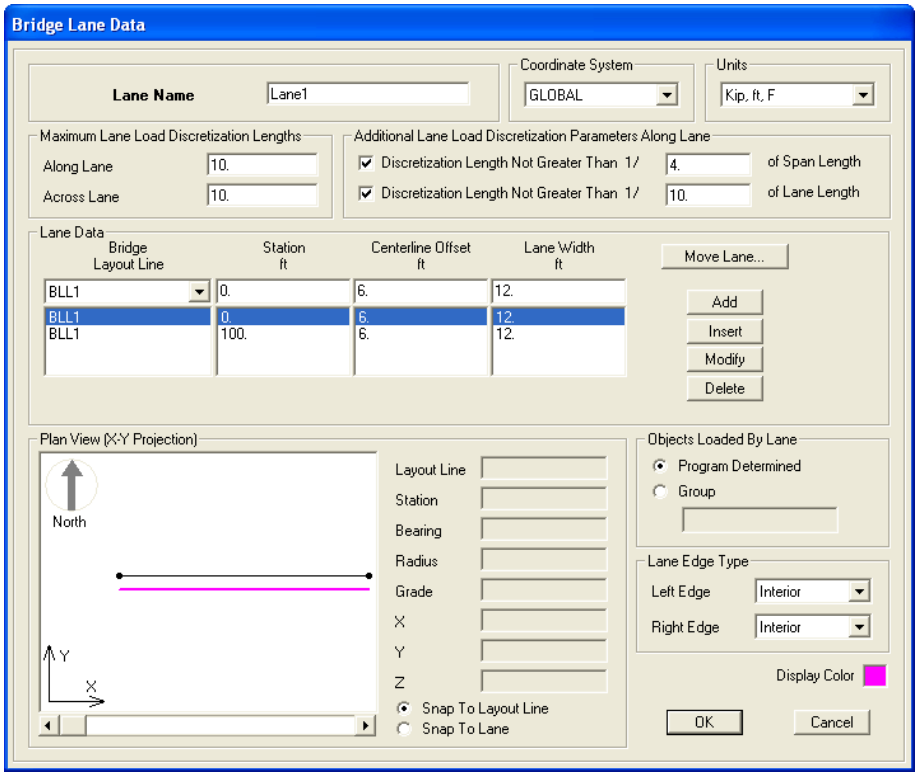
Table 4-2 Form Data - Layout > Lanes >

Command	Data Form Parameters
Add	Displays the Define Lane Data form. This form has an option to select the layout line to be used as the reference in aligning the lane (see Section 4.1). Edit boxes can be used to specify the station location measured from left to right relative to the start of the bridge, a centerline offset (relative to the layout line), and a lane width. Note that the centerline offset can be varied by station, thereby offsetting only a segment of the entire lane. The entire lane can be moved by clicking the **Move Lane** button, which will display the Move Lane form. Use that form to move the entire lane by changing the offset by a specified amount or by moving the start of the lane to a specified offset. The lane can be loaded by the program, or based on user-selected groups of model objects. Options are available to specify how the load is discretized along lanes and across them, with additional discretization possible along the span and along the lane lengths. The left and right edges of the lanes can be specified as interior or exterior. The form shows a Plan View plot of the lane, along with the bearing based on the layout line, and it shows the radius, grade, and X, Y, and Z coordinates of the mouse cursor when the cursor is moved along the lane in the plot. Two lines of data are required to define a single lane. One data line is needed to define the lane start station location and the second is needed to define the lane end station location.
Copy	Creates a copy of the lane definition selected in the Current Lane drop-down list. The definition data can be modified as described for the preceding **Add** command.
Modify	Displays the lane definition selected in the Current Lane drop-down list. The definition data can be modified as described for the preceding **Add** command.
Delete	Deletes the lane definition selected in the Current Lane drop-down list

4.2.1 Bridge Lane Data Form – Screen Capture

Use these commands (see Table 4-2) to display this form:

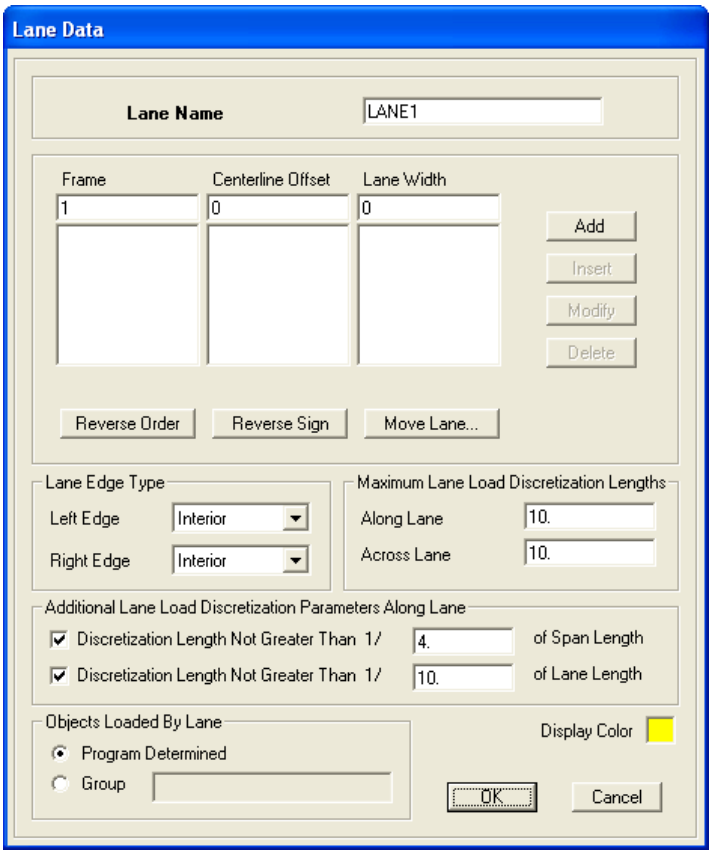
Lane > Add Or **Lane > expand arrow** on *Lanes* panel > **Add New Lane Defined From Layout Line** button



4.2.2 Lane Data Form – Screen Capture

Use this command (see Table 4-2) to display this form:

Lane > expand arrow on Lanes panel > **Add New Lane Defined From Layout Line** button **> Add New Lane Define From Frames** button



The Lane Data form has many of the same options as the Bridge Lane Data form – centerline offset, lane width, lane edge type, lane loading that is program determined or based on grouped model objects, discretization of loads along and across lanes, and discretization along span and lane lengths.

Similar to specifying the layout line to be used as the reference for the lane, the Lane Data form requires that the frame member label be input

to identify the frame member to be used to locate the lane. Multiple frame members may be used to define a single lane in the case where a particular lane is longer than a single frame member.

Note that the centerline offset applies to the entire length of the frame, not just a portion of it. Therefore, to move the entire lane, change the offset.

TIP: Use the **Home > View > Set Display Options** command to display the Display Options for Active Window form. Click the **Labels** check box in the Frames/Cable/Tendons area of the form to display frame labels.

CHAPTER 5 Components

The **Components** tab consists of the commands that allow efficient access to the *data* forms needed to add, copy, or modify definitions for material, frame, cable, tendon, and link properties (*Properties* panel); deck sections, diaphragms, and parametric variations (*Superstructure* panel); bearings, restrainers, foundation springs, abutments, and bents (*Substructure* panel). A command to specify rebar sizes is available (*Properties* panel), and a delete command is available for deleting a selected definition. A method for displaying the *definition* form that lists all definitions and that has buttons that perform the same functions as the commands on the **Components** tab also is available. This chapter identifies those *data* and *definition* forms.

If the Quick Bridge template was used to start the bridge model, the program will have created default definitions for material and frame section properties; a deck section; a bearing; an abutment; and a bent. These definitions can be viewed using the commands on the **Components** tab.

If the ***Bridge Wizard*** is being used, highlighting the Materials, Frame Sections, and Links items, the Deck Sections item, the Bearings item, the Abutments item, and the Bents item in the Summary Table and clicking the **Define/Show Material Properties**, **Define/Show Frame Sections**, **Define/Show Link Properties**, **Define/Show Deck Sections**, **Define/Show Bearings**, **Define/Show Abutments**, and **Define/Show Bents**

buttons will display the *definition* forms that can then be used to access the same *data* forms as those that can be accessed directly using the **Components** tab commands**.**

The commands on the **Components** tab also can be used if the Blank option was used to start the model and the **Bridge Wizard** is not being used (i.e., the model is being built from scratch or by importing model data).

5.1 Components > Properties

Figure 5-1 shows the commands on the *Properties* panel of the **Components** tab.

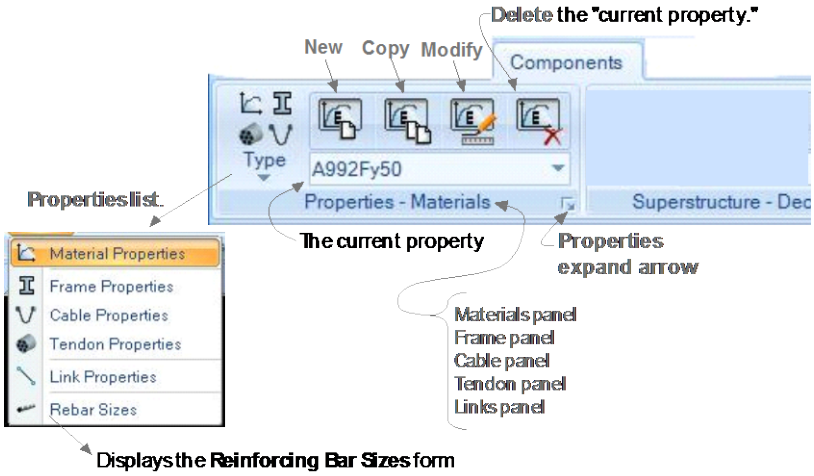


Figure 5-1 Components tab with annotated Properties panel

As suggested in Figure 5-1, clicking the **Type** command displays a drop-down list of property types (material, frame, cable, tendon, link) and rebar sizes. The name of the panel changes depending on the property type selected (i.e., panel names are *Properties – Materials*; *Properties – Frame*; *Properties – Cable*; *Properties – Tendon*; *Properties – Links*). After a property type has been selected, clicking the expand arrow displays one of the forms shown in Figure 5-2.

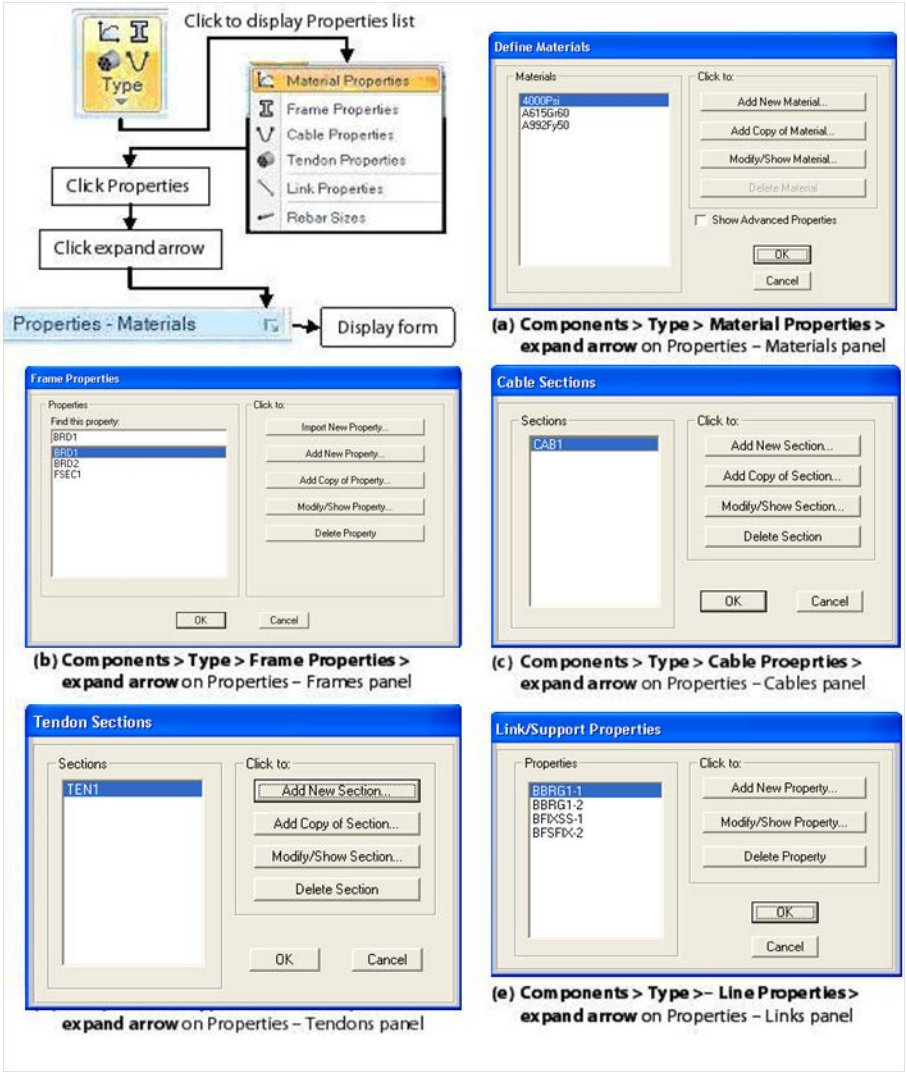


Figure 5-2 Definition forms showing previously defined properties

These forms list all previously defined property definitions in display areas on the left-hand side of the forms. Generally, the buttons in the "Click to" area on the right-hand sides of the forms display the data forms listed in the second column of Table 5-1.

Referring to Figure 5-1, the **New**, **Copy**, and **Modify** commands on the *Properties – {Type}* panel bypass the definition forms listed in column one of Table 5-1 and display the forms listed in column two, thereby creating a short cut to the data forms. The forms listed in column three generally are displayed by checking a check box (e.g., Show Advanced Material Properties) or clicking a topic-relevant button on the Data Form.

Table 5-1 Forms List - Components > Type > {Property}

Click: > Expand arrow	**> Buttons on Definitions Forms**	**> Check Box or Topic-Relevant Button on Data Forms**
Definition Forms	**Data Forms**	**Parameter Forms**
Define Materials	*Add New Material	---
	Material Property Data	▪ Material Property Options** ▫ Nonlinear Material Data ▫ Time Dependent Properties ▫ Additional Material Damping ▫ Thermal Properties
Frame Properties	***Import Frame Section Property	▪ Section Property File ▪ Data form ▪ {Section Type} Section
	Add Frame Section Property	▪ Bridge Section ▪ {Section Type} Section
Cable Sections	Cable Section Data	▪ Cable Property/Stiffness Modification Factors
Tendon Sections	Tendon Section Data	---
Link/Support Properties	Link/Support Property Data	▪ Advanced Link P-Delta Parameters ▪ {Line Type} Direction Properties

*NOTE: In the *Add Material Property* form, select the Region, Material Type, Standard and Grade that are pre-defined in the CSiMaterial-Library*.xml located in subfolder "Property Libraries" under the CSiBridge installation folder to define the new material.

NOTE: The Material Property Options form is the “gateway” to the buttons that provide access to Advanced Material Property Data (e.g., Nonlinear Material Data, Time Dependent Properties, Additional Material Damping, Thermal Properties). The Material Property Options form displays when the *Show Advanced Properties* check box on the Define Materials form is checked, or when the *Switch to Advanced Property Display* check box is checked on the Material Property Data form. In both cases, clicking the **Modify/Show Material Properties button displays a version of the Material Property Data form that includes buttons that provide ac-

cess to the forms for Advanced Material Property Data. The Material Property Options form also includes a *Material Properties are Temperature Dependent* check box that when checked displays the Temperature Dependent Material Properties form; use that form to designate temperatures at which specified parameters apply.

***NOTE: The Import Frame Section Property form can be displayed only by clicking the **Components > Type > Frame Properties > Expand arrow > Import New Property** button. Use the Import Frame Section Property form to access a database of frame sections, including a user specified database.

TIP: If the Material Property Options form displays when the **Components > Type > Material Properties > Add** or **Copy** or **Modify** command is clicked, but Advanced Material Property Data is not needed for a material definition, use the **Components > Type > Material Properties > Expand arrow** command to display the Define Materials form and uncheck the Show Advanced Properties check box. Then either click the appropriate add, copy, or modify button on the Define Materials form, or close that form and click the **Components > Type > Material Properties > Add** or **Copy** or **Modify** command to display the standard Material Property Data form.

Context-sensitive help can be accessed by displaying the form and then depressing the F1 key.

Table 5-2 provides a brief description of data that is input or included in the data forms identified in column two of Table 5-1.

Table 5-2 Form Data - Components > Properties > {Type} > Add, Copy, Modify

Type	Data Form Parameters
Material Properties	Displays the Material Property Data form. Material types may be steel, concrete, rebar, or tendon. Definitions include weight and mass, isotropic parameters (modulus of elasticity, Poisson's Ratio, coefficient of thermal expansion, and shear modulus) and other parameters (such as minimum yield stress, specified concrete compressive strength, and so on). Many pre-defined material properties of commonly used materials are included as defaults. New material properties can be added and default properties can be edited. Definitions may include advanced material property data, such as nonlinear ma-

Table 5-2 Form Data - Components > Properties > {Type} > Add, Copy, Modify

Type	Data Form Parameters
	terial data, time dependent properties, material damping properties and thermal properties. Material property definitions are used in frame section property and deck section property definitions.
Frame Properties	Property type may be steel, concrete, or other. Steel sections may be I/wide flange, channel, tee, angle, double angle, double channel, pipe, tube, or steel joist. Concrete sections may be rectangular, circular, pipe, tube, precast I or precast U. Built-up steel section may include hybrid I, hybrid U and cover plated I. Aluminum sections may include I and channel sections. Cold formed sections may include C, Z, and hat sections. "Other" may include general, nonprismatic, and Section Designer sections, which can be used to define rebar layouts for both longitudinal reinforcing and transverse reinforcement (see Section 5.1.2). Definitions include materials (see preceding Material Properties), dimensions, section properties (e.g., torsional constant, moment of inertia, section modulus, radius of gyration and so on), and stiffness modification factors. Concrete sections may be based on standard sections (e.g., WSDOT Standard U Girder U54G4). Frame property definitions are used in cap beams and columns in bent property definitions, girder sections in some deck section property definitions, continuous beam sections in some abutment property definitions, and frame sections in some diaphragm property definitions.
Cable Properties	Definition includes material property and specification of the cable diameter or the cable area. The program calculates the torsional constant, moment of inertia and shear area based on the cable diameter or area. Stiffness modification factors may also be specified (cross-sectional [axial] area; mass, weight). Cable property definitions are an advanced feature available for use if cables are added to the model.
Tendon Properties	Tendons can be modeled as loads or as elements. Definition includes material property and specification of the tendon diameter or area. The program calculates the torsional constant, moment of inertia and shear area based on the tendon diameter or area. Tendon property definitions are used in prestress tendon definitions as part of the Bridge Object definition (see Chapter 7).
Link Properties	Definition includes link/support type (linear, multilinear elastic, multilinear plastic, damper, gap, hook, plastic [wren] and rubber isolator); mass and weight; rotational inertia; factors for line, area, and solid springs; directional properties; and advanced P-Delta parameters. Link property definitions can be used in restrainer, bearing, and foundation spring property definitions. Using links in lieu of bearings and foundation springs gives a wider range of modeling options. Support bearings at abutments and bents may be defined as isolators using links. P-Y soil springs may also be represented as links.

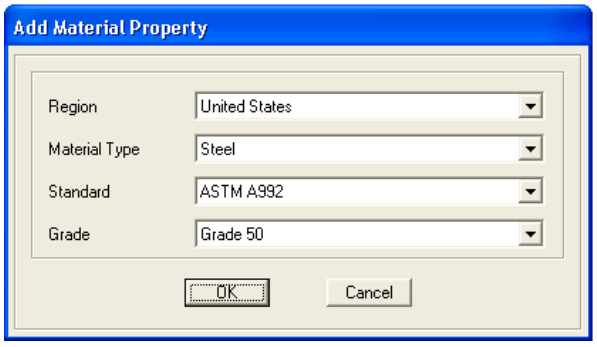
Table 5-2 Form Data - Components > Properties > {Type} > Add, Copy, Modify

Type	Data Form Parameters
Rebar Sizes	Displays the Reinforcing Bar Sizes form. The form can be used to add, modify, or delete reinforcing bar. The definition for a rebar includes the bar ID, the bar area, and the bar diameter. Rebar size definitions may be used in specifying the bridge girder deck reinforcement as part of the Bridge Object definition (see Chapter 7).

5.1.1 Material Properties Forms – Screen Captures

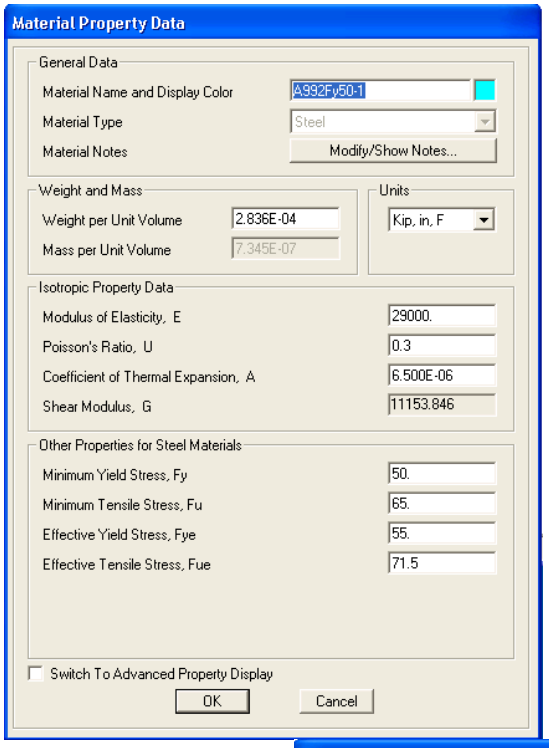
This command (see Table 5-2) displays this form:

Components > Type > Material Properties > New, or
Components > Type > Material Properties > Expand arrow >
Click the **Add New Material** button on the Define Materials form



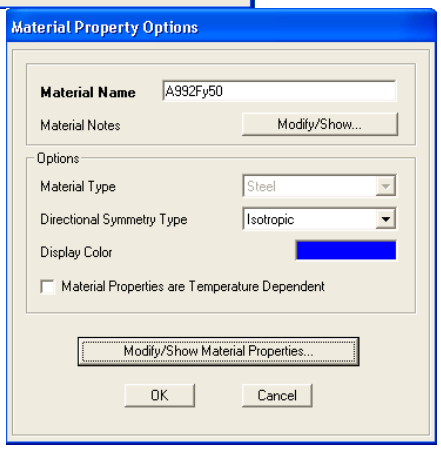
This command (see Table 5-2) displays this form:

Components > Type > Material Properties > Copy or **Modify** command



This command (see Table 5-2) displays this form**:**

Components > Type > Material Properties > Expand arrow > Check the *Show Advanced Properties* check box on the Define Materials form > Click **Add New Materials** button

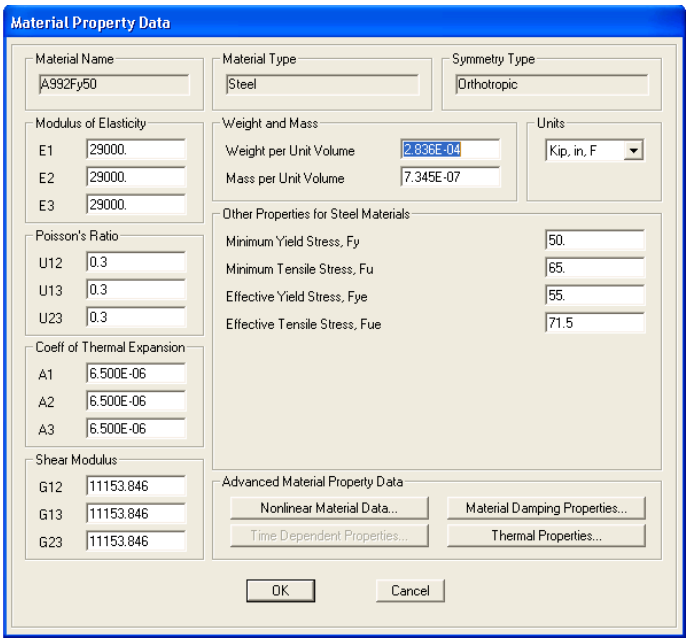


These commands (see Table 5-2) display this form**:**

Components > Type > Material Properties > Expand arrow > Check the *Show Advanced Properties* check box on the Define Materials form > Click **Modify/Show Materials** button; click the **Modify/Show Material Properties** button

OR

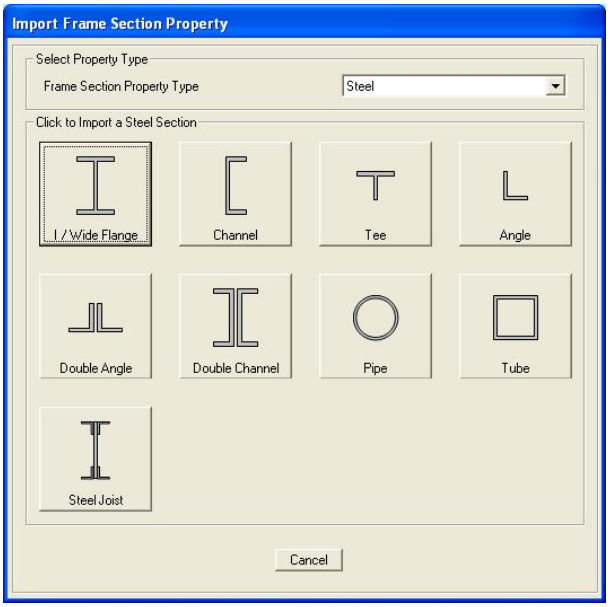
Components > Type > Material Properties > Copy, or **Modify** command > Check the *Switch to Advanced Property Display* check box on the "standard" Material Property Data form; click the **Modify/Show Material Properties** button



5.1.2 Frame Properties Forms – Screen Captures

This command (see Table 5-2) displays this form**:**

Components > Type > Frame Properties > Expand arrow > **Import New Property** button

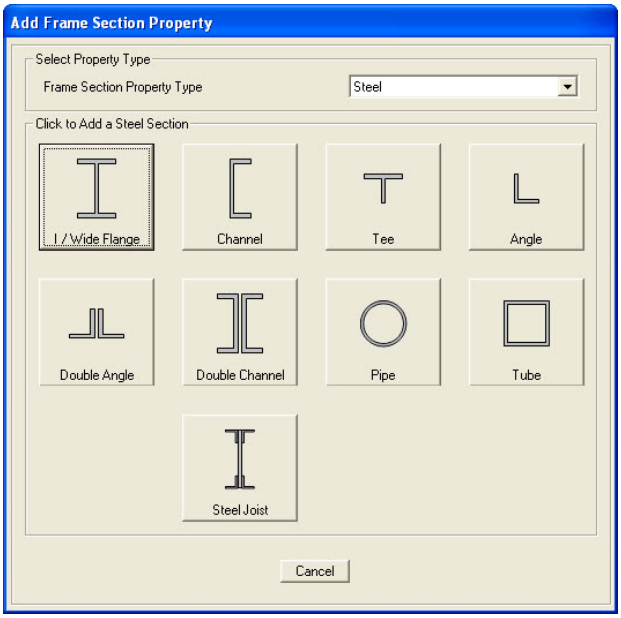


These commands (see Table 5-2) display this form**:**

Components > Type > Frame Properties > Add command

OR

Components > Type > Frame Properties > Expand arrow > **Add New Property** button

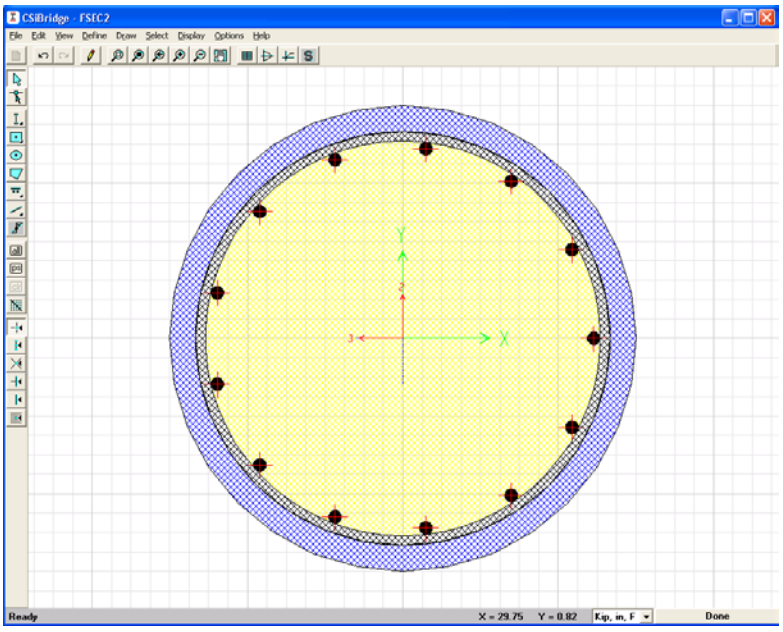


5.1.3 Section Designer – Screen Capture

Section properties may be defined based on their dimensions using the Section Designer utility.

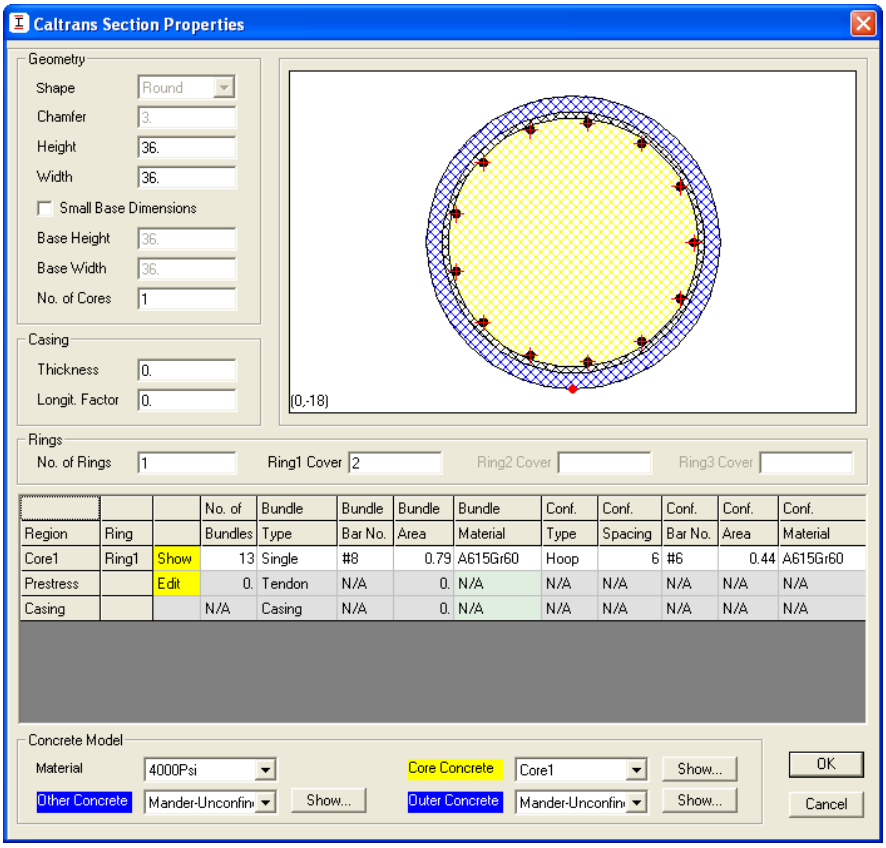
This command (see Table 5-2) displays this form**:**

Components > Type > Frame Properties > Add command > On the Add Frame Section Property form, set the Frame Section Property Type to *Other* > Click the **Section Designer** button.



View of Section Designer Window Displaying an Example of a Concrete Cross-Section Definition

The Section Designer feature provides a powerful means to define complex cross-sectional shapes and details, including rebar layouts for both longitudinal reinforcing and transverse reinforcement.



View of Section Designer Window Displaying the Column Rebar Form; Use to Locate Rebar

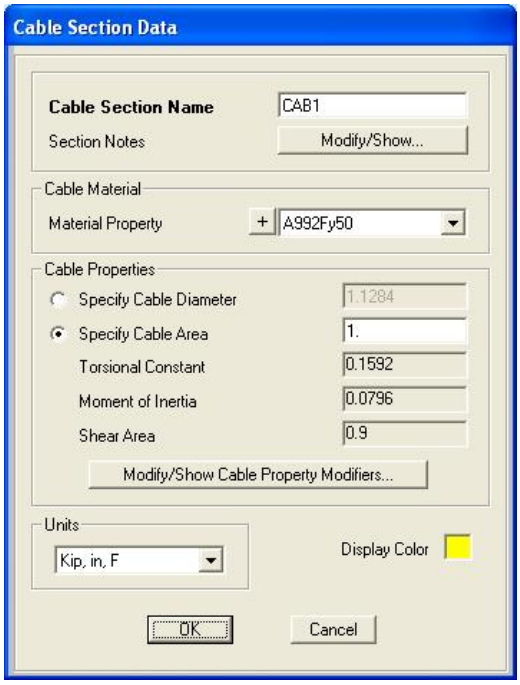
5.1.4 Cable Properties Form – Screen Capture

These commands (see Table 5-2) display this form**:**

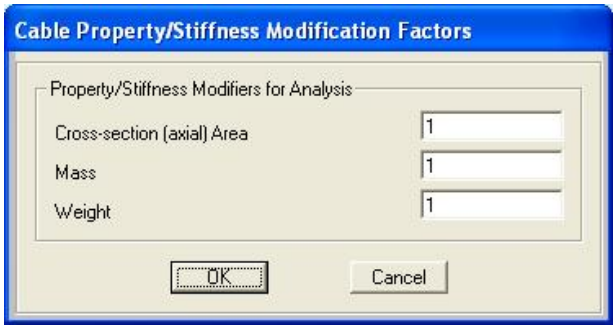
Components > Type > Cable Properties > Add, Copy, Modify command

OR

Components > Type > Cable Properties > Expand arrow > **Add New Section** button



>**Modify/Show Cable Property Modifiers** button:



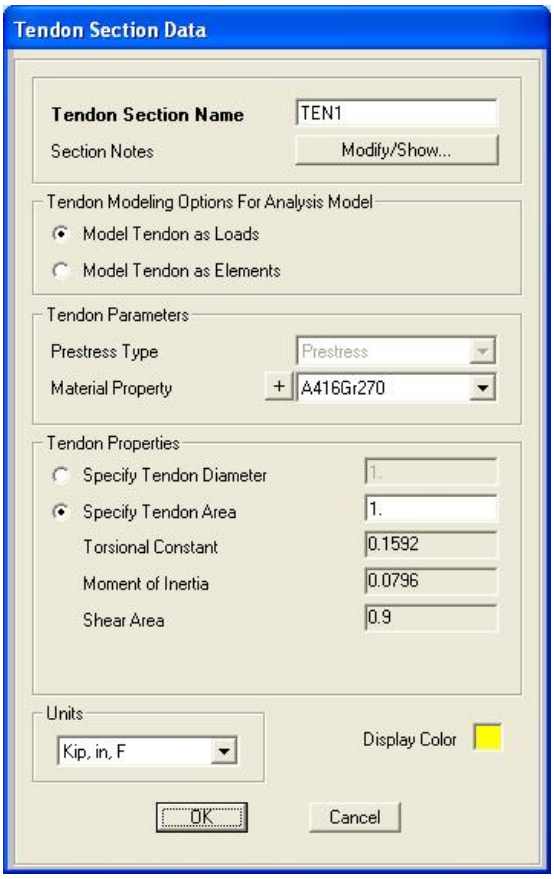
5.1.5 Tendon Properties Form - Screen Capture

These commands (see Table 5-2) display this form**:**

Components > Type > Tendon Properties > Add, Copy, Modify command

OR

Components > Type > Tendon Properties > Expand arrow > **Add New Section** button



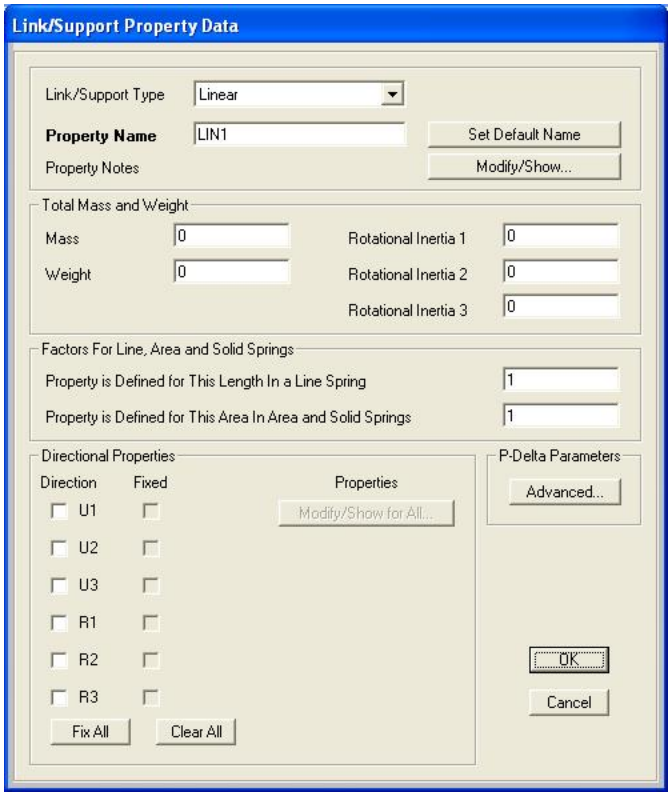
5.1.6 Link/Support Properties Form - Screen Capture

These commands (see Table 5-2) display this form**:**

Components > Type > Link Properties > Add command

OR

Components > Type > Link Properties > Expand arrow > **Add New Property** button

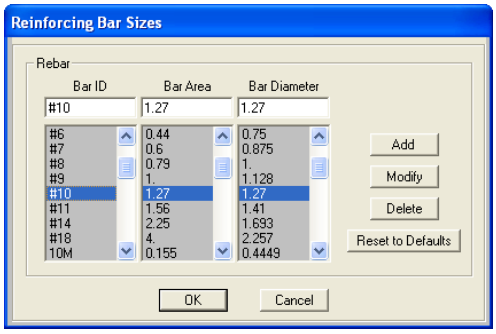


5.1.7 Rebar Properties Form - Screen Capture

Although the program includes several default reinforcement bar definitions, the Reinforcing Bar Sizes form can be used to add, modify, and delete rebar definitions.

Use this command (see Table 5-2) to display that form**:**

Components > Type > Rebar Sizes command



5.2 Components > Superstructure

The superstructure of a CSiBridge model consists of a deck section and diaphragms and is supported by a substructure. The deck section dimensions may vary in accordance with a parametric variation definition. Figure 5-5 shows the *Superstructure* panel of the **Components** tab.

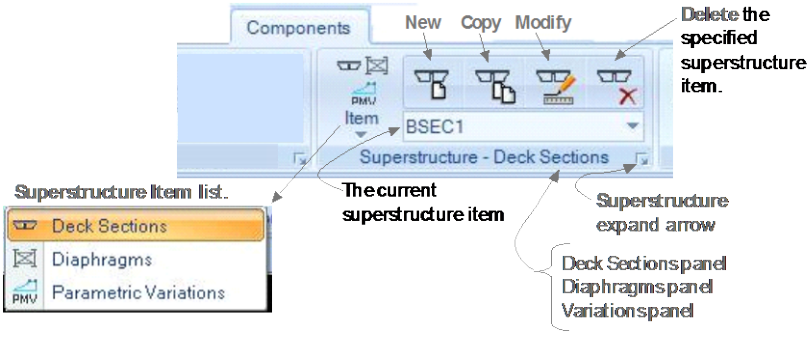


Figure 5-5 Superstructure panel on the Components tab

As suggested in Figure 5-5, clicking the Item command displays a drop-down list of the Superstructure items (i.e., Deck Sections, Diaphragms, and Parametric Variations). The name of the panel changes depending on the superstructure item selected (i.e., panels names are *Superstructure – Deck Sections*; *Superstructure – Diaphragms*; *Superstructure – Variations*). After an item has been selected, clicking the expand arrow displays the forms shown in Figure 5-6.

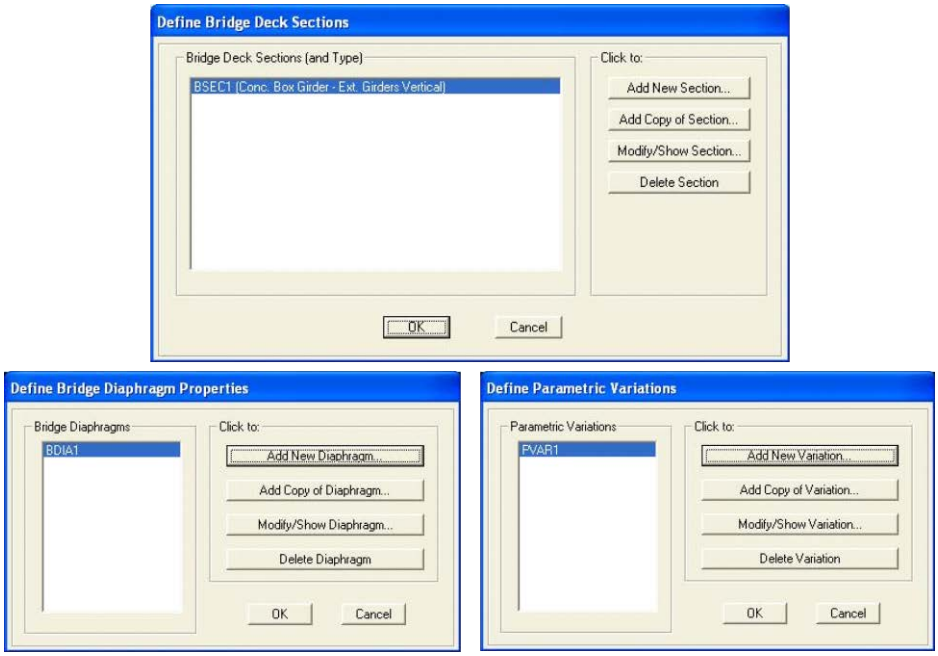


Figure 5-6 Definition forms showing previously defined superstructure items

These forms list all previously defined superstructure item definitions in display areas on the left-hand side of the forms. The buttons in the "Click to" area on the right-hand sides of the forms display the data forms shown in Section 5.2.1, 5.2.2, and 5.2.3.

Referring to Figure 5-5, the **New**, **Copy**, and **Modify** commands on the *Superstructure – {Type}* panel bypass the definition forms shown in Figure 5-6, thereby creating a short cut to the data forms described in Table 5-3.

Table 5-3 Form Data - Component > Superstructure > {Item} > Add, Copy, Modify

Item	Data Form Parameters
Deck Sections	Displays the *Select Bridge Deck Section Type* form, shown in Section 5.2.1. Concrete deck sections include box, multi-cell box, Tee beam, flat slab, precast I-girder, and precast U-girder types. Steel bridge decks include steel I-girder and steel U-girder types. Clicking a deck section type displays a Define Bridge Section Data – {Type} form. As an example, Section 5.2.1 includes the Precast Concrete I Girder form. In this example, customization of the general data for the deck section consists of naming the deck section, and defining the material properties, number of interior girders, width and girder layout. Properties for the slab thickness, haunch thickness, and constant girder sections may be modified. Girder sections available for selection include all I-girders previously defined (see **Components tab > Type > Frame Section > Expand arrow >** Frame Properties form; if no girder properties have been defined, continue to click **Add New Property > Concrete > Precast I Girder** command to define a section). Further parameters in the definition may include the deck dimensions, curb locations, and insertion point. Note that the curb locations are used to determine the extent of vehicle loading across the deck. The curb locations are used to determine the live load distribution factors (LLDF), and the extent of vehicle loading. Users may add curb and rail loads using the **Loads > Loads - Lines** command (see Chapter 6). The insertion point feature may be needed when a deck section is not centered on a layout line.
Diaphragms	Displays the Bridge Diaphragm Property form. The diaphragm types may be solid (concrete bridges only), chord and brace (steel bridges only), single beam (steel bridges only), or steel plate (steel U-girder bridges only). Section 5.2.2 shows the forms for each diaphragm type. ▪ For concrete bridges with a solid diaphragm, the definition is based on the width of the concrete diaphragm. The depth of the concrete is set to match the depth of the concrete deck section. ▪ For steel bridges with a chord and brace diaphragm, the definition includes frame section properties for the top and bottom chords and the brace; specification of the brace type (V Brace, Inverted V Brace, X Brace), and specification of the brace work point location. The brace work point is defined in terms of the change in elevation from the work point of the chord and brace to the top of the adjacent girder and the change in elevation from the bottom work point of the chord and brace to the elevation of the bottom of the adjacent girder. ▪ For steel bridges with a single beam diaphragm, the definition includes beam section property and specification of the change in

Table 5-3 Form Data - Component > Superstructure > {Item} > Add, Copy, Modify

Item	Data Form Parameters
	elevation from the top of the beam to the top of the adjacent girder. ▪ For steel U-girder bridges with a steel plate diaphragm, the definition includes the web and flange dimensions, and the material of the diaphragm.
Parametric Variations	Displays the Variation Definition form shown in Section 5.2.3. Parametric variations can define variations in the deck section along the length of the bridge. Almost all parameters used in the parametric definition of a deck section can be specified to vary. More than one parameter can vary at the same time, if necessary. Each varying parameter can have its own unique variation. The variations may be linear, parabolic, or circular. **Quick Start** button on the Variation Definition form can be used to access the Parametric Variations - Quick Start form shown in Section 5.2.3. Use the form to specify a parametric variation based on template alignments. After a selection has been made and the **OK** button has been clicked on the Parametric Variations - Quick Start form, the selected variation alignment will transfer automatically from the Quick Start form to the Variation Definition form, where the variation can be further refined.

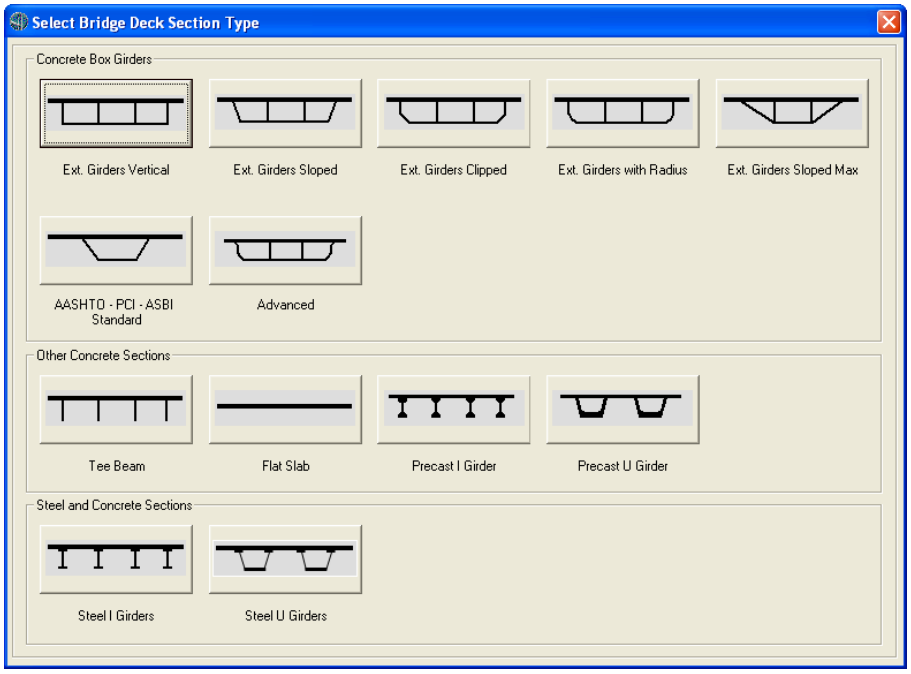
5.2.1 Bridge Deck Section Form - Screen Captures

These commands (see Table 5-3) displays this form:

Components > Superstructure Item > Deck Sections > Add command

OR

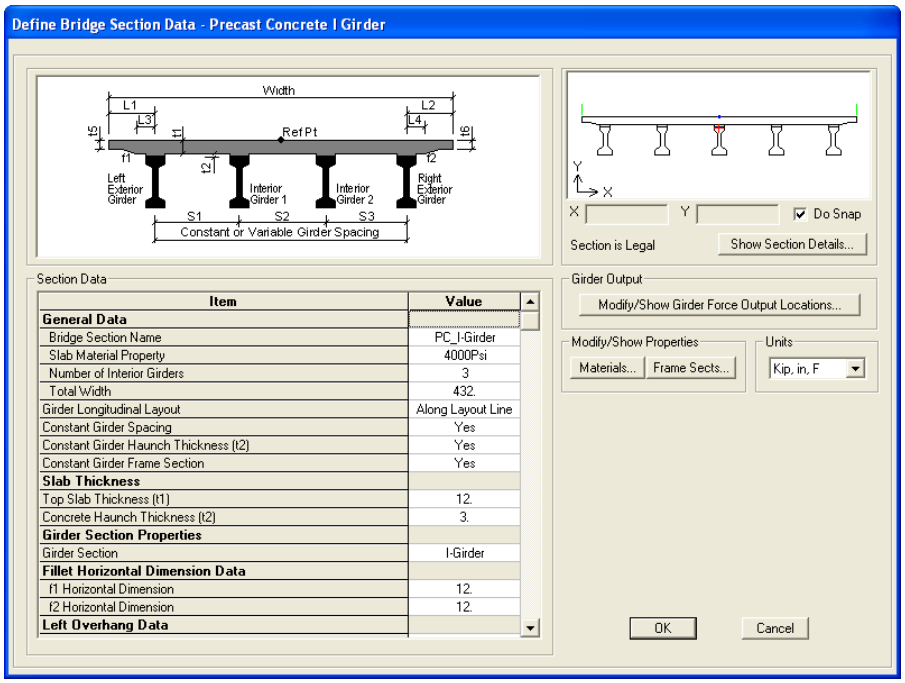
Components > Superstructure Item > Deck Sections Expand arrow > **Add New Section** button



These commands (see Table 5-3) display this form:

Components > Superstructure Item > Deck Sections > Select deck section name from **Current Superstructure Item** drop-down list **> Copy** or **Modify** command

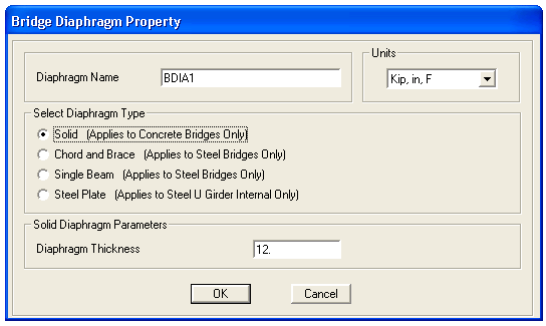
OR **Components > Superstructure Item > Deck Sections > Expand** arrow > **Add New Section** button > {**Deck Section Template**} button

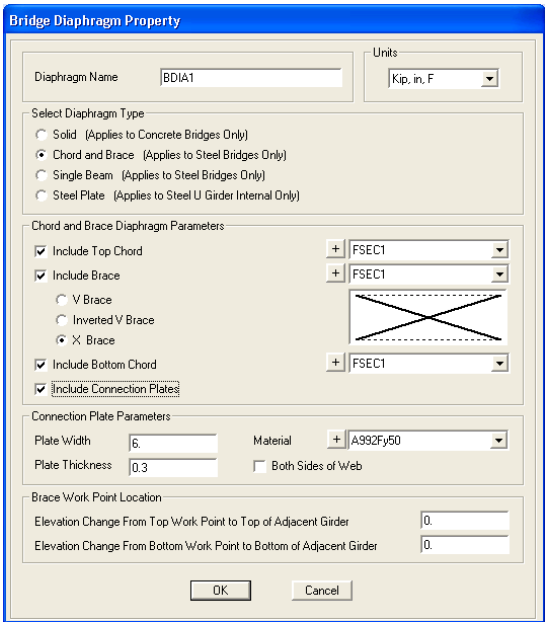


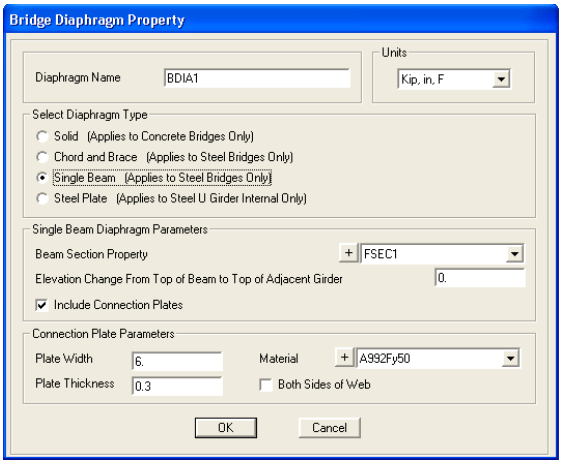
5.2.2 Bridge Diaphragm Form - Screen Captures

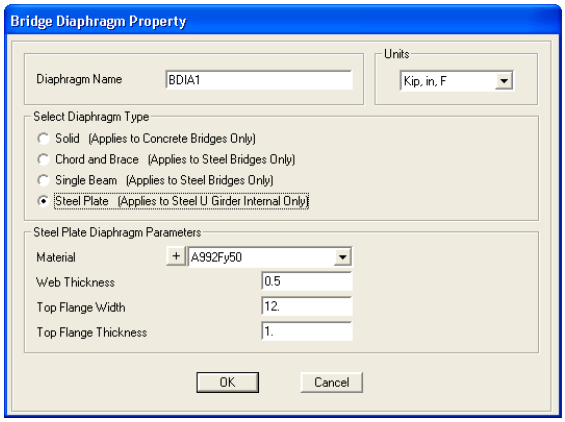
These commands (see Table 5-3) display this form:

Components > {Superstructure} Item > Diaphragm > Add command OR **Components > {Superstructure} Item > Superstructure - Diaphragm Expand** arrow > **Add New Diaphragm** button





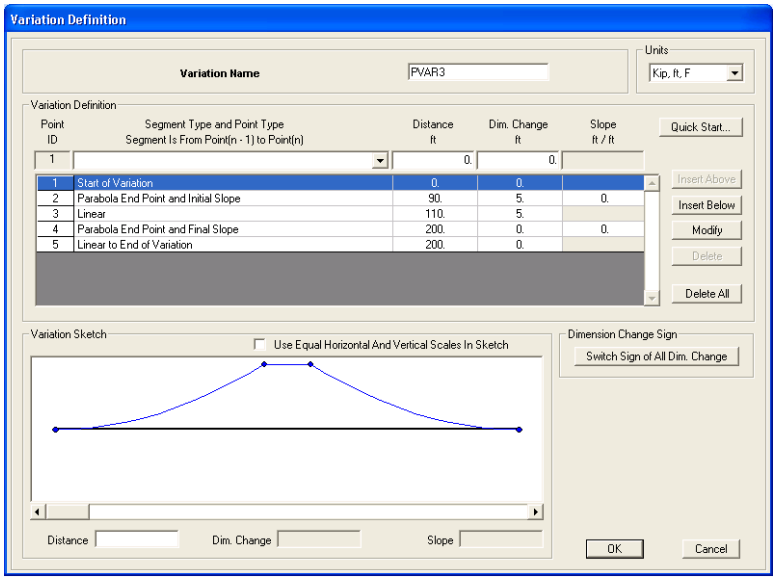

Bridge Diaphragm Property
Diaphragm Name
BDIA1
Units
Kip, in, F
Select Diaphragm Type
Solid (Applies to Concrete Bridges Only)
Chord and Brace (Applies to Steel Bridges Only)
Single Beam (Applies to Steel Bridges Only)
Steel Plate (Applies to Steel U Girder Internal Only)
Single Beam Diaphragm Parameters
Beam Section Property
FSEC1
Elevation Change From Top of Beam to Top of Adjacent Girder
0.
Include Connection Plates
Connection Plate Parameters
Plate Width
6.
Material
A992Fy50
Plate Thickness
0.3
Both Sides of Web
OK
Cancel


Bridge Diaphragm Property
Diaphragm Name
BDIA1
Units
Kip, in, F
Select Diaphragm Type
Solid (Applies to Concrete Bridges Only)
Chord and Brace (Applies to Steel Bridges Only)
Single Beam (Applies to Steel Bridges Only)
Steel Plate (Applies to Steel U Girder Internal Only)
Steel Plate Diaphragm Parameters
Material
A992Fy50
Web Thickness
0.5
Top Flange Width
12.
Top Flange Thickness
1.
OK
Cancel

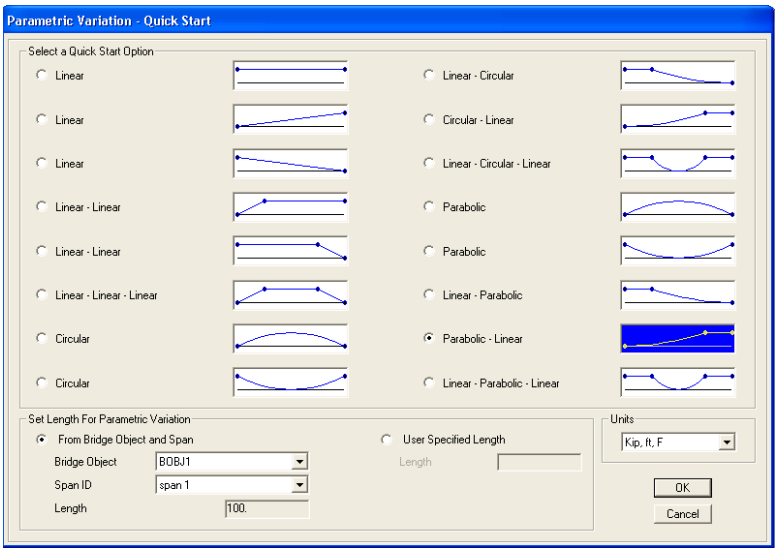
5.2.3 Parametric Variation Forms - Screen Captures

These commands (see Table 5-3) display these forms:

Components > {Superstructure} Item > Parametric Variations > Add command OR **Components > {Superstructure} Item > Superstructure - Variations Expand** arrow > **Add New Variation** button



> Click **Quick Start** button



5.3 Component > Substructure

The substructure definitions for a CSiBridge model may include bearings, restrainers, foundation springs, abutments, and bents. Each of these components, if used in a bridge model, will become part of the total bridge assembly based on their locations and definitions in the Bridge Object (see Chapter 7).

Figure 5-12 shows the *Substructure* panel of the **Components** tab.

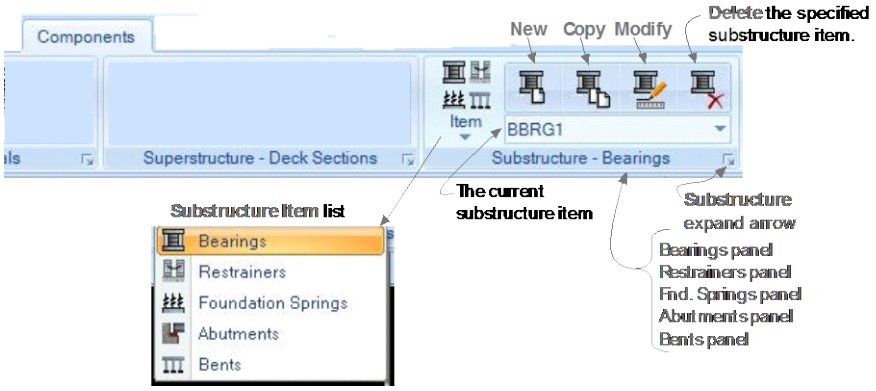


Figure 5-12 Substructure panel on the Components tab

As suggested in Figure 5-12, clicking the **Item** command displays a drop-down list of the Substructure items (i.e., Bearings, Restrainers, Foundation Springs, Abutments, and Bents). The name of the panel changes depending on the substructure item selected (i.e., panels names are *Substructure – Bearings*; *Substructure – Restrainers*; *Superstructure – Fnd. Springs*; *Substructure – Abutments*; *Substructure -- Bents*). After an item has been selected, clicking the expand arrow displays the definition forms shown in Figure 5-13.

Those forms list all previously defined substructure item definitions in display areas on the left-hand side of the forms. The buttons in the "Click to" area on the right-hand sides of the forms display the data forms shown in Section 5.3.1 through 5.3.5. Referring to Figure 5-12, the **New**, **Copy**, and **Modify** commands on the *Superstructure – {Type}* panel bypass the definition forms shown in Figure 5-13, thereby creating a short cut to the data forms, which are identified in Table 5-4.

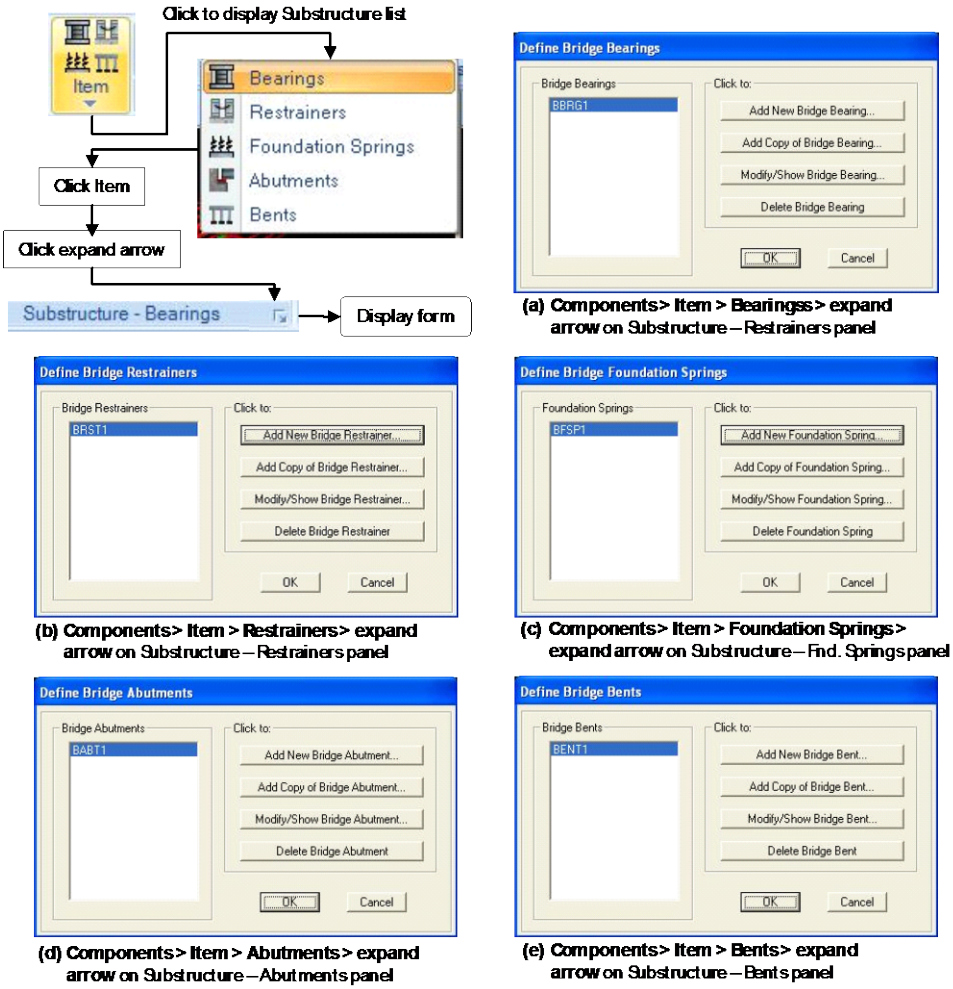


Figure 5-13 Definition forms showing previously defined substructure items

Table 5-4 Form Data - Component > Substructure > {Item} > Add, Copy, Modify

Item	Data Form Parameters
Bearings	Displays the Bearing Data form shown in Section 5.3.1. Bearing properties are used in abutment, bent and hinge assignments to the bridge object. At abutments, bearings are used in the connection between the girders and the substructure. At bents, bearings are used in the connection between the girders and the bent cap beam. At hinges, bearings are used in the connection between the girders on the two sides of the hinge. A bearing property can be specified as a Link/Support property or it can be user defined. The user defined bearing is recommended. The user defined bearing allows each of the six degrees of freedom to be specified as fixed, free or partially restrained with a specified user defined spring constant. The user may select the release type as Free, Fixed or Partial Fixity for each of the six degrees of freedom.
Restrainers	Displays the Bridge Restrainer data form shown in Section 5.3.2. Restrainer cables are used as tension ties across superstructure discontinuities. Restrainers may be assigned at abutments, hinges, and at bents where the superstructure is discontinuous over the abutment or bent. When specified, the program assumes that a restrainer cable exists at each girder location. A restrainer property can be specified as a Link/Support property or it can be user defined. The user defined restrainer is recommended. The user defined restrainer is specified by a length, area, and modulus of elasticity.
Foundation Springs	Displays the Foundation Spring data form shown in Section 5.3.3. Foundation spring properties specify data for the connection of the substructure to the ground. Foundation spring properties are used in abutment and bent property definitions. At bents, foundation springs may be used at the base of each column. In that case, the foundation springs are used as point springs. At abutments, foundation springs are used as point springs for a foundation spring-type substructure, and they are used as spring properties per unit length for a continuous beam-type substructure. A foundation spring property can be specified as a Link/Support property or it can be user defined. The user defined spring is recommended. The user defined foundation spring allows each of the six degrees of freedom to be specified as fixed, free or partially restrained with a specified spring constant. For cases where the spring property is used for a continuous beam support, a factor is specified indicating the length over which the specified properties apply.

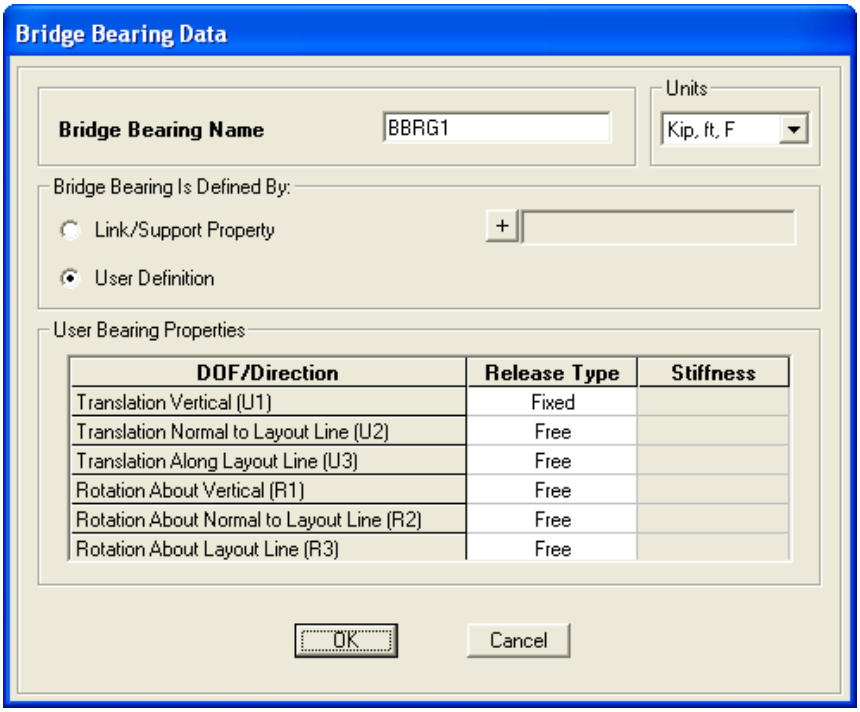
Table 5-4 Form Data - Component > Substructure > {Item} > Add, Copy, Modify

Item	Data Form Parameters
Abutments	Displays the Bridge Abutment data form shown in Section 5.3.4. Abutment (end bent) properties specify the support conditions at the ends of the bridge. Abutment properties are used in abutment assignments to the bridge object. The abutment property allows specification of the connection between the abutment and the girders as either integral or connected to the bottom of the girders only. The abutment property also allows the abutment substructure to be specified as a series of point springs (one for each girder) or a continuously supported beam.
Bents	Displays the Bridge Bent Data form shown in Section 5.3.5. Bent properties specify the geometry and section properties of the bent cap and the bent columns. They also specify the base support condition of the bent columns. Bent properties may be used in abutment assignments to the bridge object. The bent property allows specification of the connection between the abutment and the girders as either integral or connected to the bottom of the girders only. The bent property also allows specification of a single bearing line (continuous superstructure) or a double bearing line (discontinuous superstructure). The bridge superstructure will be updated as continuous or discontinuous based on the Bent Type option. The **Modify/Show Column Data** button opens the Bridge Bent Column data form shown in Figure 5-19 that is used to define the column locations, heights, seismic hinge data, and base support condition.

5.3.1 Bridge Bearing Data Form – Screen Capture

These commands (see Table 5-4) display this form:

Components > {Substructure} Item > Bearings > Add command OR **Components > {Substructure} Item > Substructure - Bearings Expand** arrow > **Add New Bridge Bearing** button



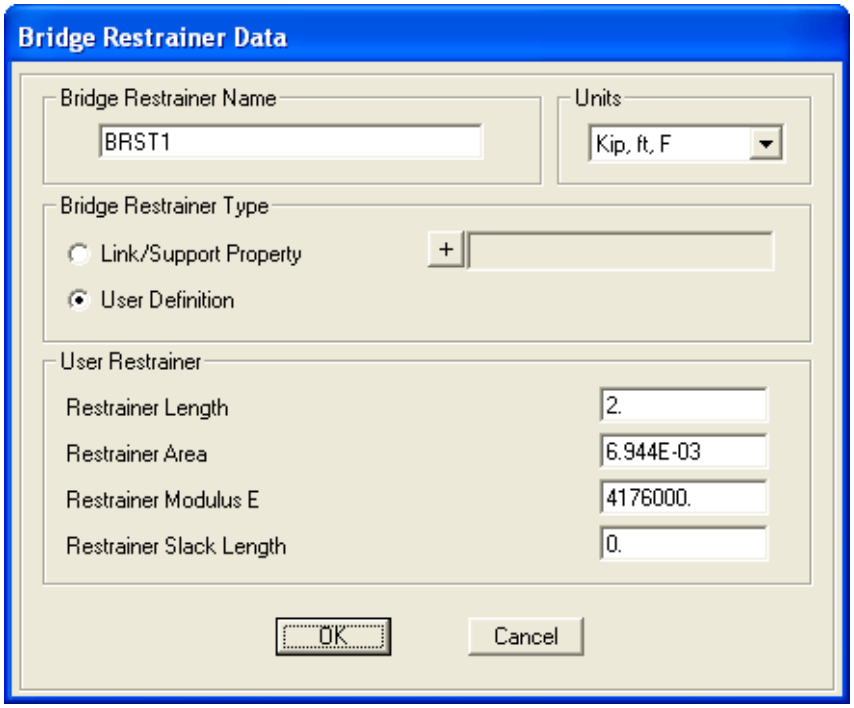
5.3.2 Bridge Restrainer Data Form – Screen Capture

These commands (see Table 5-4) display this form:

Components > {Substructure} Item > Restrainers > Add command

OR

Components > {Substructure} Item > Substructure - Restrainers Expand arrow > **Add New Bridge Restrainer** button



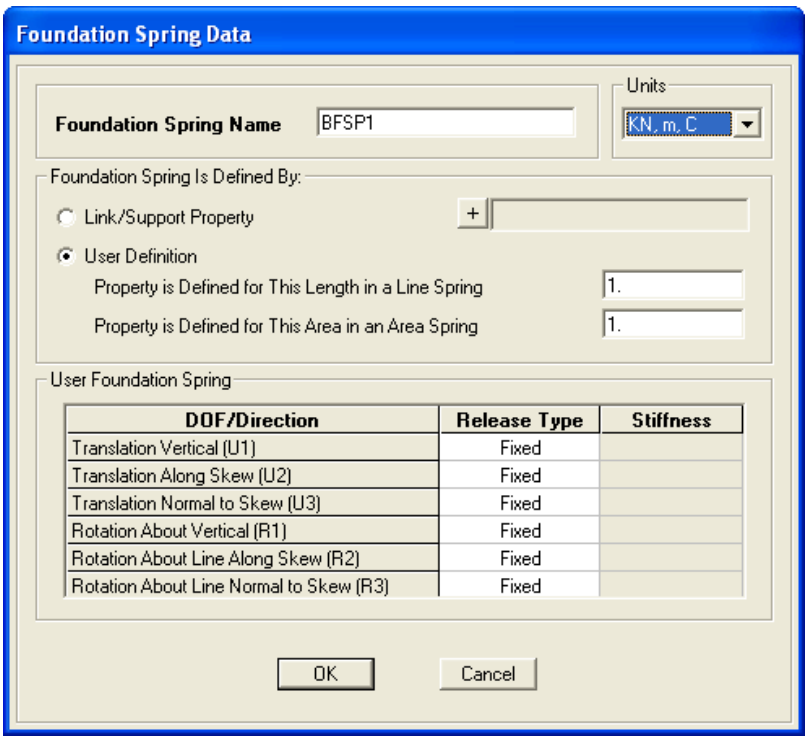
5.3.3 Foundation Spring Data Form – Screen Capture

These commands (see Table 5-4) display this form:

Components > {Substructure} Item > Foundation Springs > Add command

OR

Components > {Substructure} Item > Substructure – Fnd. Springs Expand arrow > **Add New Foundation Spring** button



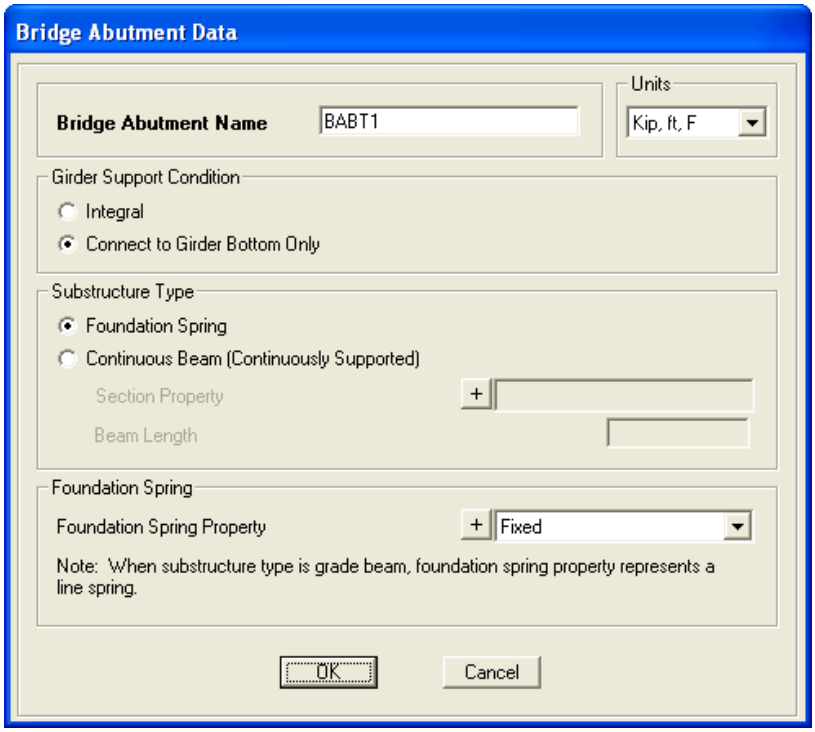
5.3.4 Bridge Abutment Data Form – Screen Capture

These commands (see Table 5-4) display this form:

Components > {Substructure} Item > Abutments > Add command

OR

Components > {Substructure} Item > Substructure – Abutments Expand arrow > **Add New Bridge Abutment** button



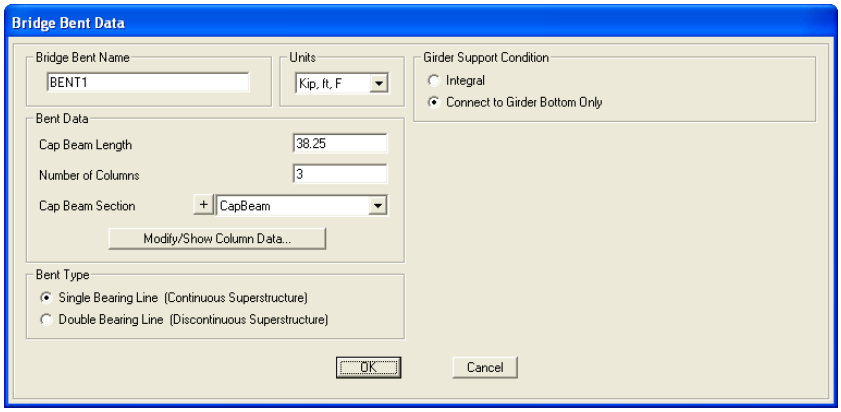
5.3.5 Bridge Bent Forms – Screen Captures

These commands (see Table 5-4) display this form:

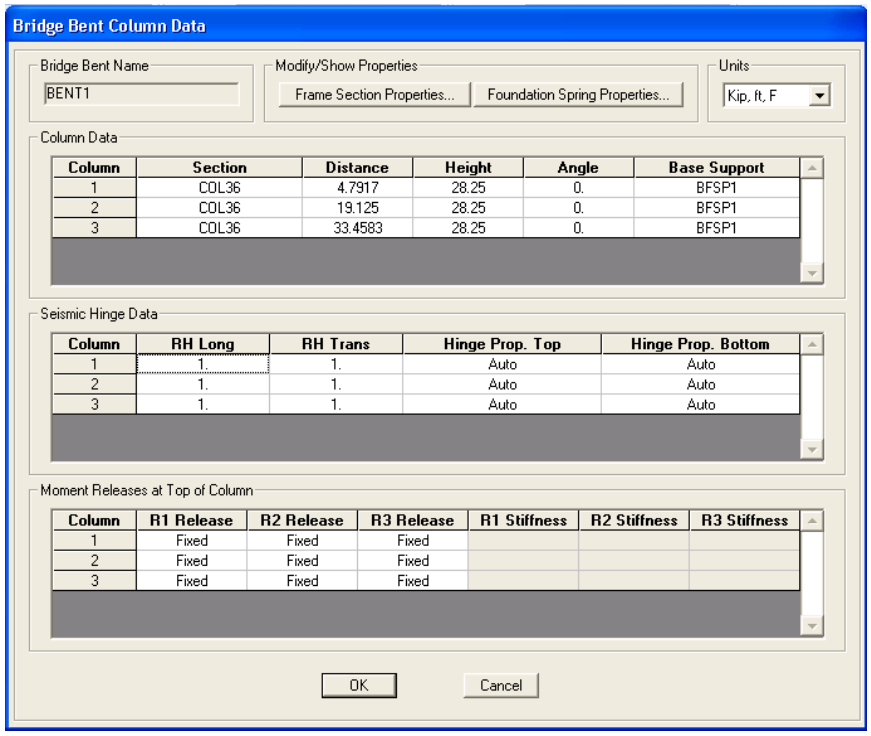
Components > {Substructure} Item > Bents > Add command

OR

Components > {Substructure} Item > Substructure – Bents Expand arrow > **Add New Bridge Bent** button



> Click **Modify/Show Column Data** button


Bridge Bent Column Data
Bridge Bent Name
BENT1
Modify/Show Properties
Frame Section Properties...
Foundation Spring Properties...
Units
Kip, ft, F
Column Data
Column | Section | Distance | Height | Angle | Base Support
1 | COL36 | 4.7917 | 28.25 | 0. | BFSP1
2 | COL36 | 19.125 | 28.25 | 0. | BFSP1
3 | COL36 | 33.4583 | 28.25 | 0. | BFSP1
Seismic Hinge Data
Column | RH Long | RH Trans | Hinge Prop. Top | Hinge Prop. Bottom
1 | 1. | 1. | Auto | Auto
2 | 1. | 1. | Auto | Auto
3 | 1. | 1. | Auto | Auto
Moment Releases at Top of Column
Column | R1 Release | R2 Release | R3 Release | R1 Stiffness | R2 Stiffness | R3 Stiffness
1 | Fixed | Fixed | Fixed
2 | Fixed | Fixed | Fixed
3 | Fixed | Fixed | Fixed
OK
Cancel

CHAPTER 6 Loads

The **Loads** tab consists of the commands that allow efficient access to the *data* forms needed to add, copy, or modify definitions for vehicles and vehicle classes; load patterns; and response spectrum or time history functions; and point, line or area loads. A delete command is available for deleting a selected definition. A method for displaying the *definition* form that lists all definitions and that has buttons that perform the same functions as the commands on the **Loads** tab also is available. This chapter identifies those data and definition forms.

If the Quick Bridge template was used to start the bridge model, the program will have created default definitions for vehicles, vehicle classes, load patterns and response spectrum and time history functions. These definitions can be viewed using the commands on the **Loads** tab.

If the ***Bridge Wizard*** is being used, highlighting the Vehicles, Vehicle Classes, Response Spectrum Functions, Time History Functions, and Load Pattern Definitions items in the Summary Table and clicking the **Define/Show Vehicles**, **Define/Show Vehicle Classes**, **Define/Show Resp. Spec. Funcs.**, **Define/Show Time History Funcs.**, and **Define/Show Load Patterns** buttons will display the *definition* forms that can then be used to access the same *data* forms as those that can be accessed directly using the **Loads** tab commands**.**

The commands on the **Loads** tab also can be used if the Blank option was used to start the model and the ***Bridge Wizard*** is not being used (i.e., the model is being built from scratch or by importing model data).

6.1 Loads > Vehicles

Vehicles must be defined to analyze a bridge model for vehicle live loads. In CSiBridge, vehicle loads are applied to the structure through lanes (for lane definitions, see Chapter 3). Each vehicle definition consists of one or more concentrated or uniform loads.

Figure 6-1 shows the commands on the *Vehicles* panel of the **Loads** tab.

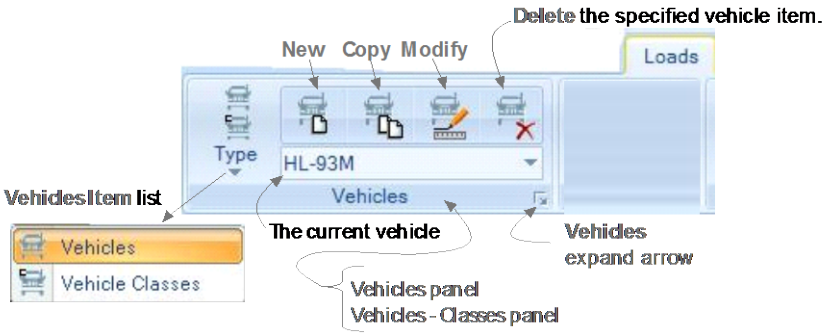
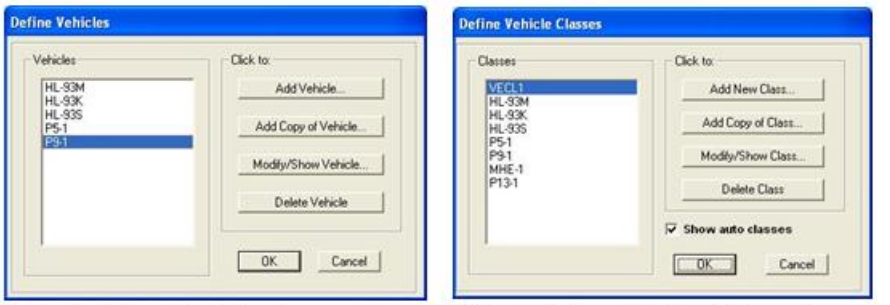


Figure 6-1 Loads tab with annotated Vehicle panel

As suggested in Figure 6-1, clicking the **Vehicles Type** command displays a drop-down list that can be used to select Vehicles or Vehicles Classes. The name of the panel changes from Vehicles to Vehicles – Class depending on the selection made. After Vehicle or Vehicle Classes has been selected, clicking the expand arrow displays one of the forms shown in Figure 6-2.



(a) Loads > Item > Vehicles > expand arrow on Vehicles panel

(b) Loads > Item > Vehicle Classes > expand arrow on Vehicle – Classes panel

Figure 6-2 Definition forms showing previously defined vehicles and vehicle classes

These forms list all previously defined vehicles and vehicles classes in display areas on the left-hand side of the forms. Generally, clicking the buttons in the "Click to" area on the right-hand sides of the forms display the data forms described in Table 6-1 and shown in Sections 6.1.2 and 6.1.3. Referring to Figure 6-1, the **New**, **Copy**, and **Modify** commands on the *Vehicles* panel or the *Vehicles Classes* panel bypass the definition forms shown in Figure 6-2, and display the forms shown in Sections 6.1.2 and 6.1.3, thereby creating a shortcut to the data forms.

Table 6-1 Form Data - Loads > Vehicles > {Type} > Add, Copy, Modify

Type	Data Form Parameters
Vehicles	Displays the Vehicle Data form or the General Vehicle Data form (see Section 6.1.1). Numerous standard vehicle definitions are built into the program. The general vehicle form can be used to create customized vehicle definitions. ▪**Standard vehicle data** – Several vehicles types to represent vehicle live loads specified in various design codes are available for selection using the drop-down lists on the form. The variables available within the various drop-down lists are interdependent. For example, when the Region is set to Europe, the Standard is automatically set to a European code and the selection of available Vehicle Types reflects that code. The associated Scale Factor, Dynamic Allowance, and Class edit boxes become enabled when an applicable Vehicle Type has been selected. Whenever a Vehicle definition is created, CSiBridge automatically creates a Vehicle Class of the same name, containing only that single vehicle definition with a scale factor of 1.0.

Table 6-1 Form Data - Loads > Vehicles > {Type} > Add, Copy, Modify

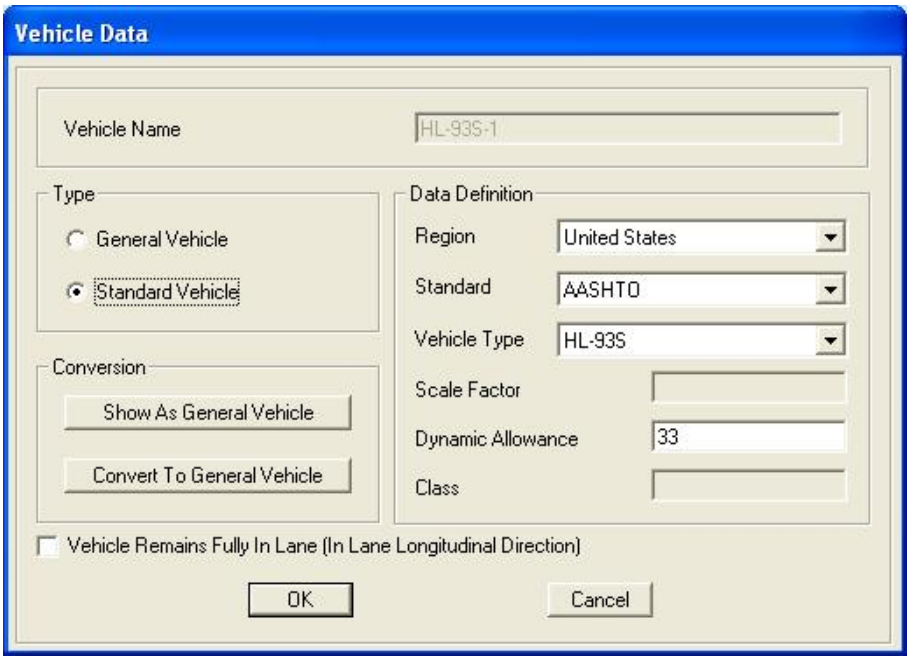
Type	Data Form Parameters
	o The integer scale factor specifies the nominal weight of the vehicle in a specific set of units. For example, for H & HS vehicles the units are tons; for UIC vehicles the units are kN/m; and so on. o A dynamic load allowance is the additive percentage by which the concentrated truck or tandem axle loads will be increased. The uniform lane load is not affected. Thus, if the dynamic allowance equals 33, all concentrated axle loads for the vehicle will be multiplied by the factor 1.33. ▪**General vehicle data** – The general vehicle may represent an actual vehicle or a notional vehicle used by a design code. The general vehicle consists of *n* axles with specified distances between them. Concentrated loads may exist at the axles. Uniform loads may exist between pairs of axles, in front of the first axle, and behind the last axle. The distance between any one pair of axles may vary over a specified range; the other distances are fixed. The leading and trailing uniform loads are of infinite extent. Additional "floating" concentrated loads may be specified that are independent of the position of the axles.
Vehicles Classes	Displays the Vehicle Class Data form (see Section 6.1.2). A vehicle class is simply a group of one or more vehicles that is used in a moving load analysis (one vehicle at a time). Vehicle classes may be defined to include any number of individual vehicles to allow consideration of the maximum and minimum response of the bridge to the most extreme of several type of vehicles rather than the effect of the individual vehicles. However, whenever a Vehicle definition is created, CSiBridge automatically creates a Vehicle Class of the same name, containing only that single vehicle definition with a scale factor of 1.0. The automatically created Vehicle Class definition cannot be modified or deleted except by modifying or deleting the vehicle. The maximum and minimum force and displacement response quantities for a vehicle class will be the maximum and minimum values obtained for any individual vehicle in that class. For influence based analysis, all vehicle loads are applied to the traffic lanes using vehicle classes. To apply an individual vehicle load, define a vehicle class that contains only a single vehicle. For step-by-step analysis, vehicle loads are applied directly without the use of classes since no enveloping is performed.

6.1.1 Vehicle Data Forms – Screen Captures

Use these commands (see Table 6-1) to display the following form and define a standard vehicle:

Loads > Type > Vehicles > Expand arrow > Add Vehicle command or **Loads > Type > Vehicles > Add** command.

Alternatively, click **Loads > Type > Vehicles >** select a **Standard Vehicle** definition from the Current Vehicle drop-down list **> Copy, Modify**



Use these commands (see Table 6-1) to display this form and define a general vehicle.

Loads > Type > Vehicles > Expand arrow > click the **Add Vehicle** button > select the General Vehicle option

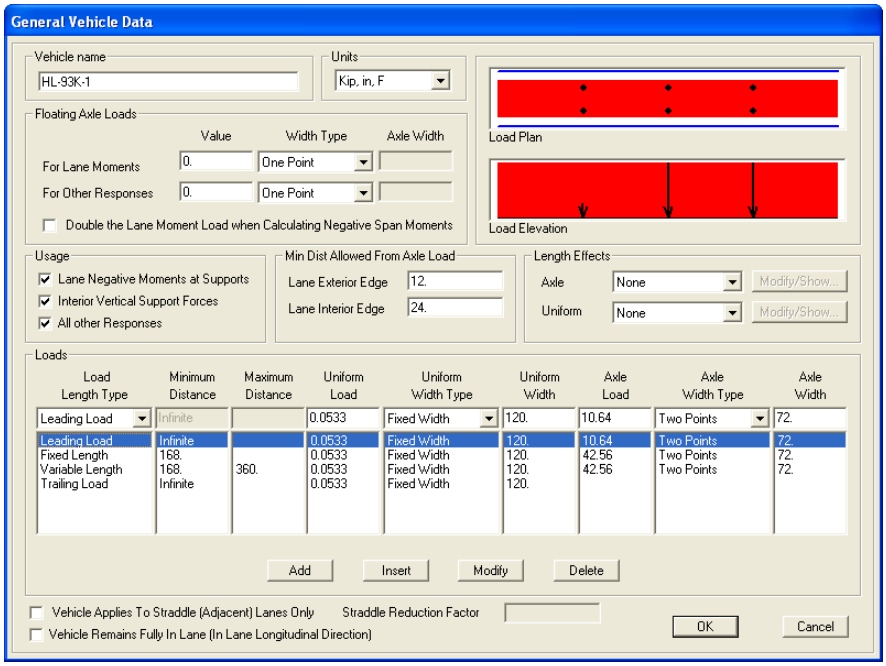
Or

Loads > Type > Vehicles > Expand arrow > click the **Add Vehicle** button > click the **Convert to General Vehicle** button > choose **Yes***

Or

Loads > Type > Vehicles > select a **General Vehicle** definition from the *Current Vehicle* drop-down list **> Copy, Modify**

*After a Standard Vehicle definition has been converted to a General Vehicle definition, the conversion cannot be reversed.



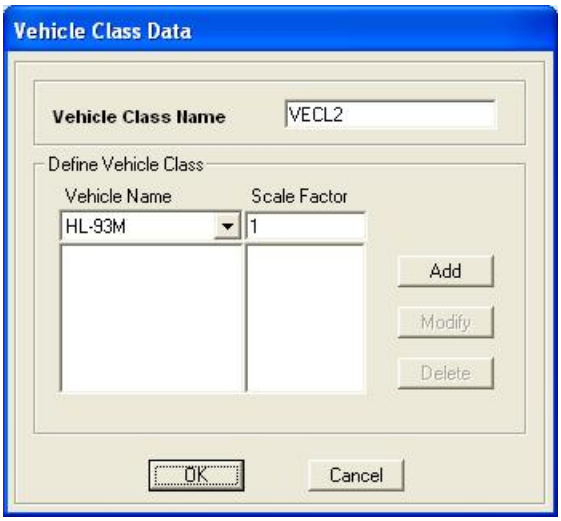
6.1.2 Vehicle Classes Data Forms – Screen Capture

These commands (see Table 6-1) display this form.

Loads > Type > Vehicle Classes > Expand arrow > Add New Class button

Or

Loads > Type > Vehicle Classes > Add, Copy, Modify



6.2 Loads > Load Patterns

Clicking the **Loads** > **Load Patterns** command immediately displays the form for defining a load pattern. A load pattern has a name, load type and a specified spatial distribution of forces, displacements, temperatures, and other effects that act upon a structure. A load pattern by itself does not cause any response in the structure. Load patterns must be applied in load cases to produce results (see Chapter 9).

Figure 6-3 shows the **Loads** > **Load Patterns** command and the resulting forms that are used to define the load pattern as well as generate a multi step bridge live load pattern. In turn, the multi step bridge live load pattern is used in a multi-step static or multi-step dynamic (direct integration time history) load case to evaluate special vehicle load responses (see *Chapter 8 Analysis*). Table 6-2 briefly describes the data used to define a load pattern in CSiBridge.

Table 6-2 Form Data - Loads > Load Patterns

Command	Data Form Parameters
Load Patterns	Displays the Define Load Patterns form. Each load pattern must have a unique name. The Type drop-down list provides access to the various load types defined in the AASHTO LRFD code for use as part of a load pattern definition. The load type is used to determine the Auto Load Combinations (see Chapter 9). The Vehicle Live load pattern is among the more significant load types for a bridge model in CSiBridge. In a vehicle Live load pattern, one or more vehicles is selected and assigned to a specific lane along with a starting time, direction, and speed. When used in a multi-step static or multi-step dynamic (direct integration time history) load case, this type of load pattern is useful in evaluating special vehicle load responses. The direct integration time history analysis will produce response spectrum output data for the user specified imposed vehicle live loads. The Self Weight Multiplier is a scale factor that multiplies the weight of the structure and applies it as a force in the gravity direction (negative global Z direction). The self-weight of the structure is determined by multiplying the weight per unit volume of each object that has structural properties times the volume of the object. The weight per unit volume is specified in the material properties (see Chapter 5)

Table 6-2 Form Data - Loads > Load Patterns

Command	Data Form Parameters
	The Auto Lateral Load Pattern is applicable to Quake, Wave, and Wind load types. With this option, the user can specify that code-compliant loads be created automatically for the load pattern. Alternatively, when a quake, wave, or wind load is being specified, the user can select "None" here and then apply loads manually by assigning them using applicable **Advanced > Assign Loads** commands (see Chapter 10).

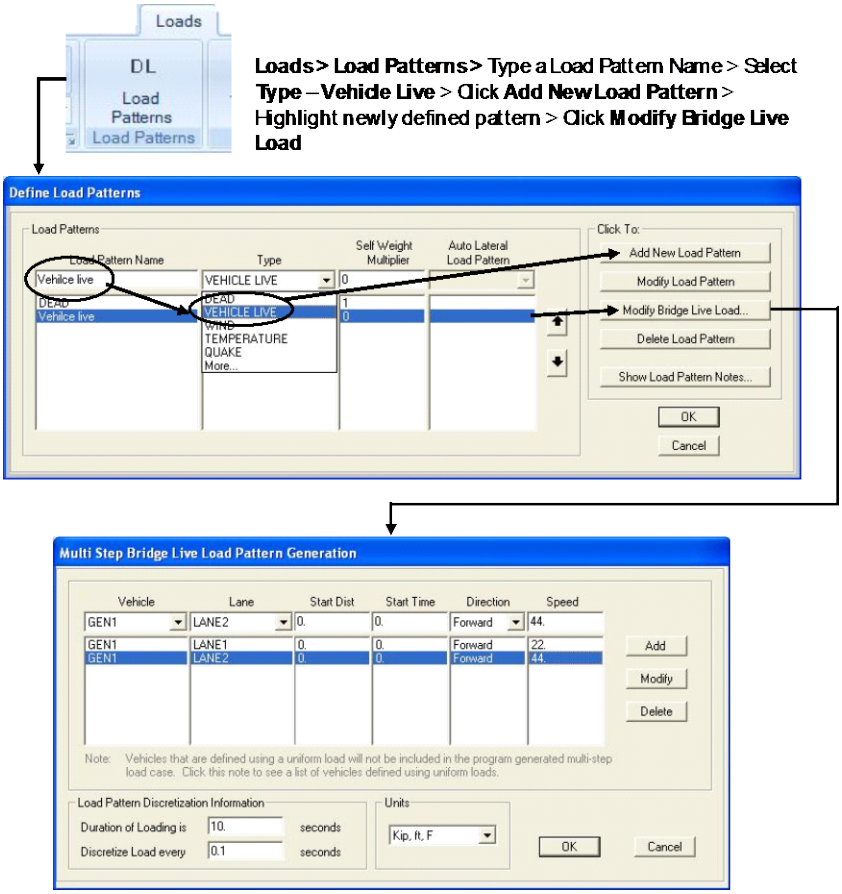


Figure 6-3 Load Patterns panel on the Loads tab, and screen captures of the forms used to define a load pattern that includes a vehicle live load

With the forms displayed, depress the F1 key for context sensitive Help.

6.3 Loads > Functions

In addition to the dead, live, and moving load cases (see previous section), CSiBridge provides for bridge structures to be analyzed using response spectrum and time history load cases (see Chapter 8). To define a response spectrum or time history load case, the user must first define a response spectrum or time history function that will be used as part of a load case definition.

Figure 6-4 shows the commands on the *Functions* panel of the **Loads** tab. As suggested in the figure, clicking the Functions Type command displays a drop-down list that can be used to select a Response Spectrum or Time History function. The name of the panel changes from Function - Response Spectrum to Function - Time History depending on the selection made. After Response Spectrum or Time History has been selected, clicking the expand arrow displays the forms shown in Figure 6-5.

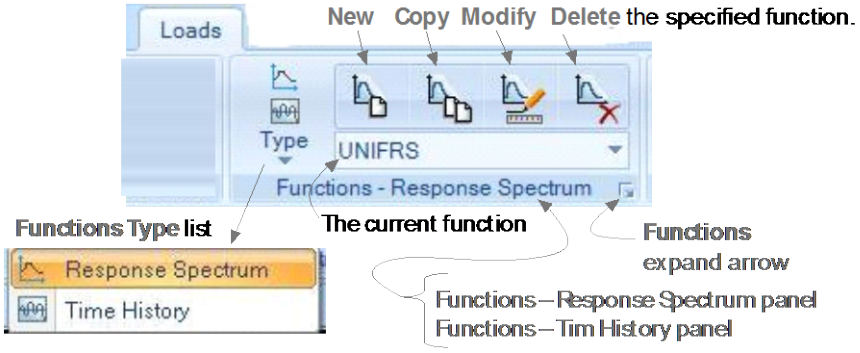


Figure 6-4 Loads tab with annotated Functions panel

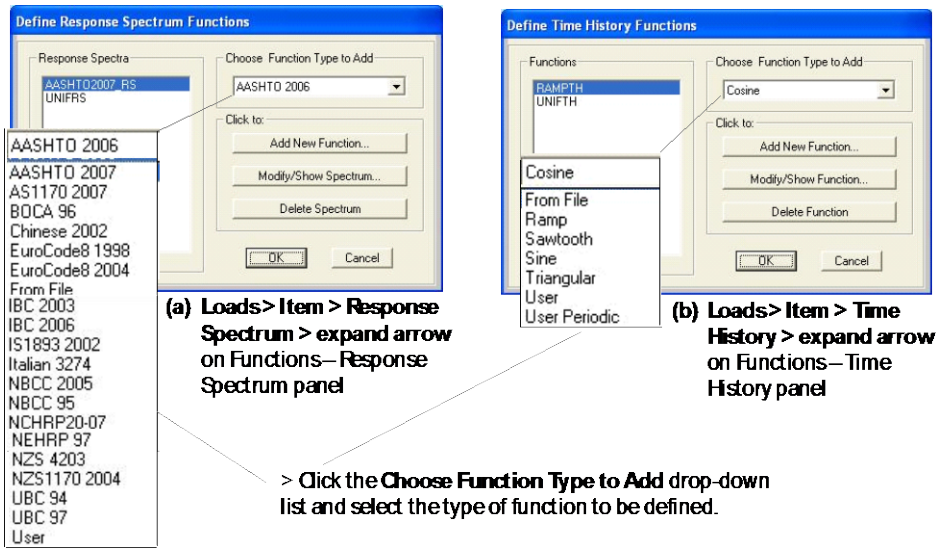


Figure 6-5 Definition forms that show all previously defined functions and that have drop-down lists for choosing the type of response spectrum or time history function to be defined

The forms in Figure 6-5a and 6-5b list all previously defined functions in display areas on the left-hand side of the forms.

Note: The type of function selected from the Choose Function Type to Add drop-down list on the Define Response Spectrum Functions form or the Define Time History Functions form determines which of the many data/definition forms will be displayed when the **Add New Function** or **Modify/Show Spectrum** button is clicked. Similarly, the Function selected in the Current Function drop-down list on the *Functions* – {*Type*} panel (see Figure 6-4) determines which form will be displayed when the **Copy** or **Modify** commands on that panel is clicked. When the **Add** command on the panel is selected, a Response Spectrum or Time History form displays with a single drop-down list for selecting the type of response spectrum function/time history function to be defined. The data form then displays, with parameters that apply to the type of function being defined.

Table 6-3 briefly describes the various types of functions. Sections 6.3.1 and 6.3.2 provide screen captures of an AASHTO 2007 code-compliant response spectrum function definition and a "Function from File" time history function definition as examples of the data forms used.

After a response spectrum or time history function has been defined, it may be used in a load case definition, which is created using the **Analysis > Load Cases > Type-Response Spectrum** or **Type-Time History** command (See Chapter 8).

Table 6-3 Form Data - Loads > Functions > {Type} > Copy, Modify

Type	Data Form Parameters
Response Spectrum	Displays a Response Spectrum {Type} Function Definition form. A response-spectrum function is a list of period versus spectral-acceleration values. In CSiBridge, the acceleration values in the function are assumed to be normalized; that is, the functions themselves are not assumed to have units. Instead, the units are associated with a scale factor that multiplies the function and that is specified when the response-spectrum load case is defined. By default, the program pre-defines a unit constant response-spectrum function. This function may be modified or deleted. A response-spectrum function may be defined for any period range, *T0* to *Tn*. For periods less than the first given period, *T0*, the function value is assumed to be constant at the function value specified for *T0*. For all periods greater than the last given period, *Tn*, the function value is assumed to be constant at the function value specified for *Tn*. Numerous options exist for defining the response spectrum function. Among the more relevant to bridge design are AASHTO 2006 and AASHTO 2007: ▪ AASHTO 2006 – Specify values for the function damping ratio acceleration coefficient and select the soil profile type. Default values for period and acceleration are provided; alternatively, click the Convert To User Defined button to display a form that allows entry of the Period and Acceleration values. ▪ AASHTO 2007 – Based on the procedures described in AASHTO Guide Specifications for LRFD Seismic Bridge Design, Section 3.4.1, the Ss, S1 and PGA values may be taken from USGS and AASHTO maps contained within CSiBridge; in which case, define the bridge location using latitude and longitude or by specifying a zip code. Alternatively, enter the Ss, S1 and PGA values directly.

Table 6-3 Form Data - Loads > Functions > {Type} > Copy, Modify

Type	Data Form Parameters
	Other function types include: ▪ From File - Any response spectrum text file may be read into a CSiBridge model file. Requires a user-specified file name and damping ratio. Files may be in a frequency or period verses values format. ▪ AS1170 2007 – Constructed as specified in AS 1170.4:2007 clause 7.2(a). ▪ BOCA96 – Based on 1996 BOCA Section 1610.5.5. The response spectrum is constructed by plotting the model seismic design coefficient, Csm, versus the modal period of vibration, Tm. For a given period, Tm, the value of Csm is determined using Equation 11-3. ▪ Chinese 2002 – Specified using the maximum value for the seismic lateral influence factor, AlphaMax; the factor of seismic lateral influence, $alpha_1$, obtained from the 2002 Chinese Design Code response spectrum for the fundamental period; the seismic intensity, SI; the damping ratio, zeta (to adjust the shape of the response spectrum curve); the characteristic ground period, Tg, in seconds; the fundamental period, T1, multiplied by the period time discount factor, PTDF, before determining the value of $alpha_1$ from the 2002 Chinese Design Code response spectrum curve. ▪ Eurocode8 1998 – Constructed as described in 1998 Eurocode ENV 1998-1-1:1994 Section 4.2.2. The ordinates of the response spectrum are calculated using Equations 4.1 through 4.4, also in Section 4.2.2. The values of bo, TB, TC, TC k1, k2, and S are taken from Table 4.1 in 1998 Eurocode ENV 19981-1:1994 Section 4.2.2. Note that the value of these parameters depends on the specified subsoil class. ▪ Eurocode8 2004 – Constructed as described in EN 1998-1-1:2004 Section 3.2.25. The ordinates of the response spectrum are calculated using Equations 3.13 through 3.16 in EN 1998-1-1:2004 Section 3.2.2.5. The value of TB, TC, TC, and S are taken from Table 3.2 or 3.3 in 1998 Eurocode EN 1998-1-1:2004 Section 3.2.2.5. Note that the value of these parameters depends on the specified ground type and spectrum type. ▪ IBC 2006 – Based on procedures described in IBC2003 Section 1613.2.1.4 (ASCE 7-05 11.4). ▪ NBCC 2005 – Based on item 72 in Commentary J of the 2005 NBCC (Canadian).

Table 6-3 Form Data - Loads > Functions > {Type} > Copy, Modify

Type	Data Form Parameters
	▪ NBCC 95 – Based on item 44(a) in Commentary J of the 1995 NBCC. ▪ NEHRP97 – Based on the procedures described in 1997 NEHRP Section 4.1.2.6. ▪ NZS1170 2004 – Constructed as specified in NZS 1170.5:2004 Section 3.1.1. ▪ NZS4203 – Construction as specified in 1992 NZS42003 Section 4.6. ▪ UBC 94 – Based on Figure 16-3 of Chapter 16 of the 1994 UBC. The digitization of these response spectra is based on Section C106.2.1 in the 1996 SEAOC Recommended Lateral Force Requirements and Commentary (more commonly known as the SEAOC Blue Book). ▪ UBC 97 – Constructed as shown in Figure 163 in Chapter 16 of the 1997 UBC. See Tables 16Q and 16R in the 1997 UBC for typical input values. With a form displayed, depress the F1 key for context sensitive help. The additional type options are shown in the list in Figure 6-5.
Time History	Displays a Time History {Type} Function Definitions form. A time history function may be a list of time and function values or just a list of function values that are assumed to occur at equally spaced intervals. The function values in a time history function may be normalized ground acceleration values or they may be multipliers for specified (force or displacement) load patterns. A time history function may be defined for any time range, *t0* to *tn*. For all times before the first given time, *t0*, the function value is assumed to be zero. At time *t0*, the function quickly ramps up to the first specified function value. Be sure to specify a final point with zero function value if the function is to end at zero value. This is particularly important for acceleration records supplied in a file from other sources. Numerous options exist for defining the response spectrum function. A listing of the type options is shown in Figure 6-5. ▪ Cosine – The cosine time history function is a periodic function. A cosine function cycle starts at its positive maximum value (positive value of amplitude), proceeds to a value of zero, continues to its negative minimum value (negative value of amplitude), and returns to a value of zero gain, and finally returns to its positive maximum value again. ▪ From File – Uses a text file of time history function data.

Table 6-3 Form Data - Loads > Functions > {Type} > Copy, Modify

Type	Data Form Parameters
	▪ Ramp – A ramp function is defined by three points (time, function value). Those points, in order, are (0,0), (Ramp time, Amplitude) and (Maximum time, Amplitude). ▪ Sawtooth – A periodic function. A single cycle is defined by seven points (time, function values). Those seven points, in order, are (0,0), (Ramp time, Amplitude), (0.5×Period Ramp time, Amplitude), (0.5×Period, 0), (0.5×Period+ Ramp time, Amplitude), (Period Ramp time, Amplitude), and (Period, 0). ▪ Sine – A periodic function. A cycle starts at a function value of zero, proceeds to its positive maximum value (positive value of amplitude), continues to a value of zero, progresses to its negative minimum value (negative value of amplitude), and returns to a value of zero again. ▪ Triangular – A periodic function. A single cycle is defined by five points (time, function value). Those five points, in order, are (0,0), (0.25×Period, Amplitude), (0.5×Period, 0), (0.75×Period, Amplitude), and (Period, 0). ▪ User – Defines a time history function based on user specified time and function values. ▪ User Periodic – A periodic function. Defines a time history function based on user specified time and function values. With a form displayed, depress the F1 key for context sensitive help.

6.3.1 Response Spectrum Forms – Example Screen Capture

Use these commands (see Table 6-3) to display this form and define an AASHTO 2007 code-compliant response spectrum function.

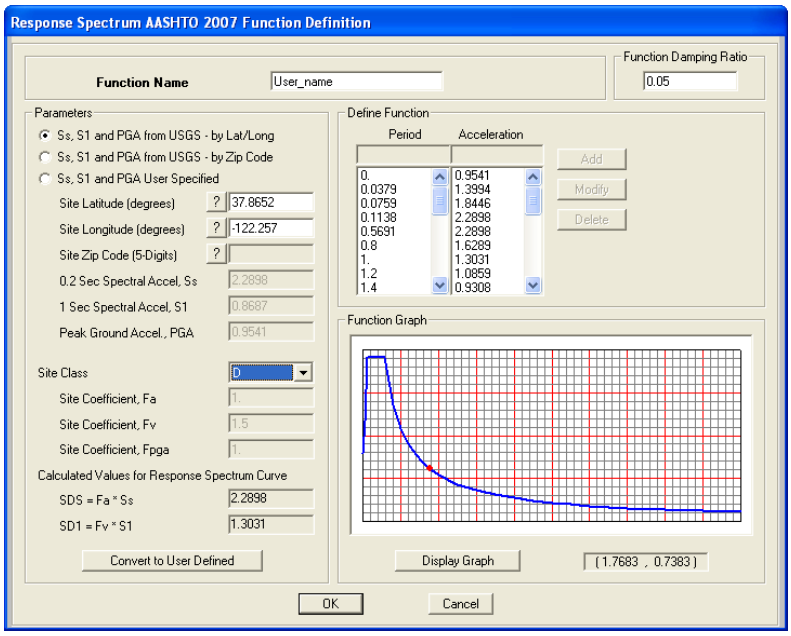
Loads > Type > Response Spectrum > Expand arrow on Functions – Response Spectrum panel > select **AASHTO 2007** from the **Choose Function Type to Add** drop-down list > **Add New Function** button

Or

Loads > Type > Response Spectrum > Add > select **AASHTO 2007** > click **OK**

Or

Loads > select a previously defined **AASHTO2007** function definition from the Current Functions drop-down list **> Copy, Modify**



6.3.2 Time History Forms – Example Screen Capture

Use these commands (see Table 6-3) to display this from and define a "from file" time history function.

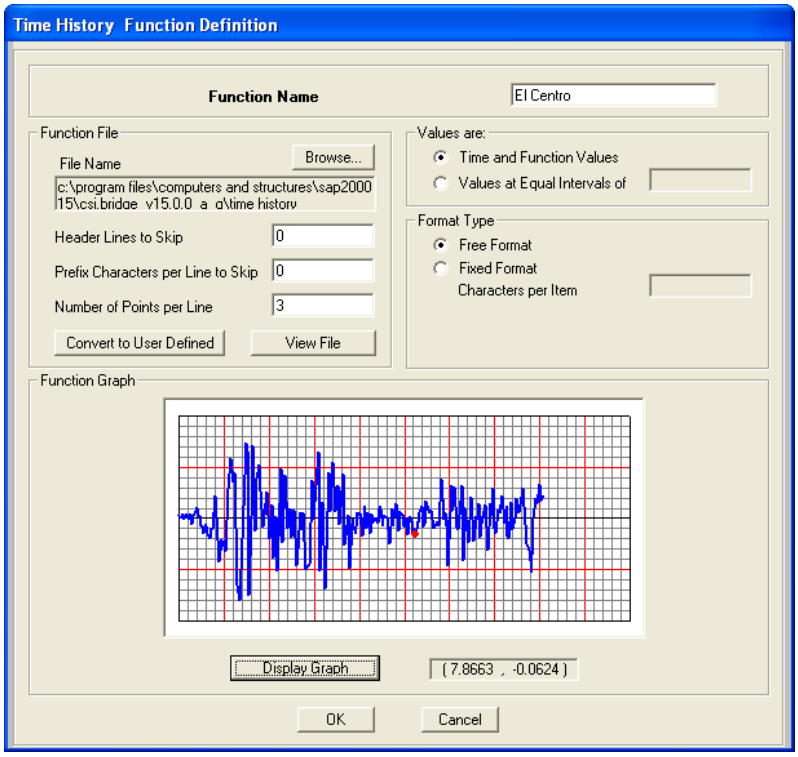
Loads > Type > Time History > Expand arrow on Functions – Time History panel > select **From File** from the **Choose Function Type to Add** drop-down list > **Add New Function** button

Or

Loads > Type > Time History > Add > select **From File** > click **OK**

Or

Loads > select a previously defined **From File** function definition from the Current Functions drop-down list **> Copy, Modify**



6.4 Loads > Loads

Point, line, area and temperature loads may be applied to the bridge model as part of the Bridge Object definition (see Chapter 7). Before the loads can be assigned, however, the loads must first be defined.

Figure 6-6 shows the commands on the *Loads* – {*Type*} panel of the **Loads** tab.

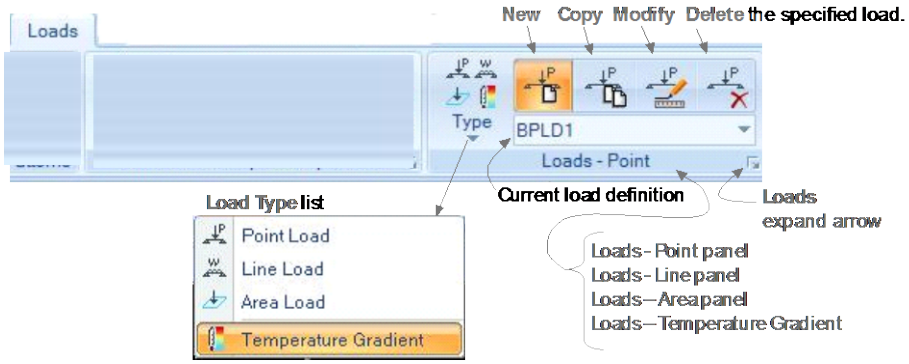
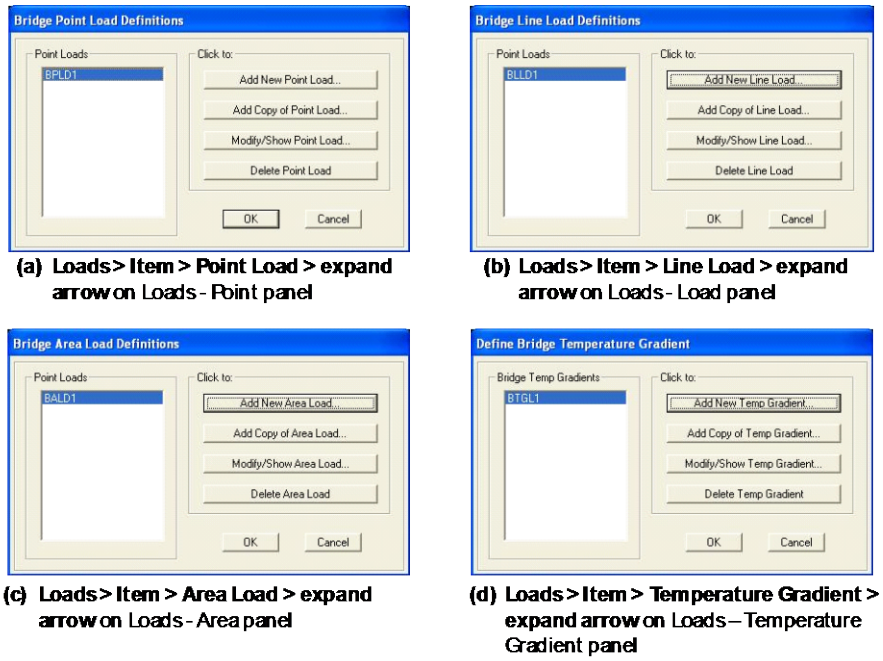


Figure 6-6 Loads tab with annotated Loads – {Type} panel

As suggested in the figure, clicking the **Loads Type** command displays a drop-down list that can be used to select Point, Line, Area, and Temperature Gradient loads. The name of the panel changes depending on the selection made. After a load has been selected, clicking the expand arrow displays the forms shown in Figure 6-7.

The forms shown in Figure 6-7a, b, c, and d list all previously defined bridge loads in display areas on the left-hand side of the forms. Generally, the buttons in the "Click to" area on the right-hand sides of the forms display the data forms described in Table 6-4 and shown in the screen captures in Sections 6.4.1 through 6.4.4. Referring to Figure 6-6, the **New**, **Copy**, and **Modify** commands on the *Loads* – {*Type*} panel bypass the definition forms shown in Figure 6-7, and display the same forms, thereby creating a shortcut to the data forms.

After the point, line, area, or temperature gradient load data have been defined, the {type} load may be assigned as a load case using the **Bridge > Bridge Object > Loads > {Point Load, Line Load, Area Load, Temperature Load}** command (see Chapter 7).



(a) Loads > Item > Point Load > expand arrow on Loads - Point panel

(b) Loads > Item > Line Load > expand arrow on Loads - Load panel

(c) Loads > Item > Area Load > expand arrow on Loads - Area panel

(d) Loads > Item > Temperature Gradient > expand arrow on Loads – Temperature Gradient panel

Figure 6-7 Definitions forms that show all previously defined loads

Table 6-4 Form Data - Loads > Loads > {Type} > Add, Copy, Modify

Type	Data Form Parameters
Point Load	Displays the Bridge Point Load Definition Data form shown in Section 6.4.1. Allows definition of a unique point load that has a user defined direction, value and location. The point load type can be specified as Force or Moment. The coordinate system may be Global or Local. When the Global coordinate system is used, the direction of the load as Gravity, X, Y or Z may be assigned. When the Local coordinate system is used, the direction may be assigned as Along the Horizontal Projection of the Layout Line (1), Vertical (2) or Normal to the Horizontal Projection of the Layout Line (3). The location of the point load in the transverse direction is made with reference to the left or right edge of the deck.

Table 6-4 Form Data - Loads > Loads > {Type} > Add, Copy, Modify

Type	Data Form Parameters
Line Load	Displays the Bridge Line Load Definition Data form shown in Section 6.4.2. Allows definition of a unique line load that has a user defined direction, value and location. The line load type can be specified as Force or Moment. The coordinate system may be Global or Local. When the Global coordinate system is used, the direction of the load as Gravity, X, Y or Z may be assigned. When the Local coordinate system is used, the direction may be assigned as Along the Horizontal Projection of the Layout Line (1), Vertical (2) or Normal to the Horizontal Projection of the Layout Line (3). The location of the line load in the transverse direction is made with reference to the left or right edge of the deck.
Area Load	Displays the Bridge Area Load Definition Data form shown in Section 6.4.3. Allows definition of a unique area load that has a user defined direction, value and location. The area load type can be specified as Force or Moment. The coordinate system may be Global or Local. When the Global coordinate system is used, the direction of the load as Gravity, X, Y or Z may be assigned. When the Local coordinate system is used, the direction may be assigned as Along the Horizontal Projection of the Layout Line (1), Vertical (2) or Normal to the Horizontal Projection of the Layout Line (3). The location of the line load in the transverse direction is made with reference to the left or right edge of the deck. Both distances are necessary to define a transverse boundary for the area load.
Temperature Gradient	Displays the Bridge Temperature Gradient Data form shown in Section 6.4.4. Allows definition of a unique temperature gradient that is based on compliance with AASHTO or Chinese JTG D60 codes, or that is defined by the user. The form includes a schematic illustrating the locations of input parameters for positive and negative temperature values. ▪ In the case of AASHTO or JTG D60 compliant definitions, those values are set and cannot be edited. ▪ In the case of a user definition, some or all of the values can be modified, depending on the Type of definition being specified (General—all values; AASHTO or JTG D60—select values).

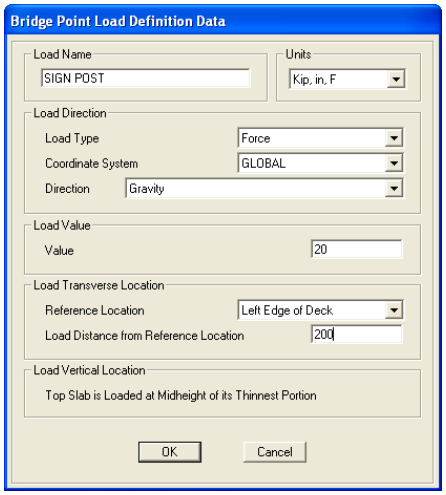
6.4.1 Point Load Form – Screen Capture

Use these commands (see Table 6-4) to display this form and define a point load.

Loads > Type > Point Loads > expand arrow on the *Loads – Point* panel > click **Add New Point Load** button

OR

Loads > Type > Point Loads > Add, Copy, Modify command



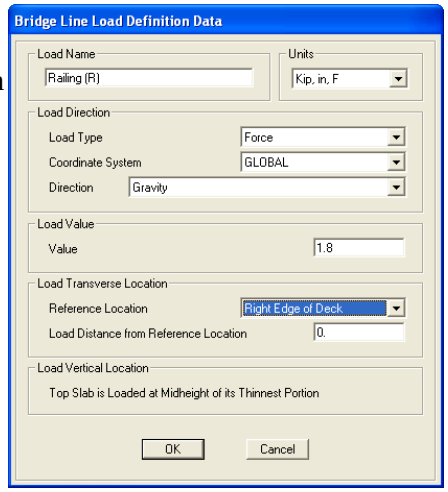
6.4.2 Line Load Form – Screen Capture

Use these commands (see Table 6-4) to display this form and define a line load.

Loads > Type > Line Loads > expand arrow on the *Loads – Point* panel > click **Add New Line Load** button

OR

Loads > Type > Line Loads > Add, Copy, Modify command



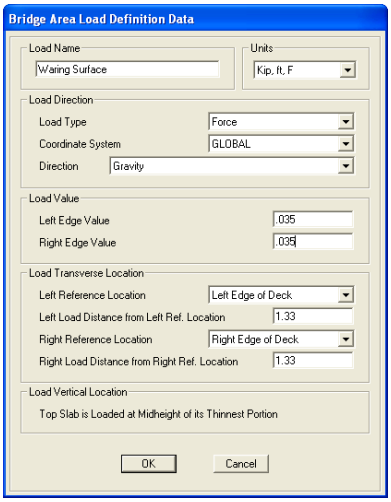
6.4.3 Area Load Form – Screen Capture

Use these commands (see Table 6-4) to display this form and define an area load.

Loads > Type > Area Loads > expand arrow on the *Loads – Area* panel > click **Add New Area Load** button

Or

Loads > Type > Area Loads > Add, Copy, Modify command

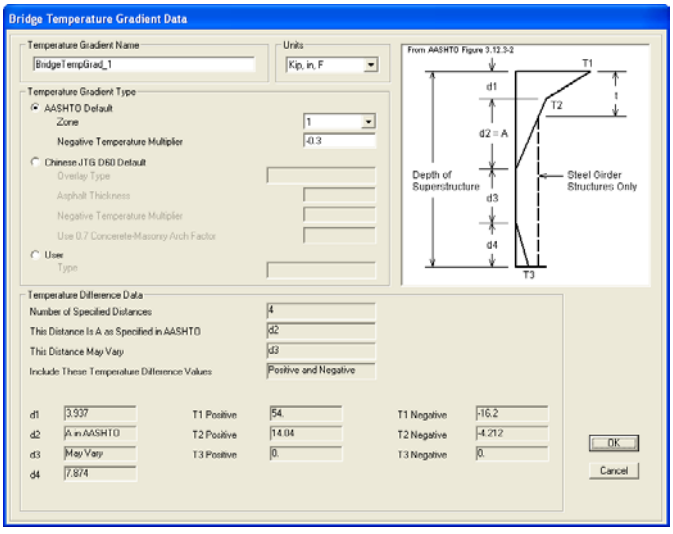


6.4.4 Temperature Gradient Form – Screen Capture

Use these commands (see Table 6-4) to display this form and define a temperature gradient load.

Loads > Type > Temperature Gradient > expand arrow on the *Loads – Temperature Gradient* panel > click **Add New Temp. Gradient** button

OR **Loads > Type > Temperature Gradient > Add, Copy, Modify** command



CHAPTER 7 Bridge

The **Bridge** tab consists of the commands that allow efficient access to the forms needed to add, copy, or modify Bridge Object definitions as well as delete a selected Bridge Object definition and update a bridge model manually or automatically. Other commands provide access to the data forms needed to review spans, span items (diaphragms, hinges, descretization), supports (abutments, bents), superelevation, prestress tendons, and loads as part of the process of assigning them to the specified Bridge Object. A command on this tab can be used to add girder rebar, and another command can be used to specify bridge groups for use in staged construction analysis.

If the Quick Bridge template was used to start the bridge model, the program will have created a Bridge Object using default assignments, including spans, abutments, bents, and a bridge group for staged construction analysis. These assignments can be viewed using the commands on the **Bridge** tab.

If the ***Bridge Wizard*** is being used, highlighting the Bridge Object Assignments – Deck Sections, Discretization Points, Abutments, Bents, Hinges, Diaphragms, Superelevation, Prestress Tendon, Concrete Girder Rebar, Staged Construction Groups, Point Loads, Area Loads, Temperature Loads – and clicking the **Assign/Show Deck Sections**, **Assign/Show Disc. Points**, **Assign/Show Abutments**, **Assign/Show Bents**,

Assign/Show Hinges, **Assign/Show Diaphragms**, **Assign/Show Superelevation**, **Assign/Show Tendons**, **Assign/Show Girder Rebar**, **Assign/Show Staged Groups**, **Assign/Show Point Loads**, **Assign/Show Line Loads**, **Assign/Show Area Loads**, and **Assign/Show Temp Loads** buttons will display the same forms that are displayed when the commands on the **Bridge** tab are used**.**

The commands on the **Bridge** tab also can be used if the Blank option was used to start the model and the ***Bridge Wizard*** is not being used (i.e., the model is being built from scratch or by importing model data).

7.1 Bridge > Bridge Objects

A bridge model is represented parametrically with a set of high-level objects: layout (alignment) lines, spans, bents (pier supports), abutments (end supports), deck cross sections, diaphragms, prestress tendons, superelevation, groups, and more. Using assignments, the Bridge Object definition brings together these Layout, Component, and Load definitions in preparation for generating a spine, area object, or solid object model (see Section 7.2). Typically a single Bridge Object represents the entire structure, although multiple Bridge Objects may be needed if a model includes parallel structures, or if merges or splits are to be considered.

Figure 7-1 shows the *Bridge Object* panel commands on the **Bridge** tab.

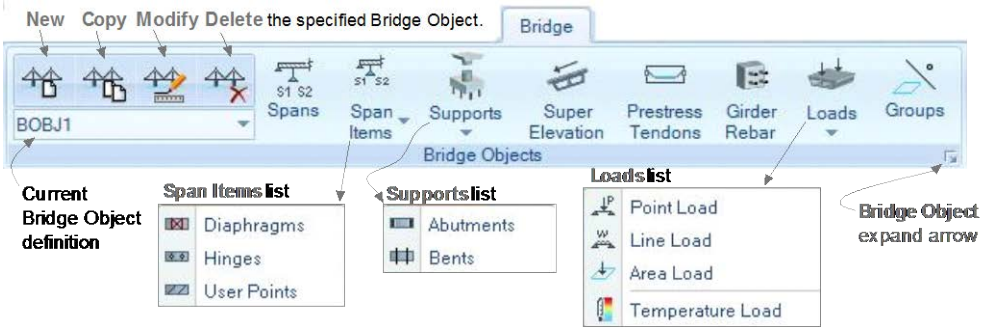
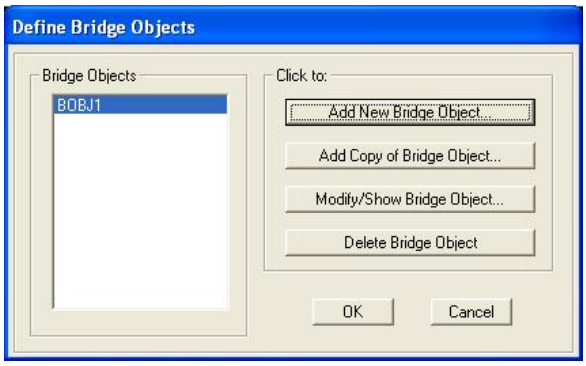


Figure 7-1 Bridge tab with annotated Bridge Object panel

Clicking the expand arrow in the lower right-hand corner of the *Bridge Object* panel displays the Define Bridge Objects form shown in Figure 7-2.



Bridge > Expand arrow on the Bridge Object panel

Figure 7-2 Definition form showing previously defined Bridge Objects

All of the previously defined Bridge Objects are listed on the left-hand side of the form. Clicking the buttons in the "Click to" area on the right-hand side of the form display the same form as is displayed when the **New**, **Copy**, and **Modify** commands on the *Bridge Object* panel are used. That form is shown in Figure 7-3.

The *Modify/Show Assignments* list on the right-hand side of the Bridge Object Data form (Figure 7-3) identifies the various assignments that can be made to a Bridge Object. Selecting an item and clicking the **Modify/Show** button displays the same forms as are displayed when the individual commands on the *Bridge Object* panel of the **Bridge** tab are clicked. That is, for example, clicking the Spans item in the *Modify/Show Assignments* list and then clicking the **Modify/Show** button on the Bridge Object Data form displays the same form as is displayed when the **Bridge > Bridge Object > Spans** command is used. Thus, the commands on the *Bridge Object* panel are intended to be used as shortcuts to the data/assignments forms. It may be helpful to define an initial Bridge Object using the *Modify/Show Assignments* list on the Bridge Object Data form and then use the individual commands on the **Bridge** tab to adjust the definition.

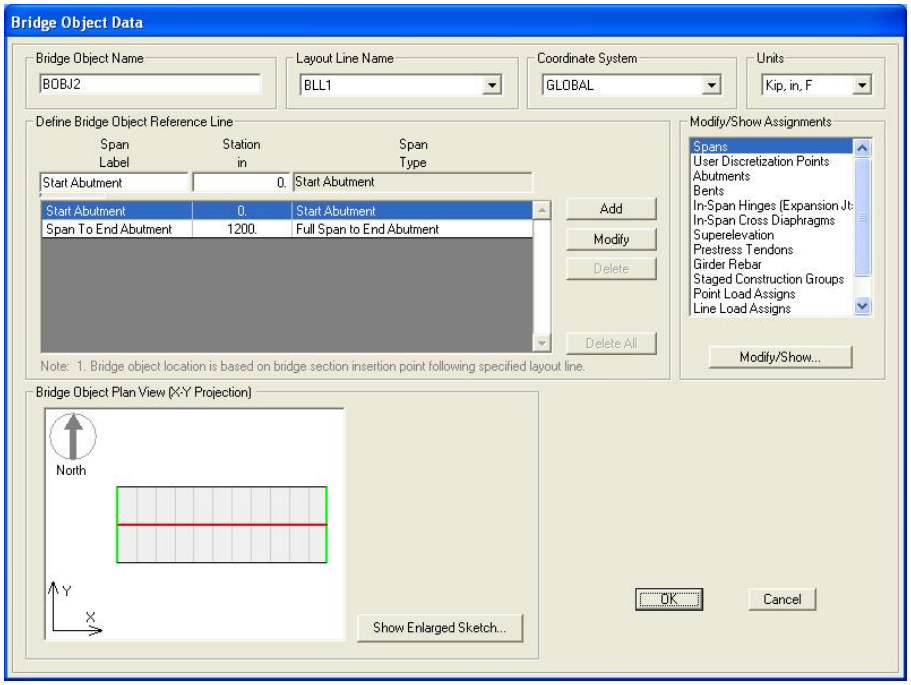


Figure 7-3 Bridge Object Data form that can be used to assign to the Bridge Object the definitions created using the Layout, Component, and Loads tabs

Table 7-1 briefly identifies the data/assignments available from the *Bridge Object* panel on the **Bridge** tab. The name that appears in the *Modify/Show Assignments* list on the Bridge Object Data form is shown in parenthesis in the *Command* column when the names differ.

Table 7-1 Form Data/Assignments - Bridge > Bridge Object > {Command}

Command	Data / Assignment / Definition Forms
Spans	Displays the Bridge Object Span Assignments form (see Section 7.1.1) That form is used to assign the deck section definition specified using the **Components > Item > Deck Sections** command. It is also possible to apply a parametric variation(s) (depress F1 to access context sensitive help, or see *Chapter 5 Components* for an explanation of parametric variations).
Span Items	Span Items are used to define the locations of in-span cross diaphragms, hinges, and user discretization points.

Table 7-1 Form Data/Assignments - Bridge > Bridge Object > {Command}

Command	Data / Assignment / Definition Forms
Span Items (continued) (In-Span Cross Diaphragm)	> **Diaphragms** – Displays the Bridge Object In-Span Cross-Diaphragm Assignments form (see Section 7.1.1). The in-span diaphragm assignment includes selection of the span to which the diaphragm is being assigned (along its length), identification of the diaphragm property, specification of the location of the diaphragm relative to the beginning of the span, and indication of the bearing (skew) measured in degrees relative to the bridge layout line.
[In-Span Hinges (Expansion Jts)}	> **Hinges** – Displays the Bridge Object Hinge Assignments form; use the buttons in the "Click to" area of that form to display the Bridge Object Hinge Assignment Data form (see Section 7.1.1). Use that data form to specify the hinge location and orientation; the bearing property, elevation, and rotation angle from the bridge default; the restrainer property and elevation; the diaphragm properties before and after the hinge; and the initial gap openings at the top and bottom of the superstructure. A **Modify/Show Overwrites** button is available that displays forms that can be used to overwrite the bearing and restrainer properties on a girder-by-girder basis.
(User Discretization Points)	> **User Points** – Displays the Bridge Object Discretization Points Assignments form (see Section 7.1.1). In most models it is not necessary to create user discretization points because the discretization specified when the linked model is updated generally is sufficient (see Section 7.2). However, because user discretization points supplement the discretization specified when the linked model is updated, this form provides a means of controlling the mesh points along the span. That is, user discretization points allow specification of points along the span where the bridge object will be discretized. Thus, user discretization points may be used if output at a specific point is needed. A skew also can be specified associated with the discretization point.
Supports	Abutments (end bents) and Bents (interior supports) can be assigned as part of a Bridge Object definition. > **Abutments** – Displays the Bridge Object Abutment Assignments form (see Section 7.1.2). Abutment assignments allow specification of the following items at the start and end of the bridge: end skews; end diaphragm property, if any; substructure assignment for the abutment, which may be none, an abutment property, or a bent property; vertical elevation and horizontal location of the substructure; and the bearing property, elevation and rotation angle from the bridge default. The elevation of the bearing refers to the action point of the bearing, which is the point at which all translations and rotations occur. Care should be taken to provide the proper bearing elevations (and restraint definitions) because

Table 7-1 Form Data/Assignments - Bridge > Bridge Object > {Command}

Command	Data / Assignment / Definition Forms
	of the kinematics that are captured by CSiBridge.
Supports (continued)	> **Bents** – Displays the Bent Object Bent Assignments form (see Section 7.1.2). The bent assignments allow specification of the following items for each bent: superstructure assignments, including diaphragm property (for bents at superstructure discontinuities, a diaphragm property can be specified on each side of the discontinuity, as well as a restrainer property, restrainer vertical elevation and initial gap openings at the top and bottom of the superstructure); bent property and bent orientation; vertical elevation and horizontal location of the bent; the bearing property, elevation and rotation angle from the bridge default (for bents at superstructure discontinuities, bearings are separately specified on each side of the discontinuity).
Super-elevation	Displays the Bridge Object Superstructure Assignment form (see Section 7.1.3). A superelevation assignment for a bridge object is referenced to the layout line. The superelevation is specified in percentage, and it indicates the rotation of the superstructure about its longitudinal axis. The superelevation may be constant, or it may vary along the bridge. In most bridge models including superelevation is probably an unnecessary refinement.
Prestress Tendons	Displays the Assign Prestress Tendons form, which displays all previously defined tendons and has buttons that provide access to the Bridge Tendon Data form (see Section 7.1.4). Use the data from to assign the following: the location of the start and end of the tendon; the vertical and horizontal layout of the tendon; tendon section properties, loss parameters and jacking options; tendon loads as a force or a stress; and modeling of tendons as loads or elements.
Girder Re-bar	Displays the Bridge Girders Reinforcement Layout form (see Section 7.1.5). Vertical and longitudinal girder reinforcing may be added to spans girder-by-girder. The girder rebar is used in the bridge rating calculations (see *Chapter 9 Design/Rating*). Transverse reinforcing is specified in terms of the area, spacing, number of spaces, and the start and end locations. Similarly, the longitudinal reinforcing is specified by the rebar area, and the distance from the top or bottom edge of the girder along with the start and end station locations along the length of the girder.
Loads	Point, line, area and temperature loads may be applied to the bridge model as part of the Bridge Object definition.
(Point Load Assigns)	**Point Loads** – Displays the Point Load Assignments form (see Section 7.1.6). Use that form to apply a defined point load to a defined load pattern as part of a Bridge Object definition (i.e., applies the load to the spans identified in the Bridge Object defini-

Table 7-1 Form Data/Assignments - Bridge > Bridge Object > {Command}

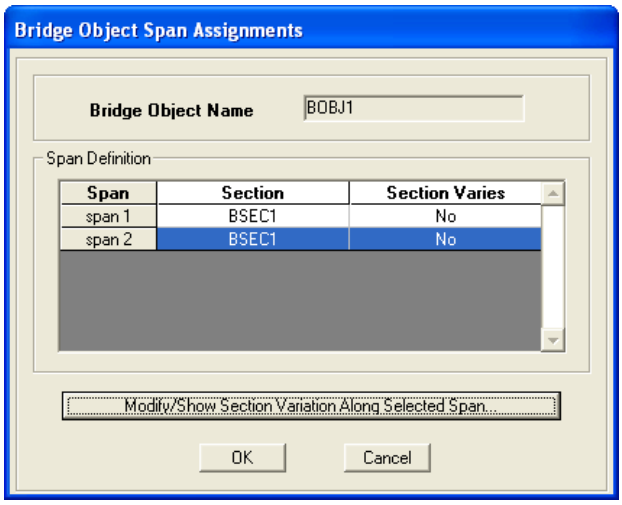
Command	Data / Assignment / Definition Forms
	tion).
Loads Point Loads (continued)	The point load and load pattern can be defined using the appropriate commands on the **Loads** tab (see *Chapter 6 Loads*), or using the forms that display when **Define Load Patterns** and **Define Point Loads** buttons on the Point Load Assignments form are used. A transverse variation can be applied using a parametric variation definition (use the F1 key to access context sensitive help or see *Chapter 5 Components* for more information about parametric variations). The load pattern can then be used in a load case (see *Chapter 8 Analysis*).
(Line Load Assigns)	> **Line Loads** – Displays the Line Load Assignments form (see Section 7.1.6) Use that form to apply a defined line load to a defined load pattern as part of a Bridge Object definition (i.e., applies the load to the spans identified in the Bridge Object definition). The line load and load pattern can be defined using the appropriate commands on the **Loads** tab (see *Chapter 6 Loads*), or using the forms that display when **Define Load Patterns** and **Define Line Loads** buttons on the Line Load Assignments form are used. A transverse variation can be applied using a parametric variation definition (use the F1 key to access context sensitive help or see *Chapter 5 Components* for more information about parametric variations). The load pattern can then be used in a load combination (see *Chapter 8 Analysis*).
(Area Load Assigns)	> **Area Loads** – Displays the Area Load Assignments form (see Section 7.1.6). Use that form to apply a defined area load to a defined load pattern as part of a Bridge Object definition (i.e., applies the load to the spans identified in the Bridge Object definition). The area load and load pattern can be defined using the appropriate commands on the **Loads** tab (see *Chapter 6 Loads*), or using the forms that display when **Define Load Patterns** and **Define Area Loads** buttons on the Area Load Assignments form are used. A transverse variation can be applied using a parametric variation definition (use the F1 key to access context sensitive help or see *Chapter 5 Components* for more information about parametric variations). The load pattern can then be used in a load combination (see *Chapter 8 Analysis*).
(Temperature Load Assigns)	**Temperature Loads** – Displays the Assign Bridge Temperature Loads form (see Section 7.1.6), which can be used to access the Bridge Temperature Load Assignments form. Use the latter form to define the temperature load assignment, which applies to the bridge superstructure. Assignments can be *uniform temperature* changes or *temperature gradient* changes over the height of the superstructure. Temperature gradient load assignments use predefined temperature gradient definitions, which are defined using

Table 7-1 Form Data/Assignments - Bridge > Bridge Object > {Command}

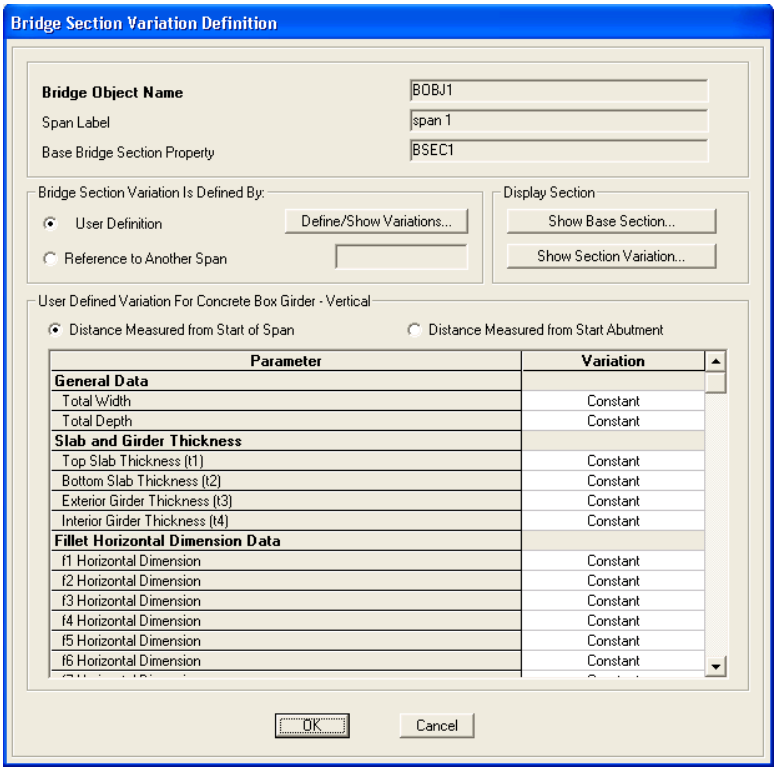
Command	Data / Assignment / Definition Forms
	the Bridge Temperature Gradient Data form (use the F1 key to access context sensitive help for that form or see *Chapter 6 Loads* for more information about that form).
Loads Tempera-ture Loads (continued)	Appropriate thermal loads are developed for the linked model (spine [frame], shell, or solid models). Those loads also can be automatically included in Load Combinations generated for AASH-TO or JTG design codes (see *Chapter 8 Analysis*).
Groups (Stage Construc-tion Groups)	Displays the Define Bridge Groups form. Clicking the Bridge Object Staged Construction Groups Assignments form (see Section 7.1.7). Use the form to add new groups to the model file and specify the bridge group type (e.g., section, top slab, web, bottom slab, beam, diaphragm/crossframe, support structure, bearing, hinge, tendon, mixed,). Groups can be used in defining a staged construction load case, or the Bridge Object to which the group has been assigned can be used in a seismic design request (e.g., foundation items). Depress the F1 key for context sensitive help with this form.

7.1.1 Bridge Object > Spans – Screen Captures

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Spans**



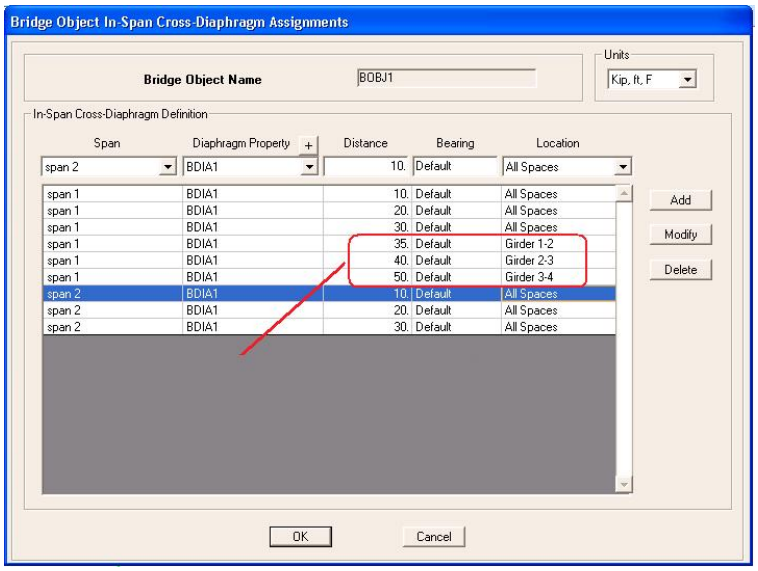
This command displays this form: **Bridge > Bridge Object > Spans > Modify/Show Section Variation Along Selected Span** button



7.1.2 Bridge Object > Span Items – Screen Captures

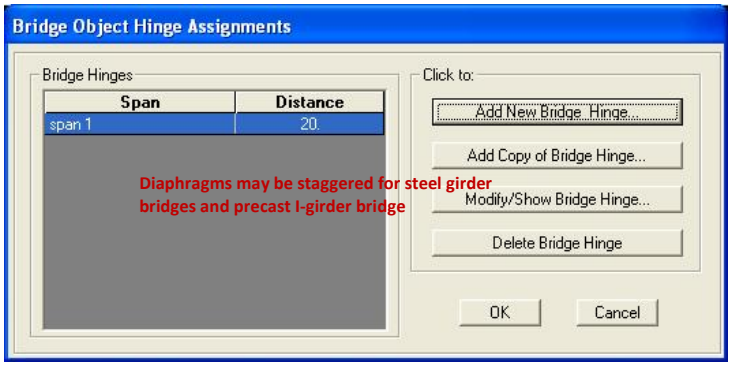
7.1.2.1 Bridge Object > Span Items > Diaphragms – Screen Capture

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Span Items > Diaphragms**

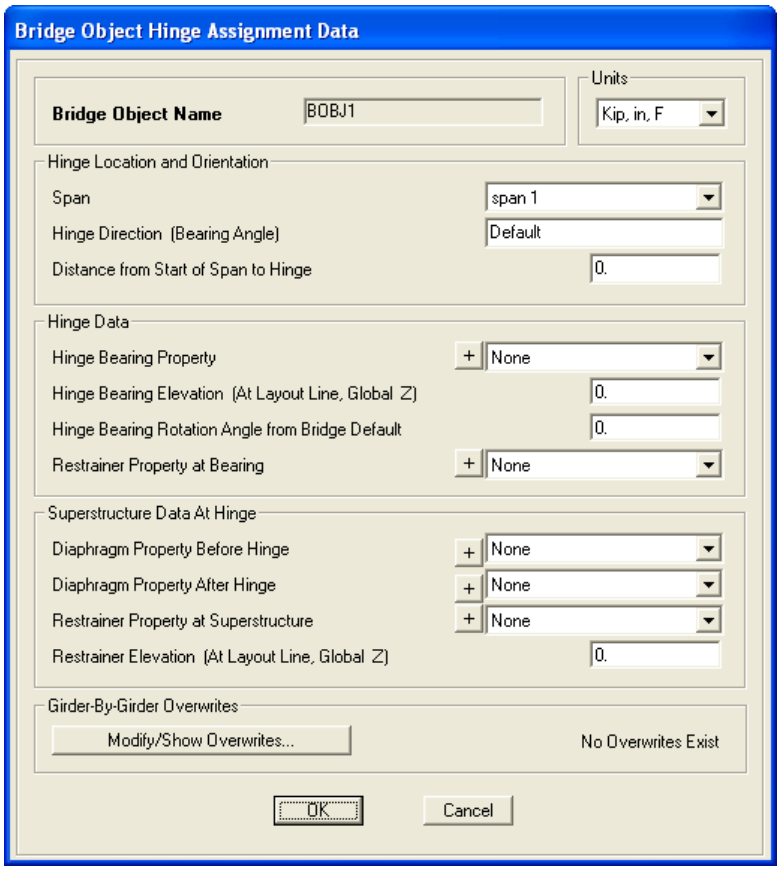


7.1.2.2 Bridge Object > Span Items > Hinges – Screen Captures

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Span Items > Hinges**

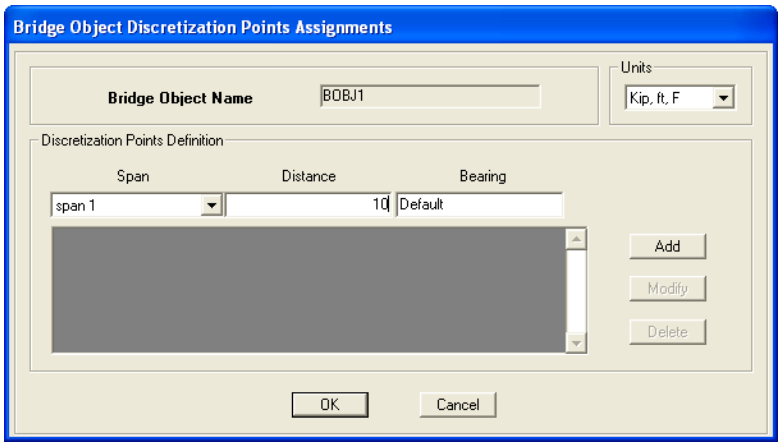


This command (see Table 7-1) displays this form: **Bridge > Bride Object > Span Items > Hinges > Add New Bridge Hinge** button



7.1.2.3 Bridge Object > Span Items > User Points – Screen Capture

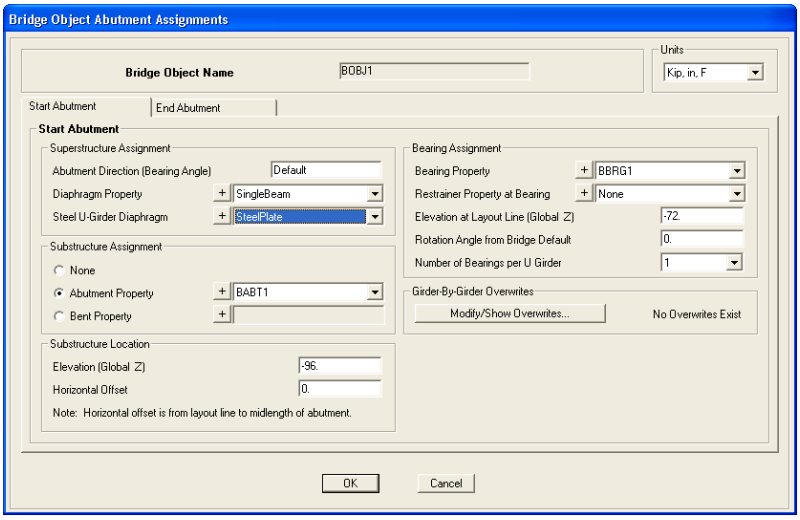
This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Span Items > User Points**



7.1.3 Bridge Object > Supports – Screen Captures

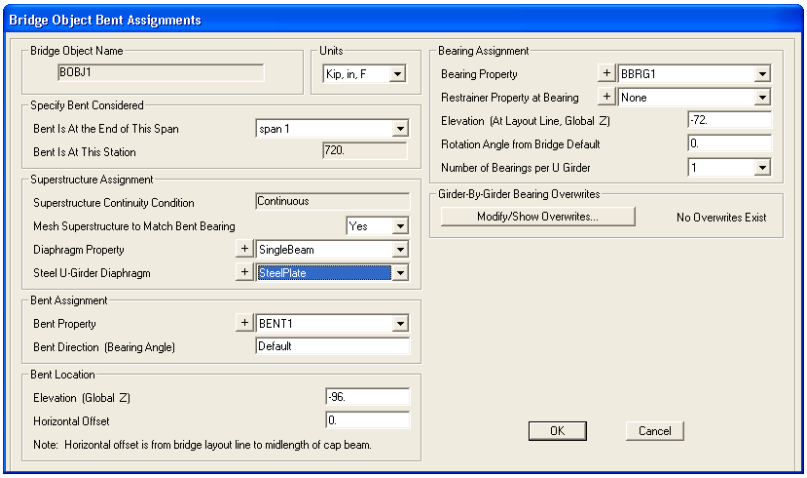
7.1.3.1 Bridge Object > Supports > Abutments – Screen Capture

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Supports > Abutments**



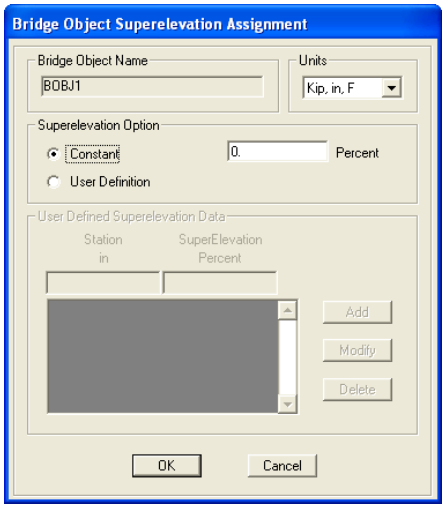
7.1.3.2 Bridge Object > Supports > Bents – Screen Capture

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Supports > Bents**



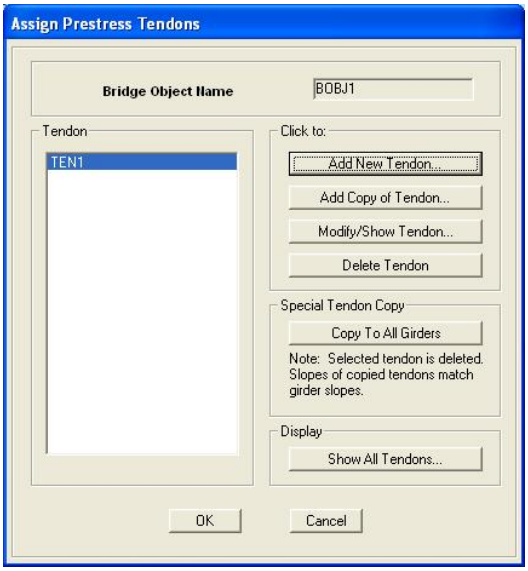
7.1.4 Bridge Object > Superelevation – Screen Capture

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Superelevation**

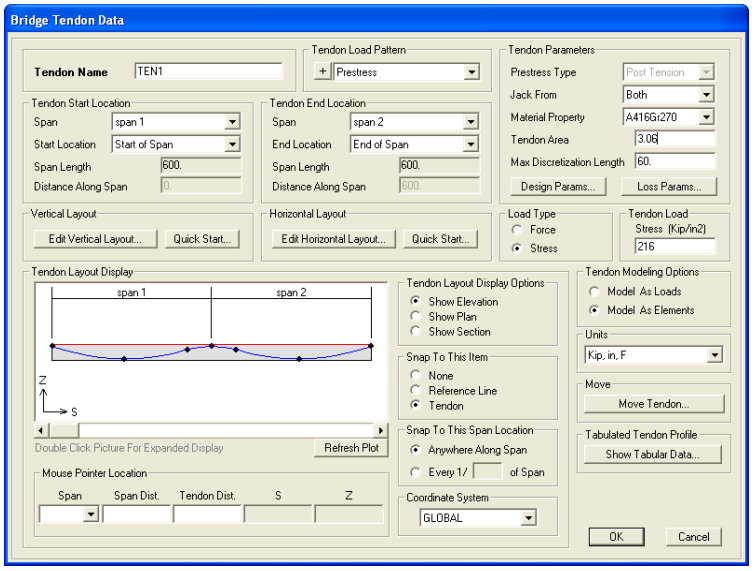


7.1.5 Bridge Object > Prestress Tendons – Screen Captures

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Prestress Tendons**

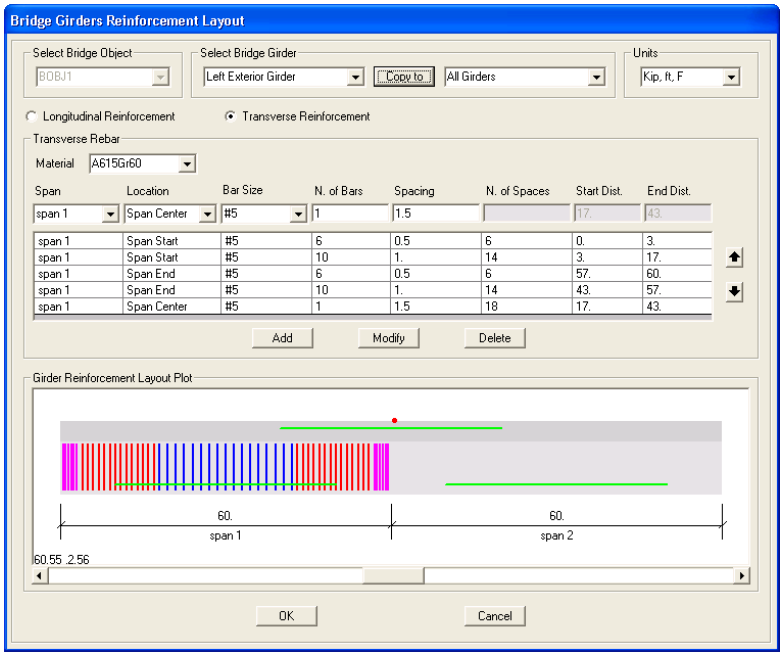


This command displays this form: **Bridge > Bridge Object > Prestress Tendons > Add New Tendon** button



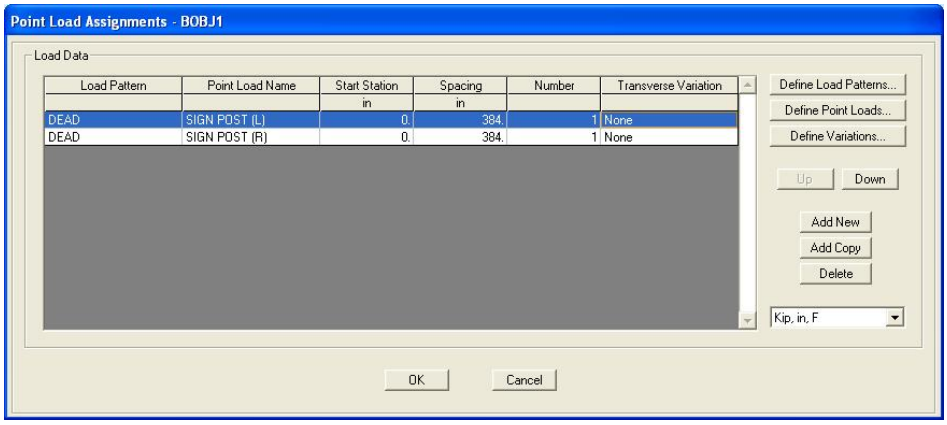
7.1.6 Bridge Object > Girder Rebar – Screen Capture

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Girder Rebar**

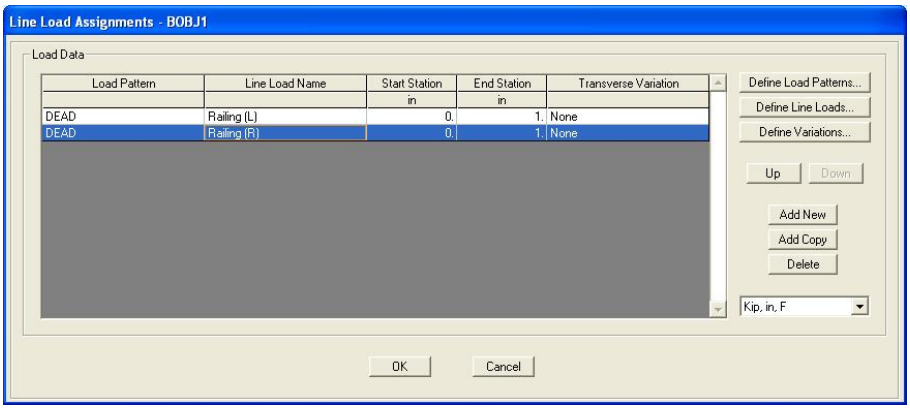


7.1.7 Bridge Object > Loads – Screen Captures

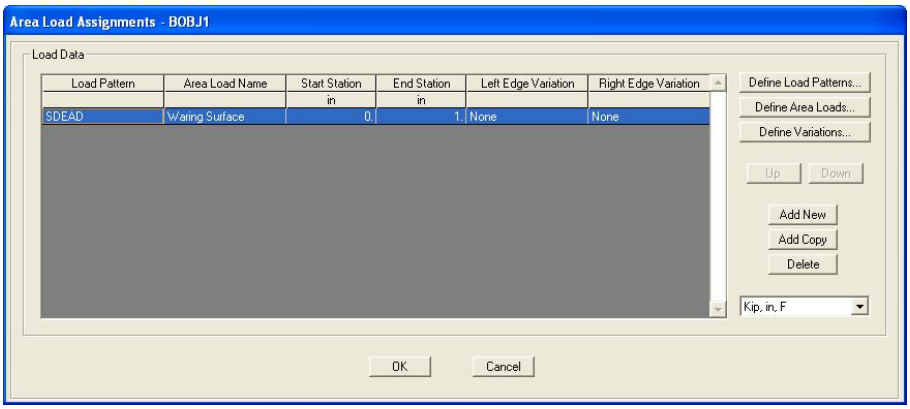
This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Loads > Point Load**



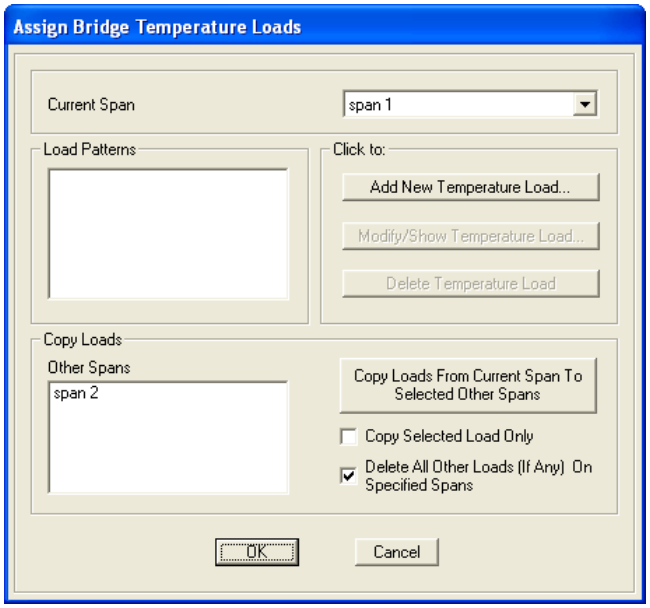
This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Loads > Line Load**



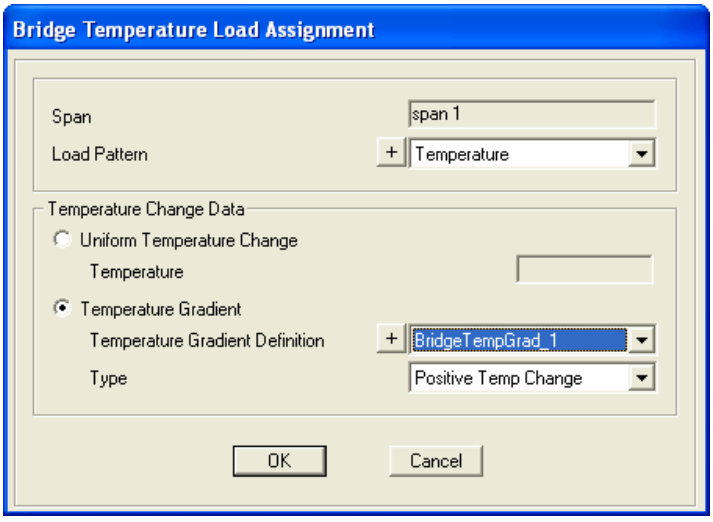
This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Loads > Area Load**



This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Loads > Temperature Load**

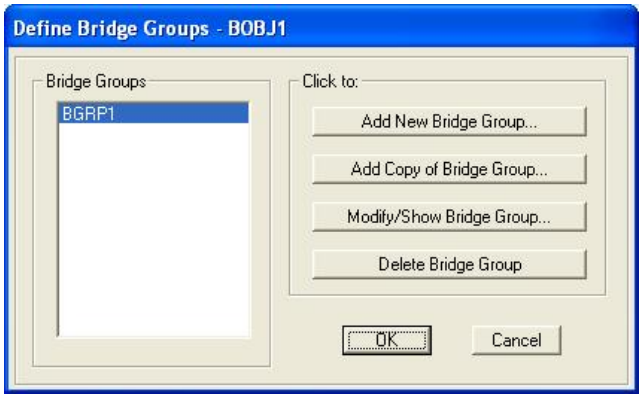


This command displays this form: **Bridge > Bridge Object > Loads > Temperature Load > Add New Temperature Load** button.

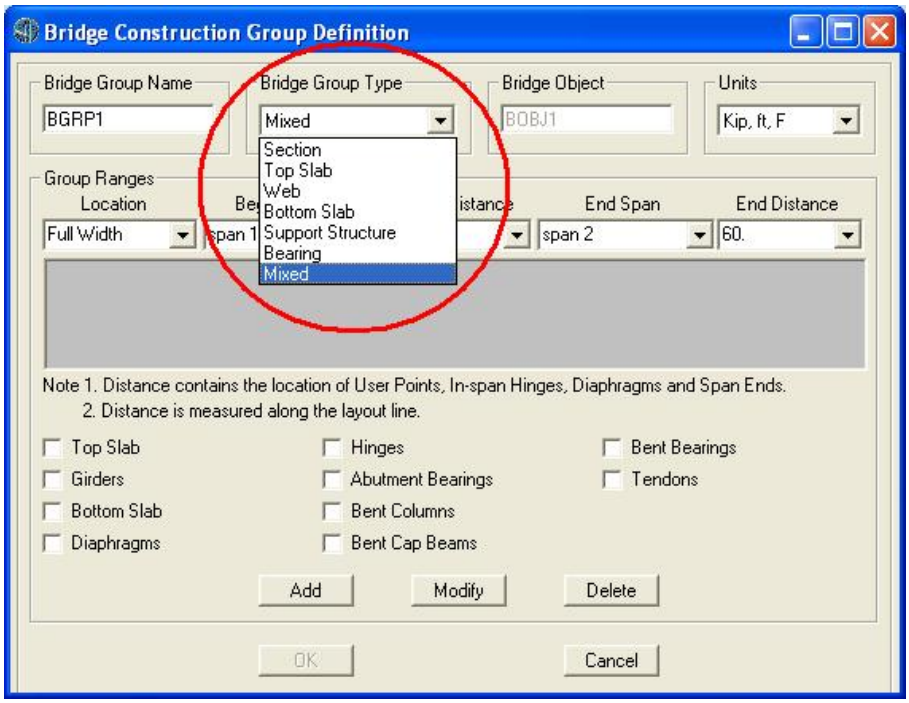


7.1.8 Bridge Object > Groups – Screen Captures

This command (see Table 7-1) displays this form: **Bridge > Bridge Object > Groups**

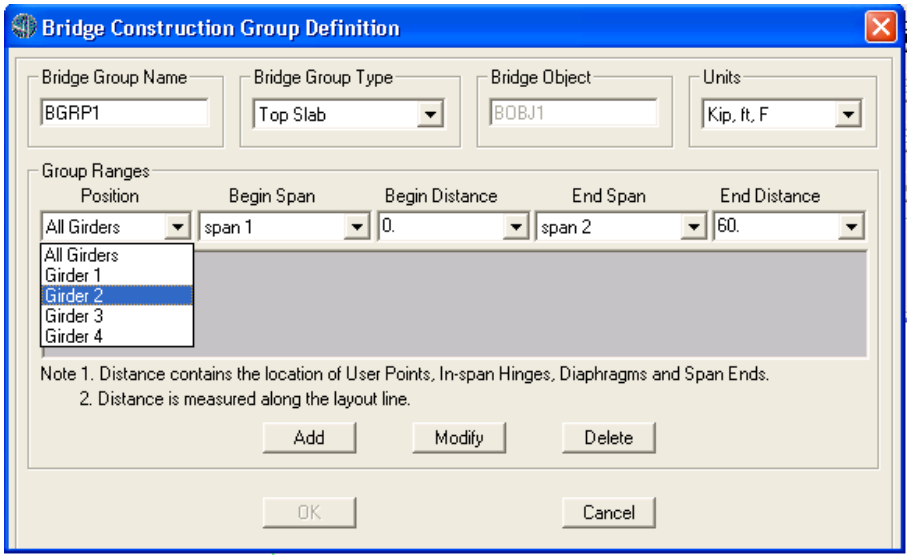


> Add New Bridge Group accesses this form:



Next, select a Bridge Group Type from the drop-down list (shown circled in the figure).

The Bridge Construction Group Definition form will change depending on the Bridge Object Type selected. Several deck section types allow for the group ranges to apply to specific girders, as noted in this figure by the circled text.



7.2 Update

After a Bridge Object has been defined, the object-based model used for analysis and design will not be assembled until the Bridge Object has been updated. The update process essentially creates or restores the object-based model based on the parametric bridge object definition.

Figure 7-4 shows the two commands on the *Update* panel.

Figure 7-4 Update panel on the Bridge tab

The next two subsections briefly describe the two commands on the *Update* panel

7.2.1 Update > Update

The first time the **Bridge > Update > Update** command is used for a given bridge object, CSiBridge creates a new object-based model for analysis and design from the parametric definition of the Bridge Object (see Section 7.1). This is done in accordance with the options selected on the Update Bridge Structure Model form which displays when the **Bridge > Update > Update** command is used, as shown in Figure 7-5.

If a linked object-based model of the selected Bridge Object already exists (i.e., the command has been used previously), the existing model is overwritten when the new object-based model is created. The new object-based model includes all of the latest changes to the Bridge Object definition. Note also, however, that the update operation may undo some changes to the object-based model if these changes were made without using the Bridge Object forms described in Section 7.1. For example, if edits are made to an updated bridge model using the draw or delete commands on the **Advanced** tab, those edits may not be reflected in the parametric Bridge Object definitions and therefore may be lost the next time the bridge model is updated. The details of how this is handled depend on the "Action" chosen.

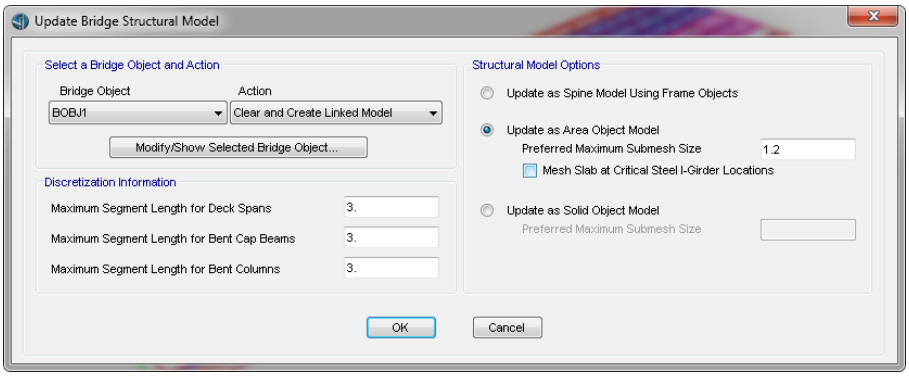


Figure 7-5 Update Bridge Structural Model form

The “Action” options on the Update Bridge Structural Model form include the following:

- Clear and Create Linked Model: This deletes all model objects previously created from the Bridge Object, if any, and then creates a new linked model. Any user modifications on previously generated model objects will not be preserved. CSiBridge will update the model as a spine, area-object, or solid-object model, depending on the option chosen on the right-hand side of the form. The type of object-based model created can be switched at any time. For the area- and solid-object models, the maximum sub-meshing size also can be specified. This option allows area and solid objects to be divided into smaller objects in the object-based model, and meshed into elements in the analysis model. Keep in mind that smaller mesh sizes require more memory and time when the analysis is run.

- Update Linked Model: When performed for the first time on the selected Bridge Object, this operation is the same as Clear and Create Linked Model described above. When the linked model was previously updated, CSiBridge will preserve user modifications on the previously generated objects, such as property modifiers, local axes, additional loads, etc., whenever possible. This can only occur when the generated object (joint, frame, shell, solid, link) is regenerated in the exact same location as before, and the user modification does not conflict with changes made to the Bridge Object parametric definition. User modifications would typically have been made using commands on the **Advanced** tab.

- Clear All from Linked Model: This deletes all model objects previously created from the Bridge Object. The Bridge Object itself is not affected and can be updated at a later time to create a new linked model. Alternatively, the Bridge Object can be deleted if it is no longer needed.

1. Convert to Unlinked Model: This is a one-way action that disconnects all model objects previously generated from the Bridge Object, but does not delete them. The Bridge Object itself is not affected and can be updated at a later time to create a new linked model.

If that is done, the generated objects could conflict with the unlinked model objects, so this should be done with care. Note that bridge display, design, and rating are not available for an unlinked model unless the Bridge Object is updated again to create a new linked model. Alternatively, the Bridge Object can be deleted if it is no longer needed.

The form also includes discretization options to specify the maximum segment length for deck spans, bent cap beams, and bent columns. These lengths determine the smoothness of curves for on-screen display and graphical printed output; the smaller the length, the smoother the curve.

For steel I-girder bridge with the girder web modeled as area objects, the option to mesh slab at critical steel I-girder locations is available. CSiBridge will subdivide the slab area objects at the critical locations such as girder staggered-diaphragm connections, girder staggered splices, and girder section-transition locations. Local girder section cuts will be generated at these critical locations in addition to the global bridge section cuts. The Bridge Response Display form will display additional girder responses at these critical location. This option is used for advanced steel I-girder bridge design and rating and is only available for steel I-girder bridges when the girder webs are modeled as area objects.

7.2.2 Update > Auto Update

The **Bridge** > **Update** > **Auto Update** command is a toggle. When enabled, CSiBridge will automatically Clear and Update the linked object-based model when any changes are made to the Bridge Object definition or its components.

CHAPTER 8 Analysis

The **Analysis** tab consists of the commands that allow efficient access to the forms needed to manage load case definitions; define a construction schedule that is useful when performing staged construction analysis of a bridge model; convert load combinations to nonlinear cases; show a tree of load case; specify results to be saved for all moving load cases; and run the analysis, including specifying the analysis options and displaying the results of the last analysis. The tab also has a command to unlock a locked model and lock an unlocked one, and another pair of commands that modify the geometry of a deformed shape and then reset the original geometry.

If the Quick Bridge template was used to start the bridge model, the program will have created default load case definitions.

If the ***Bridge Wizard*** is being used, highlighting the Load Cases, Construction Scheduler, and Moving Load Case Results Saved items in the Summary Table and clicking the **Define/Show Load Cases**, **Define/Show Schedule** and **Define/Show Results Save**d buttons will display the same forms that are displayed when the commands on the **Analysis** tab are used**.** The other commands available on the **Analysis** tab are not duplicated on the ***Bridge Wizard***.

The commands on the **Analysis** tab also can be used if the Blank option was used to start the model and the ***Bridge Wizard*** is not being used (i.e., the model is being built from scratch or by importing model data).

8.1 Analysis > Load Cases

A load case defines how loads are to be applied to the structure (e.g., statically or dynamically), how the structure responds (e.g., linearly or nonlinearly), and how the analysis is to be performed (e.g., modally or by direct-integration).

Figure 8-1 shows the *Load Cases* panel commands on the **Analysis** tab.

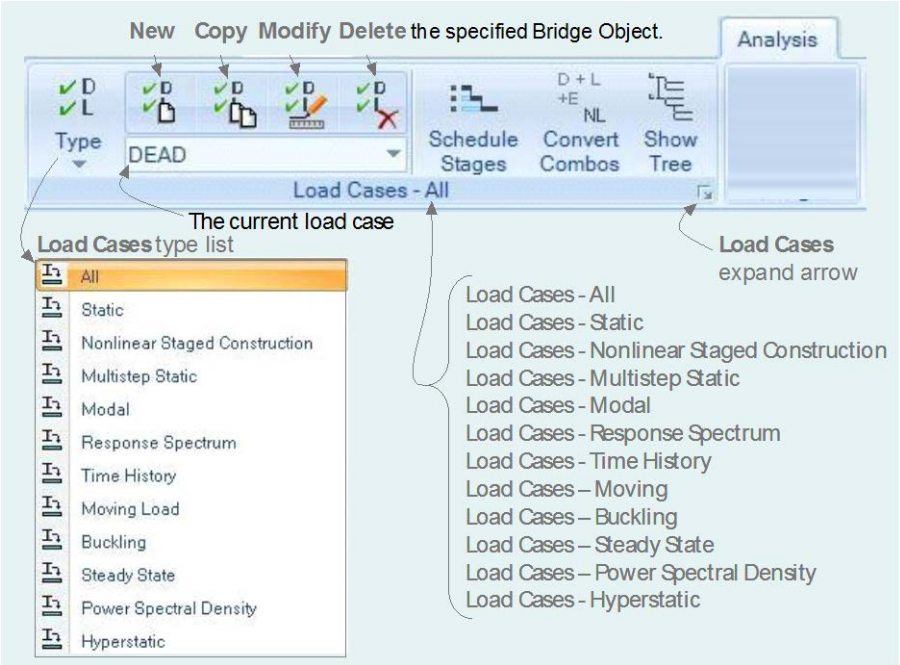


Figure 8-1 Analsis tab with annotated Load Cases panel

As shown in the figure, clicking the **Type** command displays a drop-down list of load case types. Any of these load case types can be used when analyzing a bridge model.

Static, response spectrum, and time history load case types are useful for seismic analysis. Pushover analysis can be performed using a nonlinear static load case. Stage construction analysis is also performed using nonlinear static load cases.

Several options are specialized for analyzing vehicle live loads. Moving load load cases compute influence lines for various quantities and solve all permutations of lane loading to obtain the maximum and minimum response quantities.

Multi-step static and multi-step dynamic (direct integration time history) load cases can be used to analyze one or more vehicles moving across the bridge at a specified speed. These multi-step load cases are defined using special bridge live load patterns that define the direction, starting time, and speed of vehicles moving along lanes (see *Chapter 6 Loads*).

To access the forms, select a Type and then click the **Add**, **Copy**, or **Modify** commands. In all cases, except the "All" type, the data form needed to complete the load case definition displays.

When the **Analysis > Load Cases > All** command is used, the Define Load Cases form displays. Note that clicking the **Analysis > Load Cases > Expand arrow** on the *Load Case* panel (arrow in the lower right-hand corner of the panel; see Figure 8-1) also displays this form, regardless of the Type selected. Figure 8-2 displays the Define Load Cases form. All previously defined load cases are listed on the left-hand side of the form.

The buttons in the "Click to" area on the right-hand side of the form display the Load Case Data form, defaulted to the Linear Static type. The *Load Case Type* drop-down list on the right-hand side of that form can be used to select a different load case type. The forms that can be accessed using that drop-down list are the same forms that display when the **Analysis > Load Cases > Type > {Select from list} > Add**, **Copy**, or **Modify** commands are used. Thus, the commands on the *Load Cases* panel bypass the need to use the Load Case Type drop-down list on the Load Case Data form, providing a shortcut to the data forms needed to define a particular load case type.

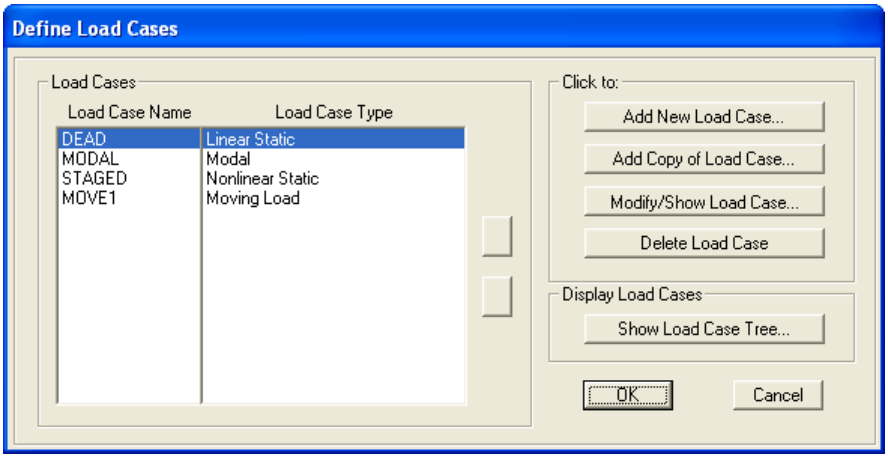


Figure 8-2 Definitions form showing all previously defined load cases

The **Schedule Stages**, **Convert Combos**, and **Show Tree** commands provide more direct access to the forms required to complete their designed functions. That is, these commands do not access a "summary" style of form that provides an indication of the previous activity completed using the commands. Note that by clicking the checkbox "Simulate Staged Construction", at the top right of the form is checked, the user can simulate the staged construction sequence without analyzing the model by clicking each stage under each staged construction load case.

Table 8-1 briefly explains the commands on the *Load Cases –{Type}* panel of the **Analysis** tab.

Table 8-1 Form Data - Analysis > {Command} >

Command	Data / Assignment / Definition Forms
Type > Add > Copy > Modify	Any of the following load case types can be used in the analysis of a bridge model. > **All** – Displays the Define Load Cases form. An explanation of this command precedes this table. > **Static** – Displays the Load Case Data – Linear Static form (see Section 8.1.1.1).The most common type of analysis, loads are applied without dynamic effects. > **Nonlinear Staged Construction** – Displays the Load Case Data – Nonlinear Static Staged Construction form (see Section 8.1.1.2). A direct integration time history analysis solves equations for the entire structure at each time step, whereas a modal

Table 8-1 Form Data - Analysis > {Command} >

Command	Data / Assignment / Definition Forms
	time history analysis uses the method of mode superposition. The nonlinear staged construction load case defines loads at user-specified stages for selected operations involving user-defined groups of model objects.

Table 8-1 Form Data - Analysis > {Command} >

Command	Data / Assignment / Definition Forms
Type > Add > Copy > Modify (continued)	> **Multistep Static** – Displays the Load Case Data – Linear Multi-Step Static form (see Section 8.1.1.3). A multi step static load case completes linear static analysis for multi-stepped load cases, making this load case useful in examining vehicle responses on a bridge. This method of applying vehicle loads to a bridge takes into consideration the vehicle direction and speed. It is possible to consider multiple vehicles in multiple lanes. The multi-step static load case does not, however, consider the dynamic effects. If dynamic effects are needed, a time history analysis may be used. Defining a multistep static load case requires that a Vehicle Live Load Pattern be defined and further that a multi step bridge live load pattern be generated (see *Chapter 6 Loads*). A multi step vehicle live load pattern is used to define the vehicle, lane, start location, start time, direction and speed. It should be noted that the vehicles used as part of a multi step static load case cannot have any uniform load assignments (see definition of vehicles in *Chapter 4 Layout*). Vehicles can have only axle loads. Generation of the multi step bridge live load pattern also defines the duration of the loading and the time step discretization. After the multi-step static analysis has been run, the results may be viewed by stepping through each time step. > **Modal** – Displays the Load Case Data – Modal form (see Section 8.1.1.4). A modal load case calculates the dynamic modes of the structure using Eigenvector or Ritz-vector methods. Loads are not actually applied, although they can be used to generate Ritz vectors. The Load Case Data - Modal form is used to view and change the definition of a modal load case. A modal analysis is always linear. Eigenvector analysis determines the undamped free-vibration mode shapes and frequencies of the system. These natural modes provide an excellent insight into the behavior of the structure. Ritz-vector analysis seeks to find modes that are excited by a particular loading. Ritz-vectors can provide a better basis than do eigenvectors when used for response-spectrum or time-history analyses that are based on modal superposition > **Response Spectrum** – Displays the Load Case Data – Response Spectrum form (see Section 8.1.1.5). A response spectrum load case statistically calculates the response caused by acceleration or displacement inertia loads. A response spectrum function must be defined before this load case can be used (see *Chapter 6 Loads*). > **Time History** – Displays the Load Case Data – Linear Modal History form (see Section 8.1.1.6). Time-varying loads are applied for this load case type. A time history function must be defined before this load case can be used (see *Chapter 6 Loads*). The solution may be by modal superposition or direct integration methods and the solution may be linear or nonlinear.

Table 8-1 Form Data - Analysis > {Command} >

Command	Data / Assignment / Definition Forms
> Time History (continued)	For bridge applications, a time history load case may be used to determine the dynamical response to a vehicle live load. After a Bridge Live load pattern has been defined (see information about load patterns in *Chapter 6 Loads*), a time history load case may be defined using the Vehicle Live load pattern name.
	> **Moving Load** – Displays the Load Case Data – Moving Load form (see Section 8.1.1.7). Vehicles, lanes and vehicle classes must be defined before a moving load load case can be defined (see *Chapter 4 Layout*). A moving load case calculates the most severe response resulting from vehicle live loads moving along lanes on the structure. The load case uses defined vehicle loads and defined lanes rather than the load patterns that are used by other analysis types. With Influence-based enveloping analysis, vehicles are moved in both directions along each Lane of the bridge. Using the influence surface, vehicles are automatically located at such positions along the length and width of the lanes so as to produce the maximum and minimum response quantities throughout the structure. Each vehicle may be allowed to act on every lane or be restricted to certain lanes. The program can automatically find the maximum and minimum response quantities throughout the structure resulting from the placement of different vehicles in different lanes. For each maximum or minimum extreme response quantity, the corresponding values for the other components of response also can be computed.
	Buckling – Displays the Load Case Data – Buckling form (see Section 8.1.1.8). A buckling load case calculates the buckling modes under the application of loads. Linear buckling analysis seeks the instability modes of a structure as a result of the P-delta effect under a specified set of loads. Each eigenvalue-eigenvector pair is called a buckling mode of the structure. The modes are identified by numbers from 1 to n in the order in which the modes are found by the program. The eigenvalue, λ, is called the buckling factor. It is the scale factor that must multiply the applied loads to cause buckling in the given mode. Any number of linear buckling Load Cases can be created. For each case a combination is specified of one or more Load Patterns or Acceleration Loads that make up the load vector $\boldsymbol{r}$. The number of modes to be found also may be specified as well as a convergence tolerance. It is strongly recommended that more than one buckling mode be sought, since often the first few buckling modes may have very similar buckling factors. A minimum of six modes is recommended.

Table 8-1 Form Data - Analysis > {Command} >

Command	Data / Assignment / Definition Forms
> Time History > Buckling (continued)	It is important to understand that buckling modes depend on the load. There is no one set of buckling modes for the structure in the same way that there is for natural vibration modes. Buckling for each set of loads of concern must be explicitly evaluated. For each linear buckling load case, it may be specified that the program use the stiffness matrix of the full structure in its unstressed state (the default) or the stiffness of the structure at the end of a nonlinear load case. The most common reasons for using the stiffness at the end of a nonlinear case are to include P-delta effects from an initial P-delta analysis, include tension-stiffening effects in a cable structure, and consider a partial model that results from staged construction > **Steady State** – Displays the Load Case Data – Steady State form (see Section 8.1.1.9). A steady state load case solves for the response of the structure resulting from cyclic (harmonic, sinusoidal) loading at one or more frequencies of interest. Steady-state analysis seeks the response of the structure at one or more frequencies to loading. After analysis, we can plot the deflected shape or force/stress response at any of the requested frequencies and at any phase angle. For example, the response at phase angle 0° primarily represents the behavior due to horizontal loading plus a damping component due to vertical loading. We can instead plot the magnitude of the response at any requested frequency, where the magnitude is given by the square-root of the sum of the squares of the real (0°) and imaginary (90°) response components. It is also possible to display plots of any response quantity as a function of frequency, yielding a frequency spectrum. This can be completed for the component at any phase angle, or for the magnitude of the response. > **Power Spectral Density** – Displays the Load Case Data – Power Spectral Density form (see Section 8.1.1.10). A power spectral density load case solves for the response of the structure resulting from cyclic (harmonic, sinusoidal) loading over a range of frequencies, and then integrates the resulting spectrum weighted by a probabilistic power spectral density function to get a root means square (RMS) expected response. > **Hyperstatic** – Displays the Load Case Data – Hyperstatic form (see Section 8.1.1.11). A hyperstatic load case calculates the linear response of the structure with all supports removed and loaded only by the reactions from another linear static load case. This is typically to calculate the secondary forces under prestress loading.
> Delete	Deletes the load case definition selected in the Current Load Case

Table 8-1 Form Data - Analysis > {Command} >

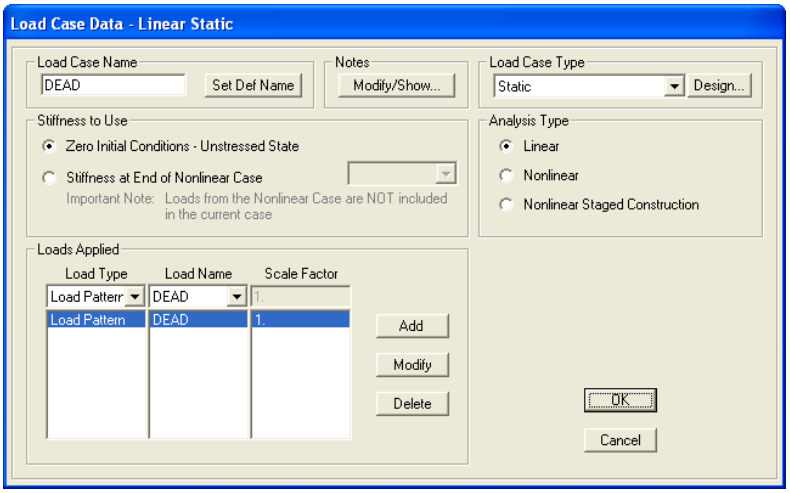
Command	Data / Assignment / Definition Forms
	drop-down list (see Figure 8-1).
Schedule Stages	Displays the Construction Scheduler forms (see Section 8.1.2). Use the cells in the rows and cells of the Schedule Table to identify the activity being scheduled, its duration, any tasks that must be completed before the current tasks can be completed (i.e., predecessors), and the Operation, which specifies how the task affects development of the structure with respect to staged construction analysis. A Stage is a collection of Operations that are executed at a given time. Each Stage has a defined duration in days that must be an integer greater than or equal to zero. Each Stage starts with initial conditions equal to the end of the previous Stage. The first stage will start with the initial conditions defined for the Staged-Construction Load Case. During the analysis, the model is first adjusted based on specified Operations and then analyzed for time dependent effects based on the duration of the Stage. If output is requested, it reflects the stress and deflection state of the structure at the end of the stage.
Convert Combos	Displays a selection list form (see Section 8.1.3). Use that form to specify which load combination(s) is to be converted to a static nonlinear load case. This command is a convenient way to create a nonlinear load while making use of an existing load combination when nonlinear effects, such as P-Delta, are required. Note that load combinations are created using the **Design/Rating > Load Combinations > Add, Copy, Modify** commands (see Chapter 9).
Show Tree	Displays the Load Case Tree form (see Section 8.1.4). This form shows the load cases that have been defined for the model file using the commands previously described in this table. Options on the form can be used to expand and view the various defined stages if a staged construction load case has been defined.

8.1.1 Analysis > Load Cases – Type

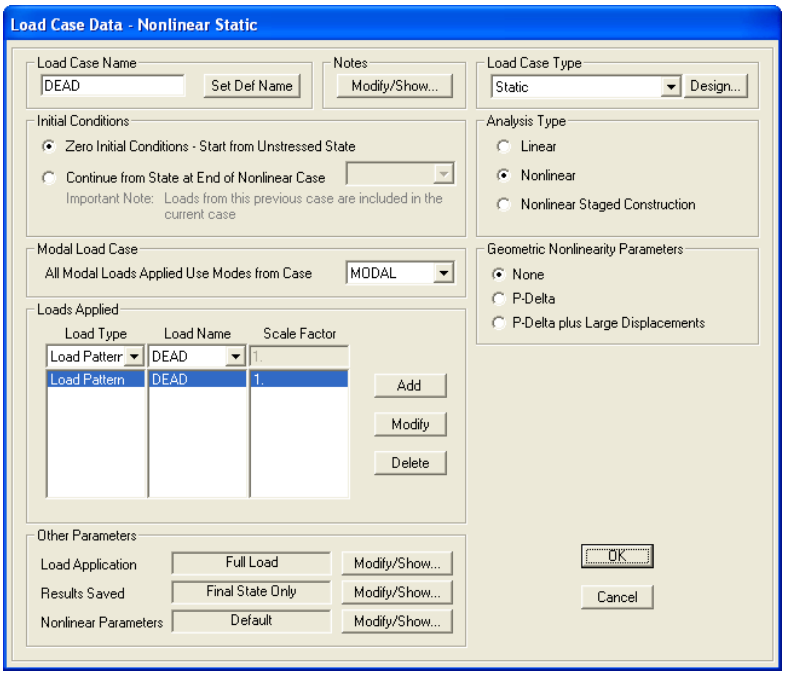
The following subsections provide screen captures of the forms that display when a Type of load case is selected on the Analysis tab and the **Add**, **Copy**, or **Modify** commands on the *Load Cases* – {*Type*} panel is used.

8.1.1.1 Static – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Static > Load Case Type: Static**

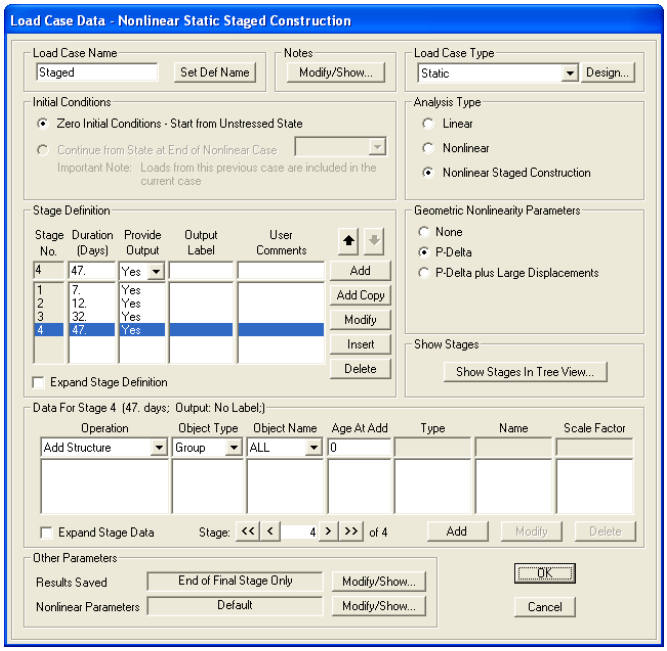


This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Static > Load Case Type: Nonlinear**.



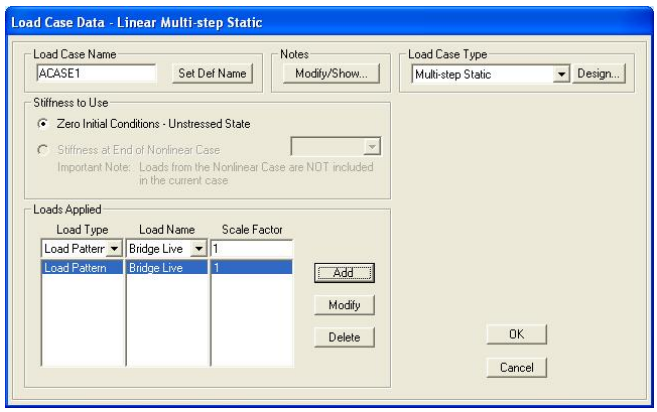
8.1.1.2 Nonlinear Staged Construction – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Nonlinear Staged Construction**



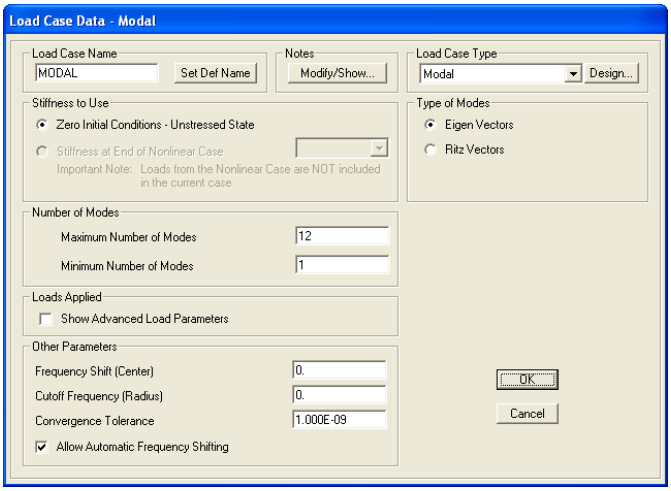
8.1.1.3 Multi-Step Static – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Multistep Static > Add, Copy, Modify**



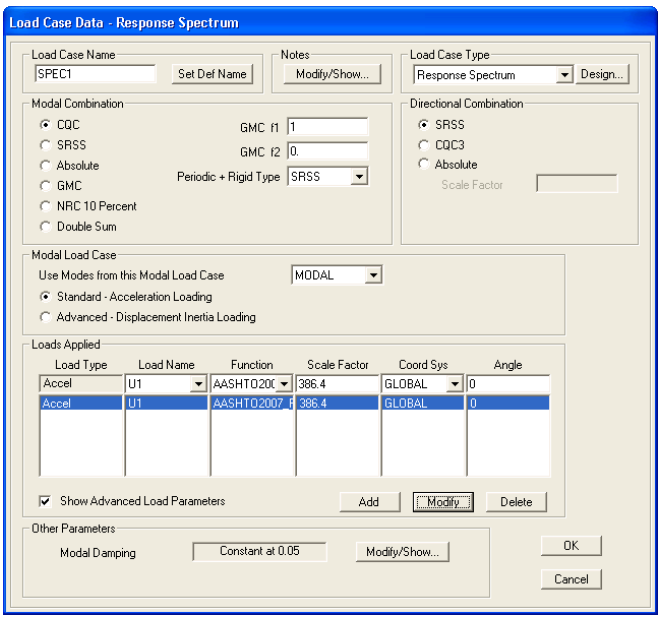
8.1.1.4 Modal – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Modal > Add, Copy, Modify**



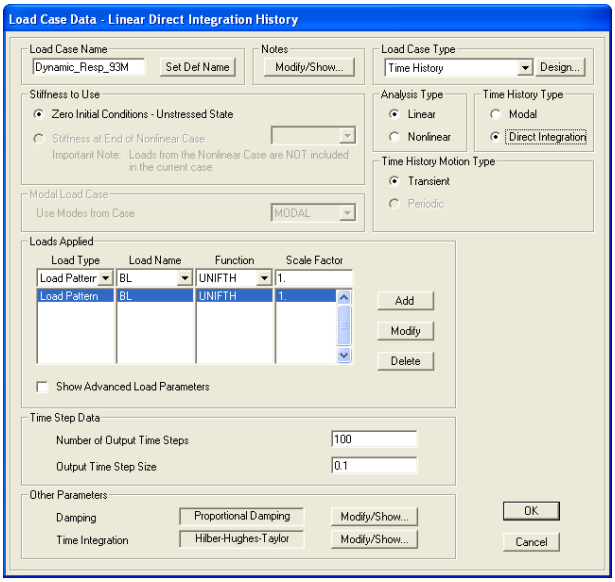
8.1.1.5 Response Spectrum – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Response Spectrum > Add, Copy, Modify**



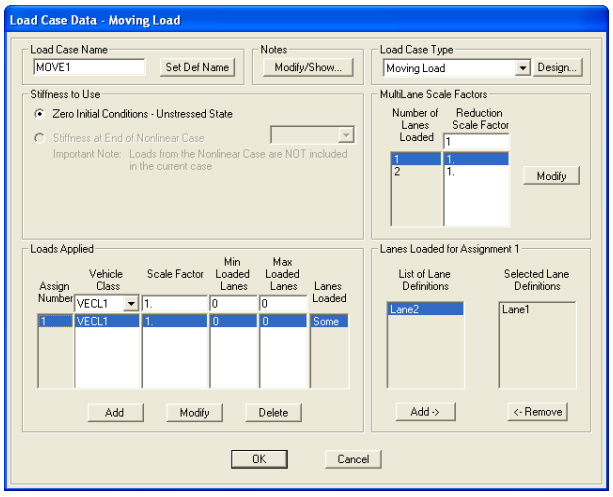
8.1.1.6 Time History – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Time History > Add, Copy, Modify**



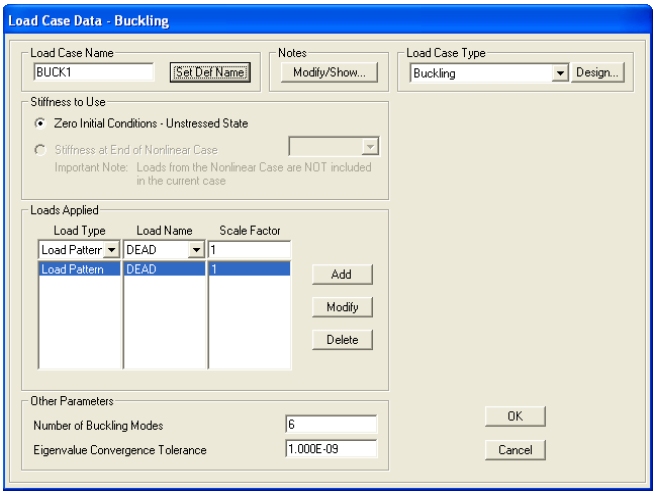
8.1.1.7 Moving Load – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Moving Load > Add, Copy, Modify**



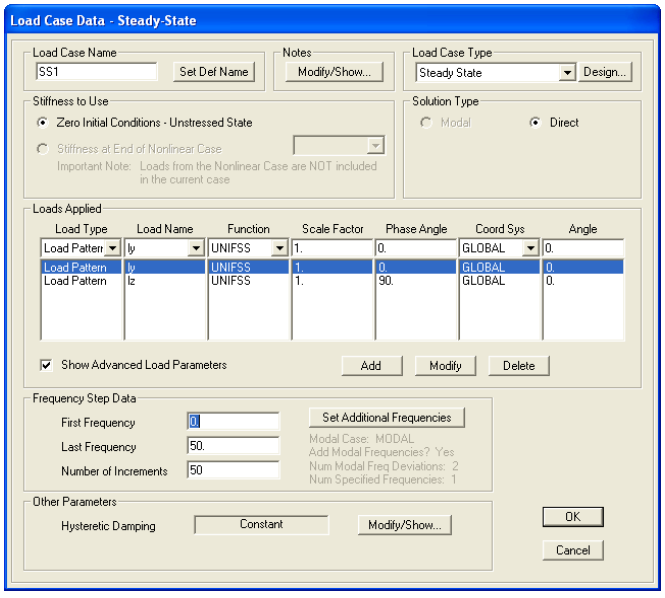
8.1.1.8 Buckling – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Buckling > Add, Copy, Modify**



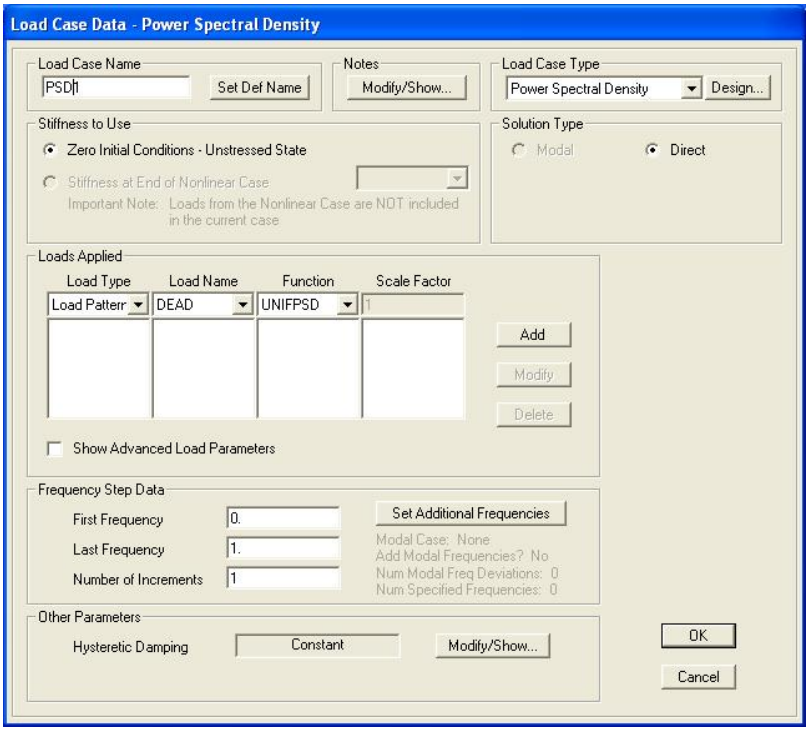
8.1.1.9 Steady State – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Steady State > Add, Copy, Modify**



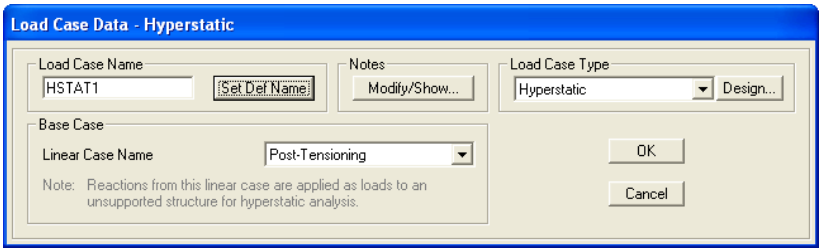
8.1.1.10 Power Spectral Density – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Power Spectral Density > Add, Copy, Modify**



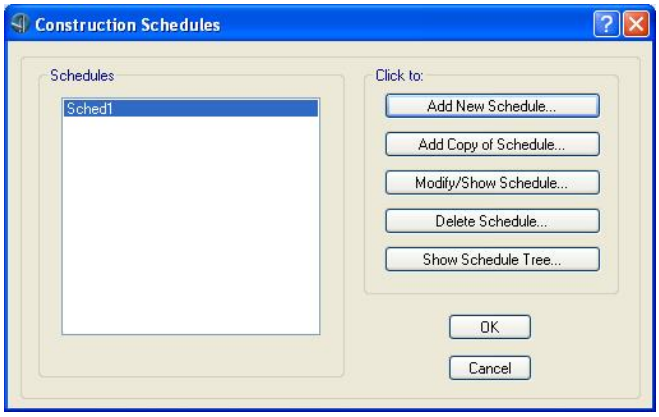
8.1.1.11 Hyperstatic – Screen Capture

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Type > Hyperstatic > Add, Copy, Modify**

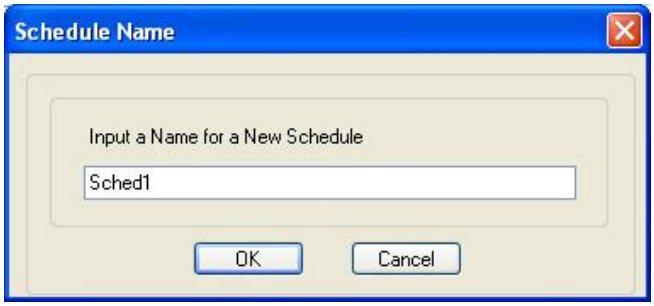


8.1.2 Analysis > Load Cases > Schedule Stages – Screen Capture

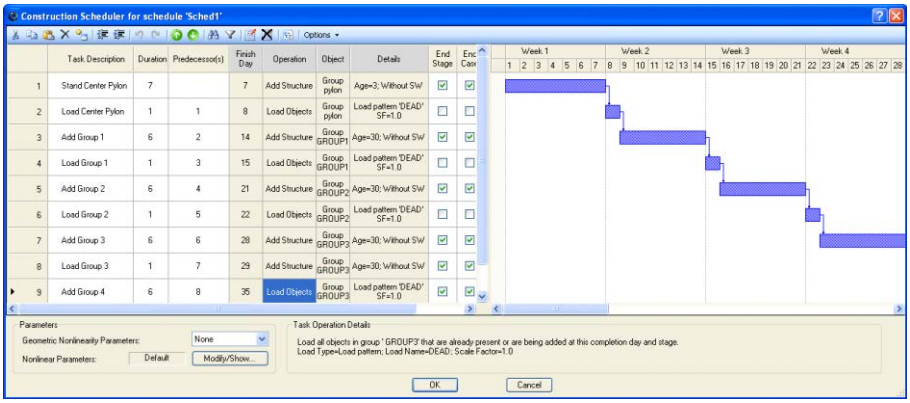
This command (see Table 8-1) display this form: **Analysis > Load Cases > Schedule Stages**



> Add New Schedule

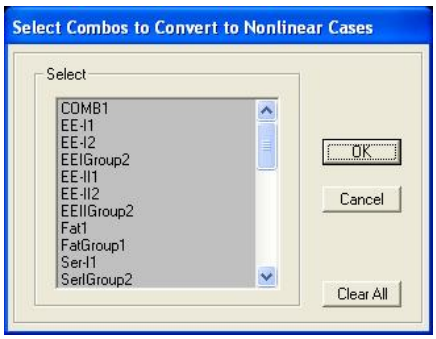


Specify a name, then **> OK**

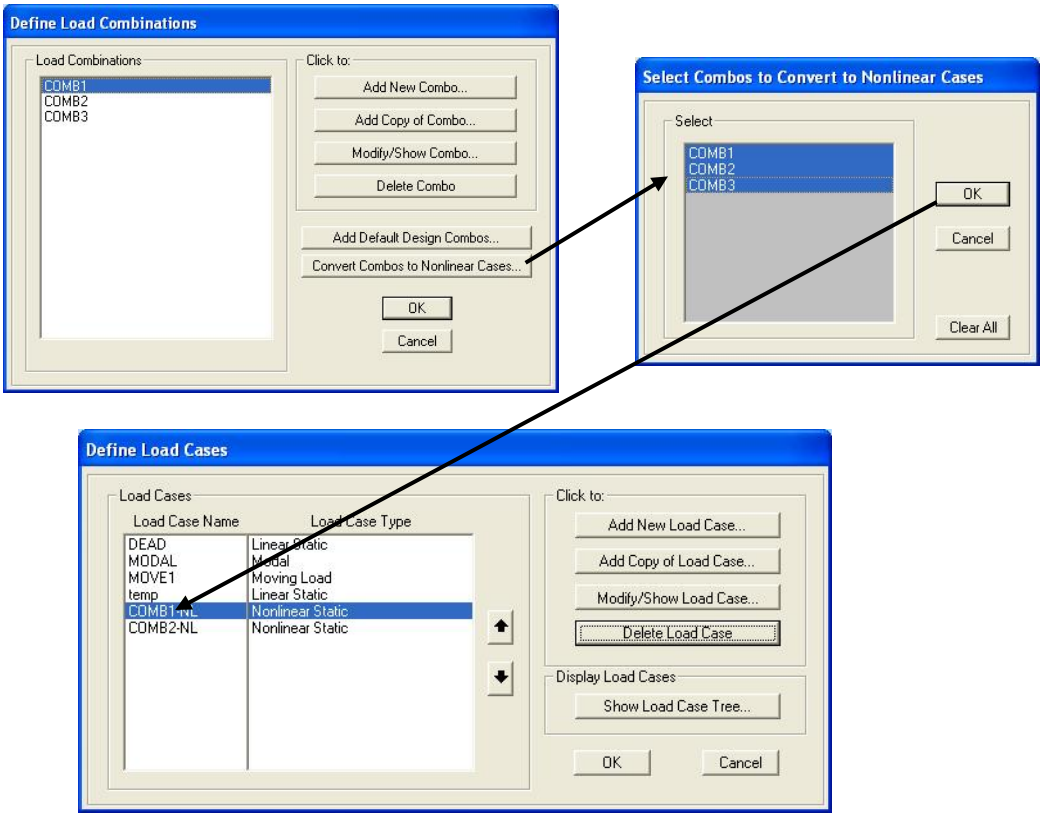


8.1.3 Analysis > Load Cases > Convert Combos – Screen Captures

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Convert Combos**

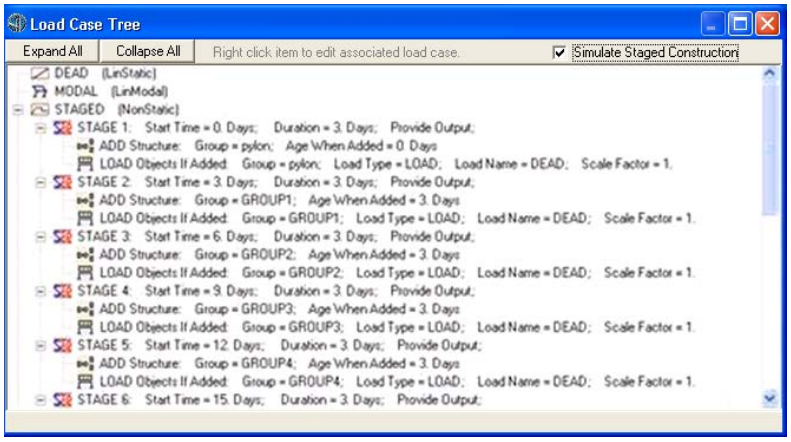


The following forms show combination names that have been converted (see *Chapter 9 Design/Rating* for more information).



8.1.4 Analysis > Load Cases >Show Tree

This command (see Table 8-1) displays this form: **Analysis > Load Cases > Show Tree**

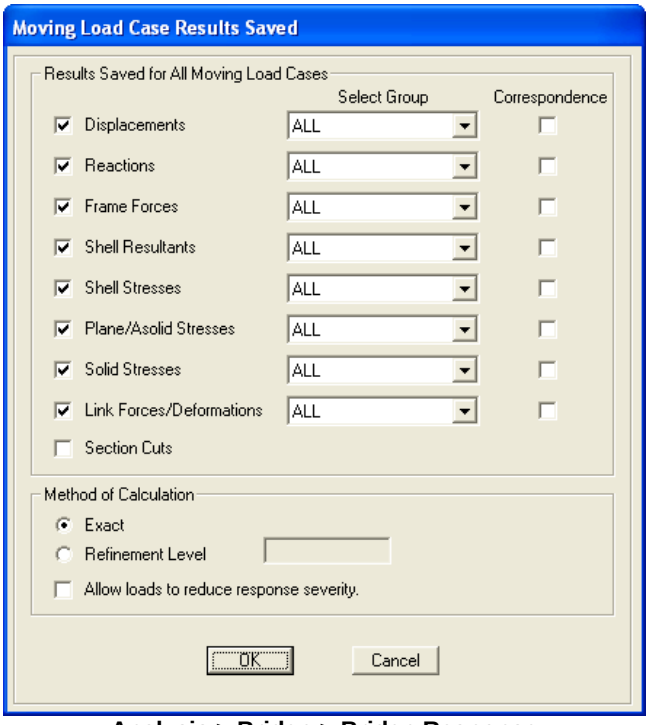


8.2 Analysis > Bridge

The *Bridge* panel on the Analysis tab has only one command: the **Bridge Response** command. Figure 8-3 shows the *Bridge* panel.

Figure 8-3 The Bridge Response command on the Bridge panel of the Analysis tab

This command displays the Moving Load Case Results Saved form, shown in Figure 8-4. The options on that form can be used to selectively control what information is calculated for joints and frame objects in the computationally intensive moving-load analysis.



Analysis > Bridge > Bridge Response

Figure 8-4 The Moving Load Case Results Saved form

As shown in Figure 8-4, response calculations can be limited to specific types of results (e.g., displacements, reactions, and so on), as well as to specified Groups of model objects, including the groups created by Bridge Object (see Chapter 7). An option exists to use the Max/Min Correspondence in design of frame sections when using moving loads; note that this is a very time-intensive operation so use it only when it is required. The method for calculating responses can be exact or with a specified degree of refinement, which is used to determine the shape of the influence line or area surface; this option provides "fast results" for preliminary review. The larger the integer, the greater the level of refinement. The integer reflects the number of points used to define the influence line or surface discretizations.

8.3 Analysis > Lock

The *Lock* panel on the **Analysis** tab has only one command: **Model Lock**. Figure 8-5 show the **Model Lock** command on the *Lock* panel.

Figure 8-5 The Model Lock command on the Lock panel of the Analysis tab

This command is a toggle, alternately locking and unlocking the model. Locking the model prevents changes being made to it. CSiBridge automatically locks the model after running an analysis to prevent any changes that would invalidate the analysis results. When the model is locked, commands on the **Layout**, **Components**, and **Loads** tabs and the **Bridge Object** > **Add**, **Copy**, **Modify** and **Delete** commands on the **Bridge** tab are not available. Load cases on the **Analysis** tab, however, may be used to define or modify a load case when the model is locked. If the definition of a load case that has already been run is changed, CSiBridge will display a warning that the analysis results will be deleted for that case and for all cases that depend on it. Design operations are performed on a locked model (see *Chapter 9 Design/Rating*). Commands that affect groups (**Bridge** tab), and views and display items (**Home** tab) are not locked. When a model for which analysis results are available is unlocked, CSiBridge will display a warning that unlocking the model will cause all analysis results to be deleted.

8.4 Analysis > Analyze

The *Analyze* panel of the **Analysis** tab has commands that provide direct access to the forms needed to select the analysis options, run the analy-

sis, and view the results of the last analysis run. Figure 8-6 shows the *Analyze* panel commands.

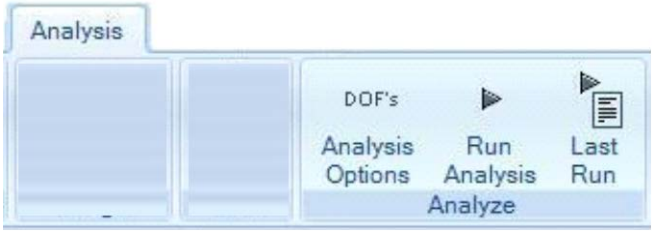


Figure 8-6 The commands on the Analyze panel of the Analysis tab

Table 8-2 Analysis > {Command} >

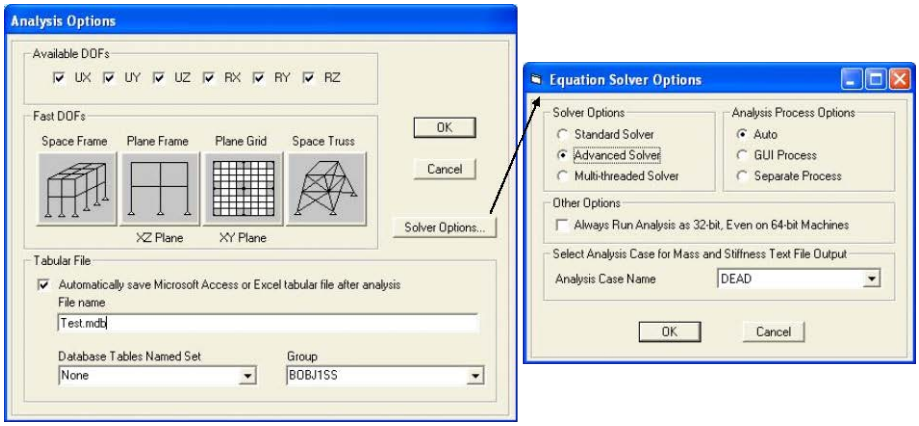
Command	Data / Assignment / Definition Forms
Analysis Options	Displays the Analysis Options form (see Section 8.4.1). Use that form to set the available degrees of freedom (DOF). If the DOF is set as available (i.e., the direction – UX, UY, YX, RX, RY, RZ – is preceded by a check mark on the form), displacements are possible in that direction at every joint. The occurrence of a displacement at a given joint will depend on the structure, loading, and restraint conditions at the joint. If the DOF is not available (i..e, the direction is not checked on the form), no displacements are possible in that direction at any joint. Effectively, every joint is restrained in that direction. Buttons on the form can be used to quickly set the six available DOF's to correspond to common structure types. CSiBridge offers equation solving options (click the Solver Options button on the form). The advanced equation solver can be one or two orders of magnitude faster than the stand solver for larger problems, and it also uses less disk space. Because the two solvers perform numerical operations in a different order, it is possible that sensitive problems may yield slightly different results because of numerical roundoffs. In extremely sensitive, nonlinear, history-dependent problems, the difference may be pronounced. The advanced solver is based on proprietary CSi technology that uses, in part, code derived from the TAUCS family of solvers. An option also is included to automatically save Microsoft Access or Excel tabular files after each analysis using a specified filename for a selected set of database tables and grouped objects.
Run Analysis	Displays the Set Load Cases to Run form (see Section 8.4.2). As the name suggests, use the form to specify the load cases to be included in the analysis run. The form also can be used to delete results for a selected load case or for all load case.

Table 8-2 Analysis > {Command} >

Command	Data / Assignment / Definition Forms
Last Run	Displays the Analysis Window (see Section 8.4.3), which displays the most recent analysis results. The form indicates the filename, the start and finish times for the analysis run, the elapsed time, and the run status (e.g., Done – Analysis Complete).

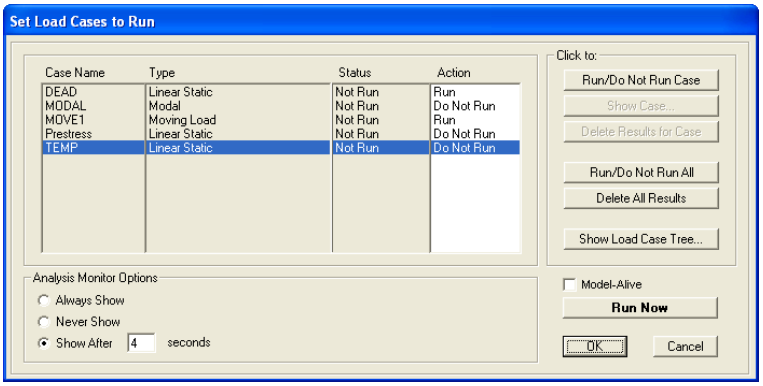
8.4.1 Analyze > Analysis Options – Screen Captures

This command (see Table 8-2) displays these form: **Analysis > Analyze > Analysis Options;** click the **Solver Options** button to display the second form.



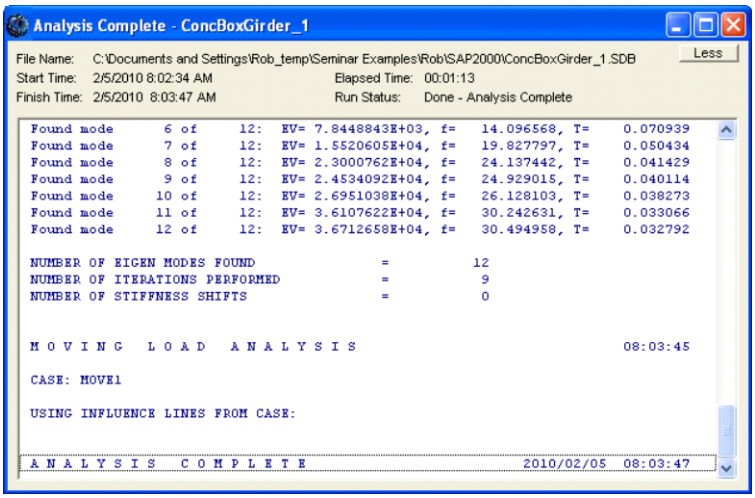
8.4.2 Analyze > Run Analysis – Screen Capture

This command (see Table 8-2) displays this form: **Analysis > Analyze > Run Analysis**



8.4.3 Analyze > Last Run Details

This command (see Table 8-2) displays this form: **Analysis > Analyze > Last Run**



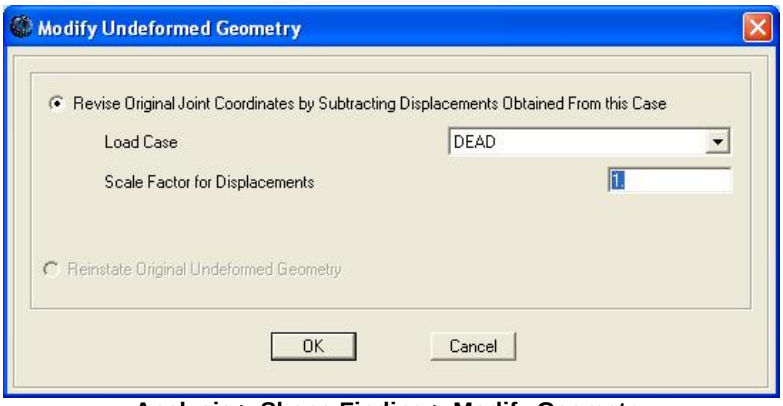
8.5 Analysis > Shape Finding

The *Shape Finding* panel on the **Analysis** tab has two commands that work in conjunction: Modify Geometry and Reset Geometry. Figure 8-7 shows the commands on the *Shape Finding* panel on the **Analysis** tab.

Figure 8-7 Commands on the Shape Finding panel of the Analysis tab

After at least one analysis has been run, the **Analysis** > **Shape Finding** > **Modify Geometry** command can be used to modify the undeformed geometry of the structure to achieve a desired deformed shape. The original undeformed geometry of the structure (geometry when the structure was first defined) is assumed to be the target for the deformed geometry of

the structure under a user-specified load case. Figure 8-8 shows the form used to apply the modification.



Analysis > Shape Finding > Modify Geometry

Figure 8-8 The form used to modify the undeformed geometry to achieve a target deformed geometry

A load case is selected and a scale factor for displacement is specified to modify the shape (inverse of the deflected shape). Use a scale factor greater than 1 if the convergence toward the target shape is too slow; use a scale factor less than 1 if the geometry is overshooting the target shape. Several iterations may be needed to achieve the target for the deformed shape. If after re-running the analysis the change in geometry significantly changes the load-carrying mechanism, convergence may be difficult. For example, a flexible flat slab may be changed to a domed shape that is much stiffer. Be aware that the deformed shape of some structures cannot be determined using this procedure, such as cables and membranes, because their shapes are determined by the loading.

The **Analysis** > **Shape Finding** > **Reset Geometry** command restores the original geometry.

CHAPTER 9 Design/Rating

The **Design/Rating** tab consists of the commands that allow efficient access to the forms needed to add, copy, or modify load combinations, including adding default combinations; set the preferences and create a superstructure design request as well as generate subsequent results; set the preferences and create a seismic design request as well as generate subsequent results; and set the preferences and generate a bridge load rating before obtaining results. The preferences include the codes and the necessary parameters for the designs. The available design codes are AASHTO LRFD 2012, AASHTO STD 2002, CAN/CSA-S6-06 and EUROCODE for the superstructure design, AASHTO Seismic 2011 for the bridge seismic design and AASHTO Rating 2011 for the bridge load rating.

Unlike the other tabs, starting a model using the Quick Bridge template has no effect on the tasks on the **Design/Rating** tab (i.e., no defaults associated with the **Design/Rating** tab are generated).

Unlike the other tabs, the ***Bridge Wizard*** can not be used to complete any of the tasks associated with the **Design/Rating** tab**.**

The commands on the **Design/Rating** tab can be used if the Blank option was used to start the model (i.e., the model is being built from scratch or by importing model data).

9.1 Design/Rating > Load Combinations

In CSiBridge, load combinations are defined by the user manually or added as defaults based on a selected code. Combinations based on a code subsequently can be modified to meet specific needs.

Combinations are used in the design process (described later in this chapter) and may be used when viewing analysis results (see context sensitive help for the **Home** > **Display** > **Show Deformed Shape** command). Thus, although combinations may be defined before or after an analysis has been run, to use them in reviewing results, the analysis must first be run, and to run a design (i.e., use the combos in design), an analysis must first be run.

Figure 9-1 shows the commands on the *Load Combinations* panel on the **Design/Rating** tab.

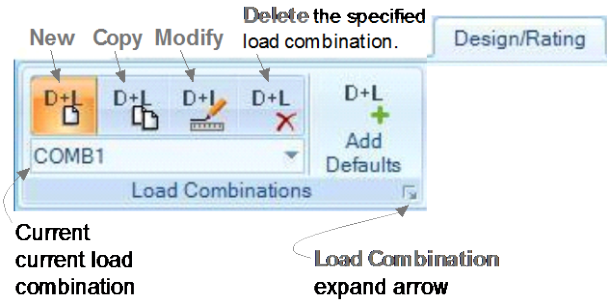
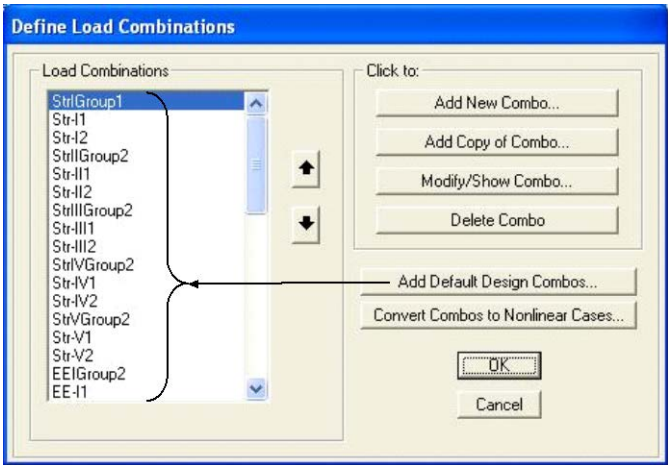


Figure 9-1 Design/Rating tab with annotated Load Combinations panel

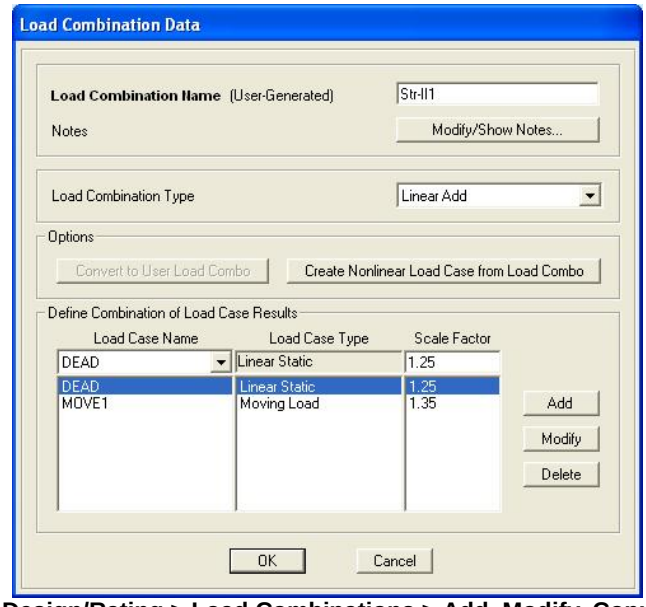
Clicking the expand arrow in the lower right-hand corner of the *Load Combinations* panel displays the Define Load Combinations form shown in Figure 9-2. All of the previously defined Load Combinations are listed on the left-hand side of the Define Load Combinations form. The buttons in the "Click to" area on the right-hand side of the form display the same form – the Load Combination Data form, shown in Figure 9-3 – as is displayed when the **New**, **Copy**, and **Modify** commands on the *Load Combinations* panel are used. Thus, the commands on the *Load Combi-*

nations panel bypass the definitions form (Figure 9-2), creating a shortcut to the data form used to define a load combination.



Design/Rating > Load Combinations > expand arrow

Figure 9-2 Definition form showing previously defined load combinations; in this figure, the default design combinations have been added



Design/Rating > Load Combinations > Add, Modify, Copy

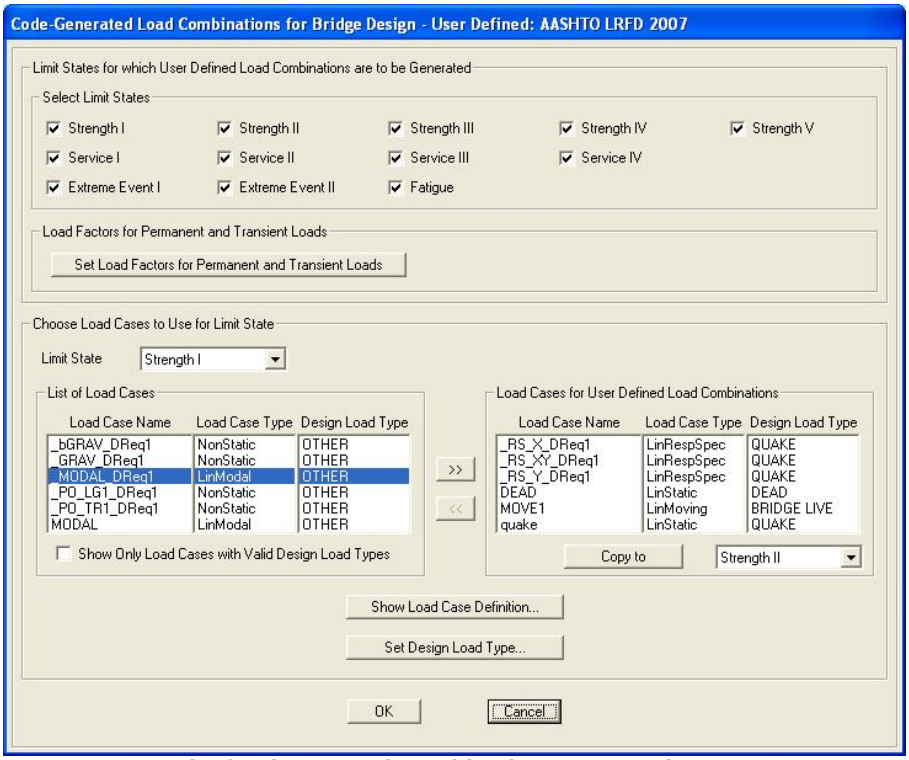
Figure 9-3 The data form used to define a load combination manually

The Load Combinations shown on the left-hand side of the form in Figure 9-2 are an example of default design combinations, which can be added by clicking the **Add Default Design Combos** button. That button duplicates the action of the **Design/Rating > Load Combinations > Add Defaults** command (see Figure 9-1). After using the button or the command, the Add Code-Generated User Load Combinations form shown in Figure 9-4 displays; that form can be used to select Bridge Design as the design type.

Design/Rating > Load Combinations > Expand arrow > Add Default Design Combos button

Figure 9-4 Form used to select the type of design (and apply the code) as the basis for generating the load combination

The **Set Load Combination Data** button on that form can be used to display another form (Figure 9-5) that has a range of options based on the selected code (see subsequent section on superstructure design and the **Design/Rating > Superstructure Design > Preferences** command). Thus, the load combination generated based on the code can be modified to meet specific needs, if necessary, using the form shown in Figure 9-5 (or one similar to it).



Design/Rating > Load Combinations > Expand arrow > Add Default Design Combos button **> Set Load Combination Data** button

Figure 9-5 An example of the fom used to review/define a load combination based on the selected code; the load cases shown were defined using the Add Defaults command

The **Convert Combos to Nonlinear Cases** button on the Define Load Combinations form (Figure 9-2) functions similar to the **Create Nonlinear Load Case for Load Combos** button on the Load Combination Data form (Figure 9-3). In both cases, linear static load cases can be converted to nonlinear load cases. This is a convenient way to create a nonlinear load while making use of an existing load combination when nonlinear effects, such as P-Delta, are required. The conversion is immediate and irreversible.

Table 9-1 briefly explains load combinations and the types of combos that are available.

Table 9-1 Load Combinations -- Design/Rating > Load Combinations

Types of Combos	Description
	A load combination is a named combination of the results from load cases (see *Chapter 8 Analysis*) or other load combinations. Combo results include all displacements and forces at the joints and internal forces or stresses in the elements. Each combo produces a pair of values for each response quantity: a maximum and a minimum. For some types of combos, both values are used. For other types, only the value with the larger absolute value is used. Each contributing load case is multiplied by a scale factor before being included in the combo. Five types of combos are available.
Linear Add.	All load case results are multiplied by their scale factor and added together. This *Combo Type* can be used for static loads.
Envelope	A max/min Envelope of the defined load cases is evaluated for each frame output segment and object joint. The load cases that give the maximum and minimum components are used for this combo. Therefore the load Combo holds two values for each output segment and joint. The *Combo Type* can be used for moving loads and any load case where the load producing the maximum or minimum force/stress is required.
Absolute Add	The absolutes of the individual load case results are summed and positive and negative values are automatically produced for each output segment and joint. Use this *Combo Type* for lateral loads.
SRSS	The Square Root Sum of the Squares calculation is performed on the load cases and positive and negative values are automatically produced for each output segment and joint. Use this *Combo Type* for lateral loads.
Range	The combined maximum is the sum of the positive maximum values from each of the contributing cases (a case with a negative maximum does not contribute), and the combined minimum is the sum of the negative minimum values from each of the contributing cases (a case with a positive minimum does not contribute). This Combo Type is useful for pattern or skip-type loading where all permutations of the contributing load case must be considered.

9.2 Design/Rating > Superstructure Design

Design using CSiBridge is based on load patterns (see *Chapter 6 Loads*), load cases (see *Chapter 8 Analysis*), load combinations (see previous section in this chapter), and design requests (described in this section). It should be noted that the design of a bridge superstructure is a complex subject and that the design codes cover many aspects of this process. CSiBridge is a tool to help the user with that process. Only the aspects of design documented in CSI's design manuals (see *Recommended Reading* in *Chapter 1 Introduction*) are automated by the CSiBridge. The user must check the results produced and address other aspects not covered by CSiBridge. Figure 9-6 shows the commands on the *Superstructure Design* panel on the **Design/Rating** tab.

Figure 9-6 Commands on the Superstructure Design panel on the Design/Ratings tab

These commands provide access to the forms needed to select the design code and other parameters; add, copy, modify, or delete a design request; select the design request to be run; and optimize design when the bridge is steel girder bridge.

Table 9-2 explains these commands briefly.

Table 9-2 Commands on the Design/Rating > Superstructure Design Panel

Command	Description
Preferences	Displays the Bridge Design Preferences form (see Section 9.2.1). Use this form to set the design code, and if available, other design parameters. Note that similar Preference commands are available on the *Seismic Design and Load Rating* panels on the **Design/Rating** tab.

Table 9-2 Commands on the Design/Rating > Superstructure Design Panel

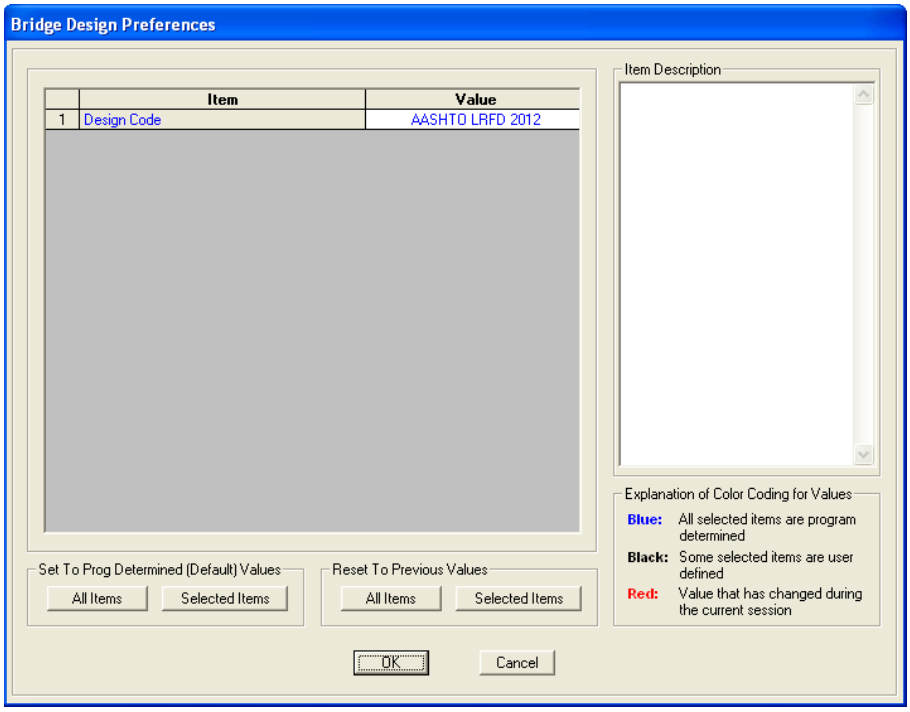
Command	Description
Design Request	Displays the Bridge Design Requests – {Code} form. That form is used to add, copy, modify, and delete design requests. The **Add New Request**, **Add Copy of Request**, and **Modify/Show Request** buttons provide access to the Bridge Design Request form (see Section 9.2.2). The design request definition requires a unique design request name, selection of the bridge object for which the design request is being defined; the check type (flexure, stress shear, and so on), station range (portion of the bridge to be designed), design parameters (e.g., stress factors), demand sets (loading combinations – see previous section), and where applicable live load distribution factors. > **Check Type** – Determines the check to be completed when the design request is run. The list of available check types reflects the deck types used in the specified Bridge Object; that is, the checks that are possible are relevant to the deck type used in the Bridge Object. > **Station Ranges** – Defines the start and end locations to be considered in the superstructure design. Use the station range to specify that the design is to be carried out over the entire length of the bridge or in just a particular zone. Multiple zones may be specified within a single design request. > **Demand Sets** – Identifies the load combination to be used in the design. Multiple demand sets may be defined for a single design request. The combinations selected for Bridge Design typically will be envelopes of other combinations, and the design will be performed for each combo within the selected enveloping combo. > **Live Load Distribution Factors (LLDF) to Girders** – Some check types include consideration of LLDFs. Users can chose how these factors are determined: user specified; in accordance with the code; directly from individual girder forces from CSiBridge; or uniformly distributed onto all girders. Note that the only time multiple lanes are necessary for a design is when the "directly from individual girder forces from CSiBridge" method is selected. Otherwise, moving live loads should be applied to only a single lane. The extent to which a vehicle load may be applied to a bridge deck is defined in the bridge deck definition (see **Components > Superstructure > Deck Sections**).
Run Super	Displays the Perform Bridge Design – Superstructure form (see Section 9.2.3). Use the form to select the design request to be run. Note that an analysis must be run before a superstructure design can be run. This is the case because each load case that is part of a load combination that is included in a design request must be

Table 9-2 Commands on the Design/Rating > Superstructure Design Panel

Command	Description
	run before starting superstructure design.
Optimize	Display the Bridge Object Superstructure Design and Optimization form (see section 9.2.4). After analysis and design of a steel girder bridge model has been completed, use the options on this form to interactively optimize design of the bridge model.

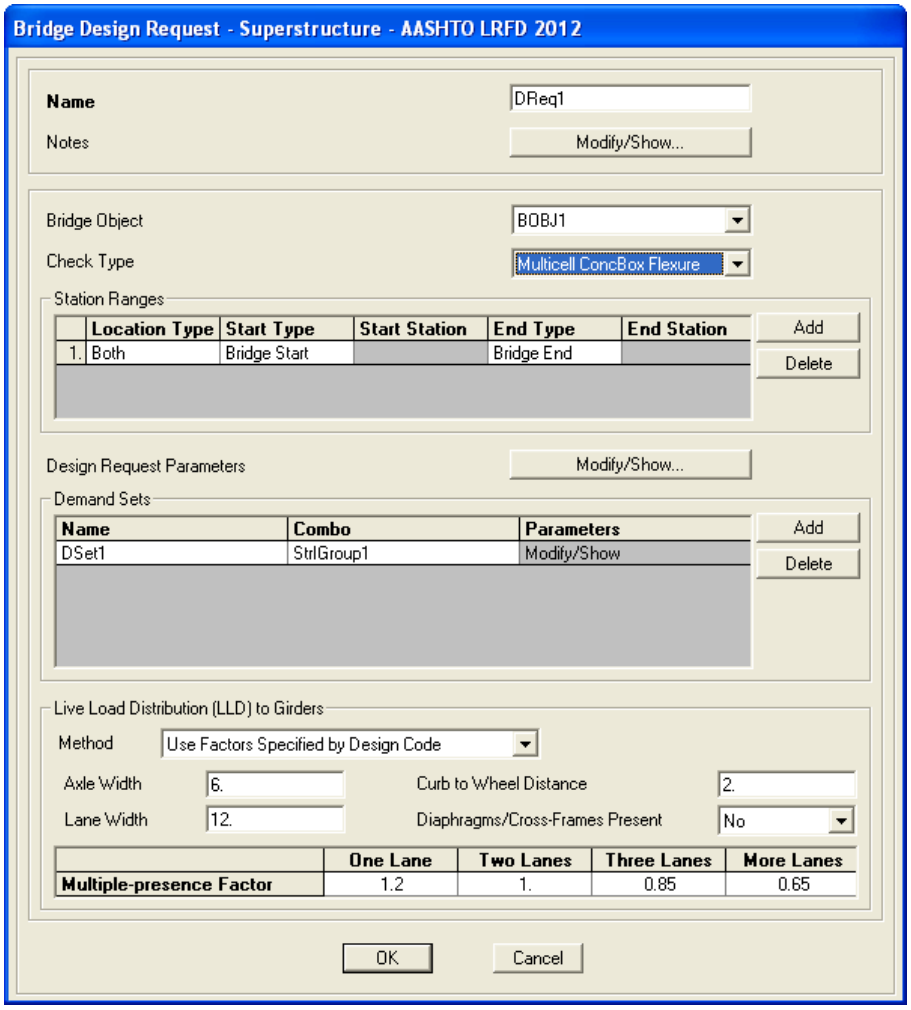
9.2.1 Superstructure Design > Preferences – Screen Capture

This command displays this form:
Design/Rating > Superstructure Design > Preferences



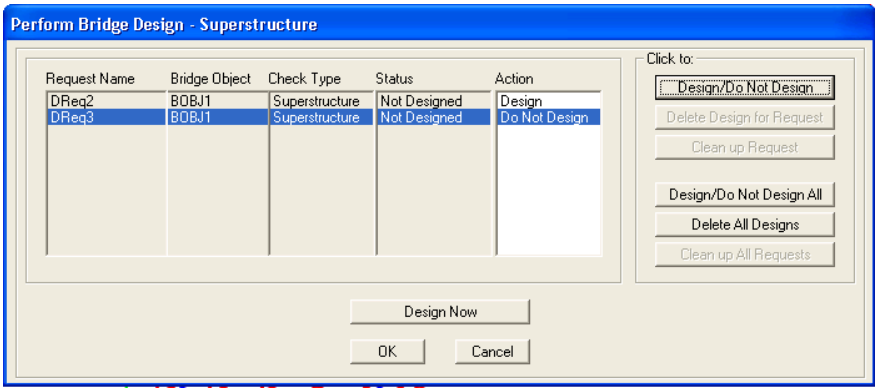
9.2.2 Superstructure Design > Design Requests – Screen Capture

This command displays this form:
Design/Rating > Design Requests > Add, Copy, Modify button



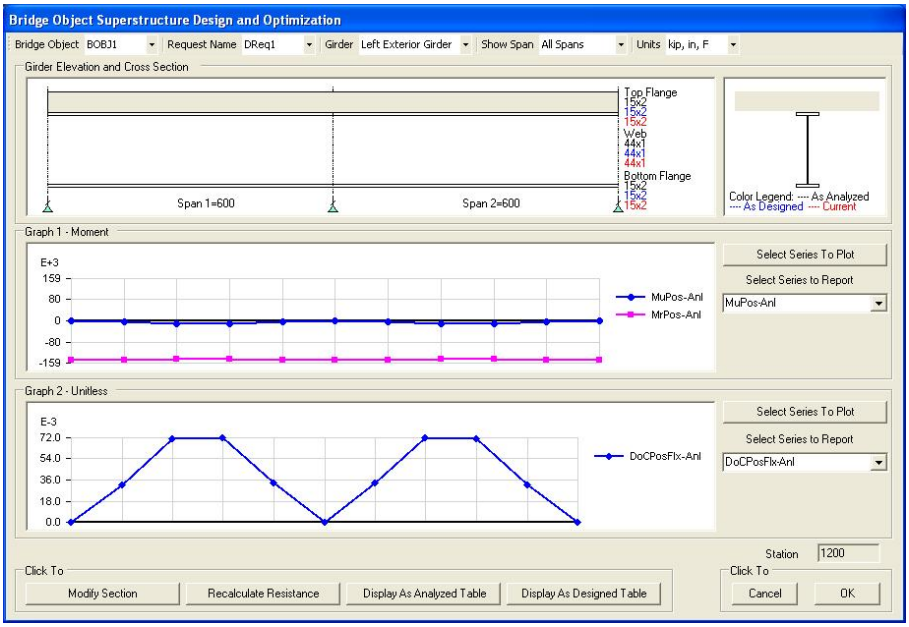
9.2.3 Superstructure Design > Run Super – Screen Capture

This command displays this form:
Design/Rating > Superstructure Design > Run Super



9.2.4 Superstructure Design > Optimize – Screen Capture

For a steel bridge after an analysis and design has been run, this command displays this form: **Design/Rating > Superstructure Design > Optimize**



9.3 Design/Rating > Seismic Design

CSiBridge allows engineers to define specific seismic design parameters to be applied to the bridge model during an automated cycle of analysis through design. By making appropriate definitions using **Components > Properties** (*Chapter 5*), **Loads > Functions – Response Spectrum** (*Chapter 6*), and **Analysis > Load Cases > Static > Nonlinear Static** (*Chapter 8*), engineers may use CSiBridge to automate the determination of cracked section properties, response spectrum demand analyses as well as the nonlinear static pushover analyses, or use program defaults. Furthermore, CSiBridge will determine the displacement demand/ capacity ratios for the Earthquake Resisting System (ERS).

Figure 9-7 shows the *Seismic Design* panel on the **Design/Rating** tab. These commands provide access to the forms needed to select the design code and other parameters; add, copy, modify, or delete a design request, including specify design parameters, such as the response spectrum function and pushover target displacement ratio among several other parameters; select the design request to be run; and generate a summary input and seismic design output report. Table 9-3 explains these commands briefly.

Figure 9-7 The Seismic Design panel on the Design/Rating tab

Table 9-3 Commands on the Design/Rating > Seismic Design Panel

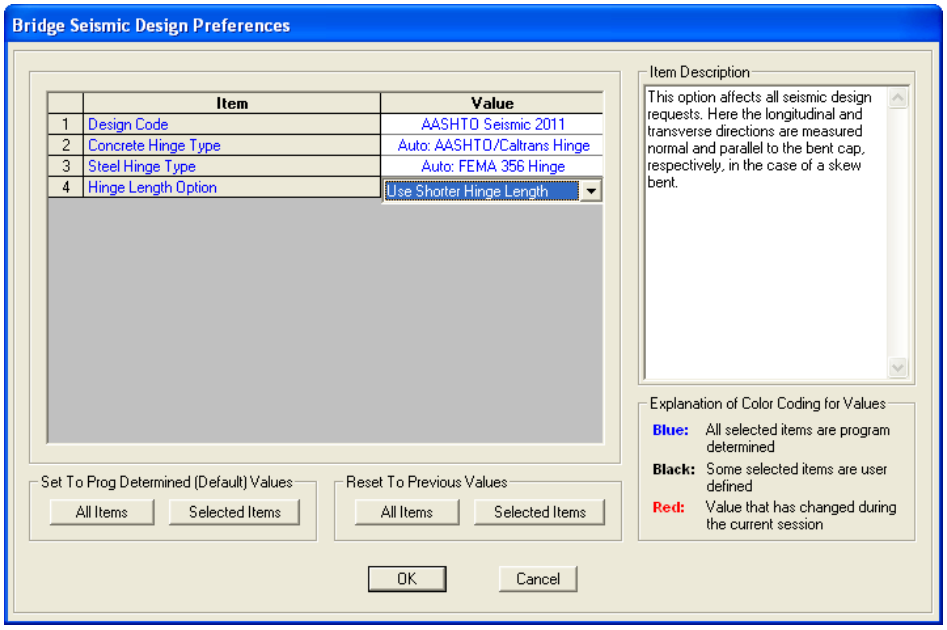
Command	Description
Preferences	Displays the Bridge Design Preferences form (see Section 9.3.1). Use that form to set the design code, and if available, other design parameters. Note that similar Preference commands are available on the *Superstructure Design and Load Rating* panels on the Design/Rating tab.
Design Request	Displays the Bridge Seismic Design Requests – {Code} form. That form is used to add, copy, modify, and delete design requests. The **Add New Request**, **Add Copy of Request**, and **Modify/Show**

Table 9-3 Commands on the Design/Rating > Seismic Design Panel

Command	Description
	Request buttons provide access to the Bridge Design Request – Substructure Seismic – {Code} form (see Section 9.3.2). That form is used to specify a unique name for the design request and to select the Bridge Object to which the design applies; clicking the Design Request Parameters **Modify/Show** button displays the Substructure Seismic Design Request Parameters form (see Section 9.3.2). That form can be used to select the response spectrum function to be used in a particular design as well as the seismic design category and other seismic design parameters. If foundations have been included in a bridge model, Support Structure should be selected as a Bridge Group Type on the Bridge Construction Group Definition form so that the Bridge Object creates a group(s) of the support structures for use in this seismic design (**Bridge > Bridge Object > Groups** command); then, in this form, the Bridge Object that includes the support structure group should be selected for the Foundation Group item (shown as number 22 in the figure in Section 9.3.2)
Run Seismic	Displays the Perform Bridge Design – Seismic form (see Section 9.3.3). Use that form to specify the seismic design to be run.
Report	A summary input and seismic design output report will be generated. The report will contain a table of contents and the images of all of the pushover plots as well as the output data files contained in the table seismic design summaries.

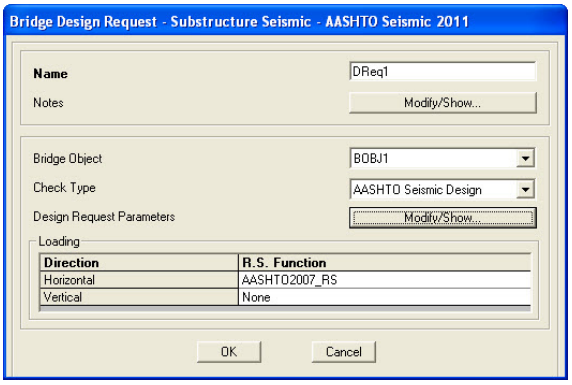
9.3.1 Seismic Design > Preferences – Screen Capture

Use this command to display this form:
Design/Rating > Seismic Design > Preferences

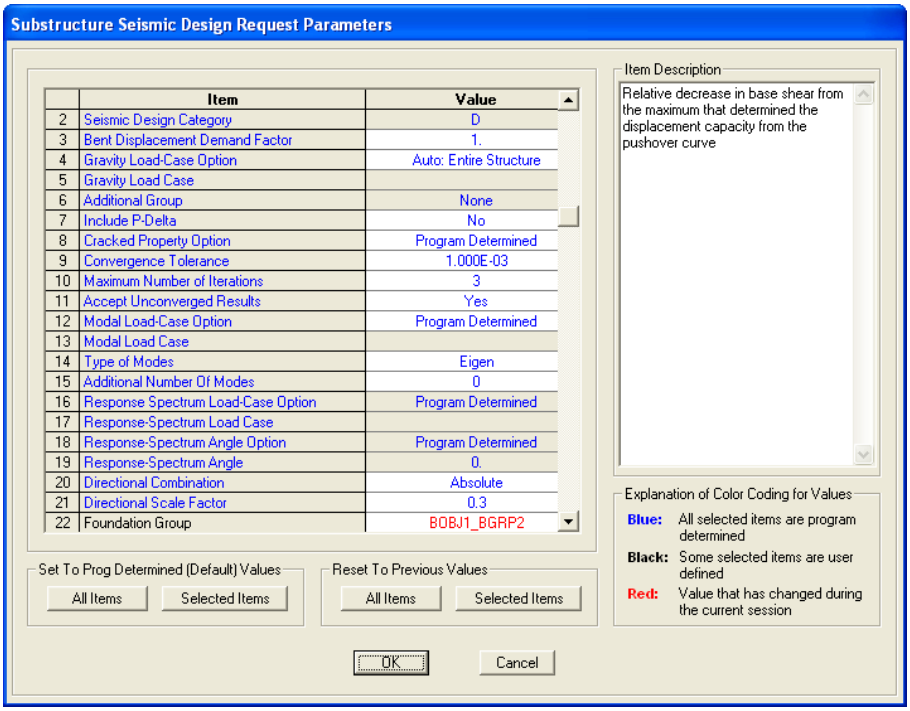


9.3.2 Seismic Design > Design Requests – Screen Capture

This command displays this form:
Design/Rating > Seismic Design > Design Requests



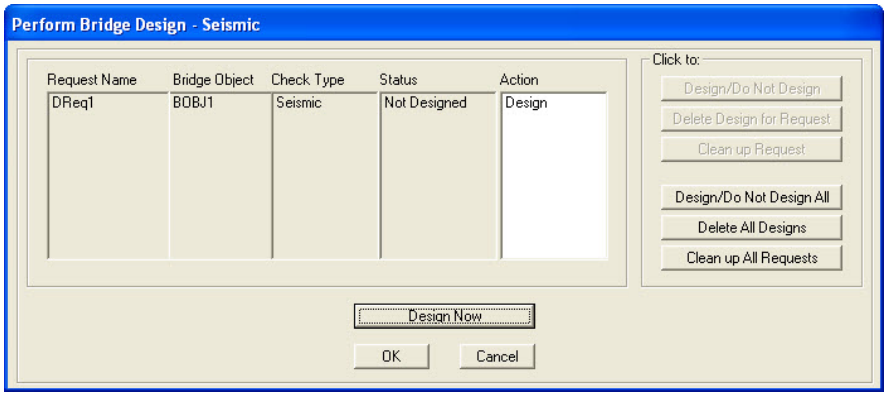
This command displays this form:
Design/Rating > Seismic Design > Design Requests



.

9.3.3 Seismic Design > Run Seismic – Screen Capture

This command displays this form:
Design/Rating > Seismic Design > Run Seismic



9.4 Design/Rating > Load Rating

CSiBridge may be used to determine the Load Rating of a concrete bridge. Three different bridge deck types may be rated. For a complete description of the load rating algorithms and methodology, display the *Bridge Rating Manual* using the **Help** > **Documentation** > **Bridge** command. A brief description of the deck types and applicable codes and equations follows.

Figure 9-8 shows the *Load Rating* panel on the **Design/Rating** tab.

Figure 9-8 The Load Rating panel on the Design/Rating tab

These commands provide access to the forms needed to select the rating code and other parameters; add, copy, modify, or delete a rating request, including specify rating parameters, such as the response spectrum func-

tion and pushover target displacement ratio among several other parameters; and select the rating request to be run. Table 9-4 explains these commands briefly.

Table 9-4 Commands on the Design Rating > Load Rating Panel

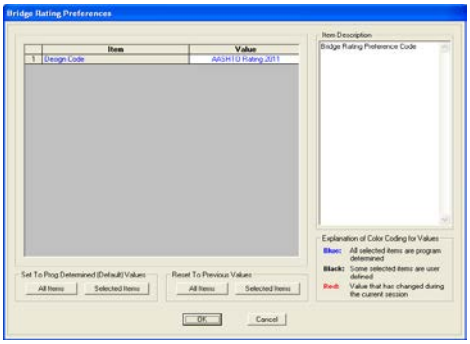
Command	Description
Preferences	Displays the Bridge Design Preferences form (see Section 9.4.1). Use this form to set the design code, and if available, other design parameters. Note that similar Preference commands are available on the *Superstructure Design and Seismic Design* panels on the Design/Rating tab.
Rating Requests	Displays the Bridge Rating Requests – {Code} form. That form is used to add, copy, modify, and delete rating requests. The **Add New Request**, **Add Copy of Request**, and **Modify/Show Request** buttons provide access to the Bridge Rating Request – Superstructure – {Code} form (see Section 9.4.2). The rating request definition requires a unique rating request name, selection of the Bridge Object to which the rating request applies; the rating type (flexure, minimum rebar, shear), station range (portion of the bridge to be rated), design rating parameters (e.g., flexure), and demand sets (loading combinations – see Section 9.1). > **Rating Type** – Determines the rating check to be completed when the rating request is run. The list of available rating types reflects the deck types used in the specified Bridge Object; that is, the rating that are possible are relevant to the deck type used in the Bridge Object. > **Station Ranges** – The Station Ranges specify where the rating applies. Use the start and end options to specify that the rating be performed for the entire length of the bridge or for just a particular zone. Multiple zones may be specified within a single rating request. > **Design Rating Parameters > Modify/Show** button. Displays the Superstructure Design Request Parameters form. Use the form to specify various parameters (e.g., phi factors for flexure, rupture and moment for minimum rebar, phi factors for shear along with positive and negative strain limits), depending on the rating type selected. > **Demand Sets** – Identifies the load combination to be used in the rating. Multiple demand sets may be defined for a single rating request. The rating factor is a ratio of the superstructure reserve capacity moment to the bridge live load demand moment.

Table 9-4 Commands on the Design Rating > Load Rating Panel

Command	Description
	> **Live Load Distribution Factors (LLDF) to Girders** – Some rating types include consideration of LLDFs. Users can chose how these factors are determined: user specified; in accordance with the code; directly from individual girder forces from CSiBridge; or uniformly distributed onto all girders. Note that the only time multiple lanes are necessary for a design is when the "directly from individual girder forces from CSiBridge" method is selected. Otherwise, moving live loads should be applied to only a single lane. The extent to which a vehicle load may be applied to a bridge deck is defined in the bridge deck definition (see **Components > Superstructure > Deck Sections**).
Run Rating	Displays the Perform Bridge Superstructure Rating form (see Section 9.4.3). Use the form to select the rating request to be run. Note that an analysis must be run before a rating request can be run. This is the case because each load case that is part of a load combination that is included in a rating request must be run before starting superstructure rating. After the rating has been run, the Bridge Object Response Display form is called up (see Section 9.4.3). The form shows the bridge response plot for the forces, stresses, and design/rating.
Optimize	Displays the Bridge Object Superstructure Design and Optimization form (see Section 9.4.4). After analysis and design of a steel girder bridge model has been completed, use the options on this form to interactively optimize rating of the bridge model.

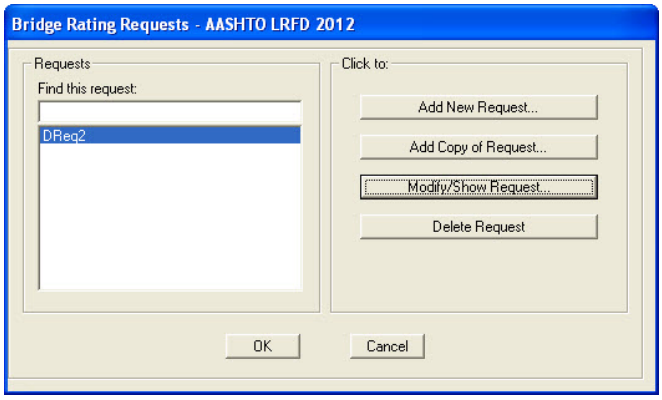
9.4.1 Load Rating > Preferences – Screen Capture

This command displays this form:
Design/Rating > Load Rating > Preferences

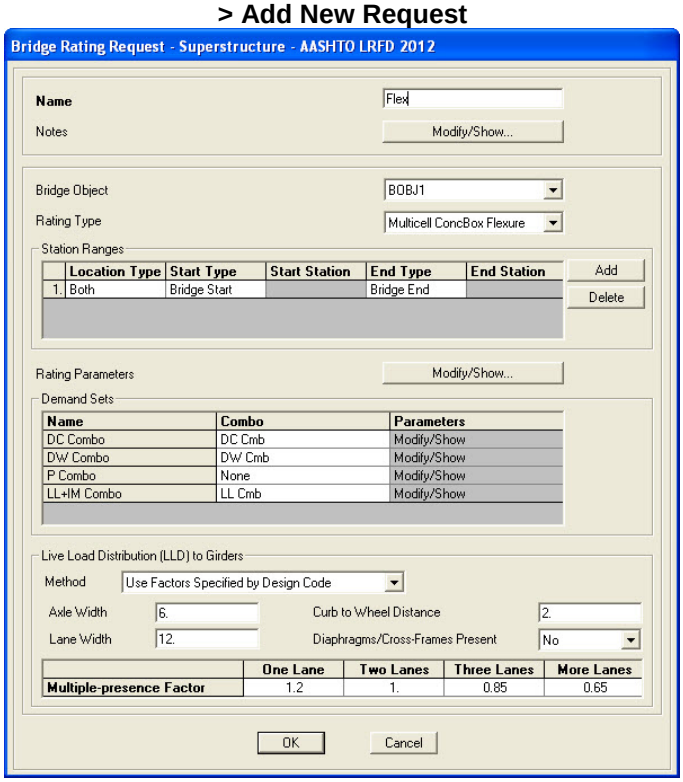


9.4.2 Load Rating > Rating Requests – Screen Capture

This command displays this form:
Design/Rating > Load Rating > Rating Requests

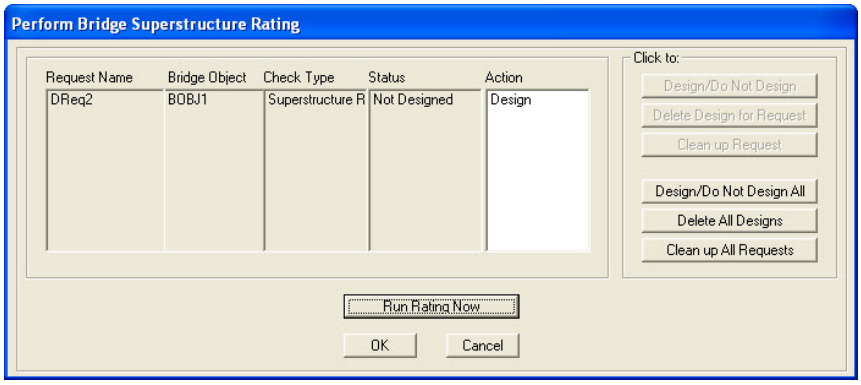


> Add New Request

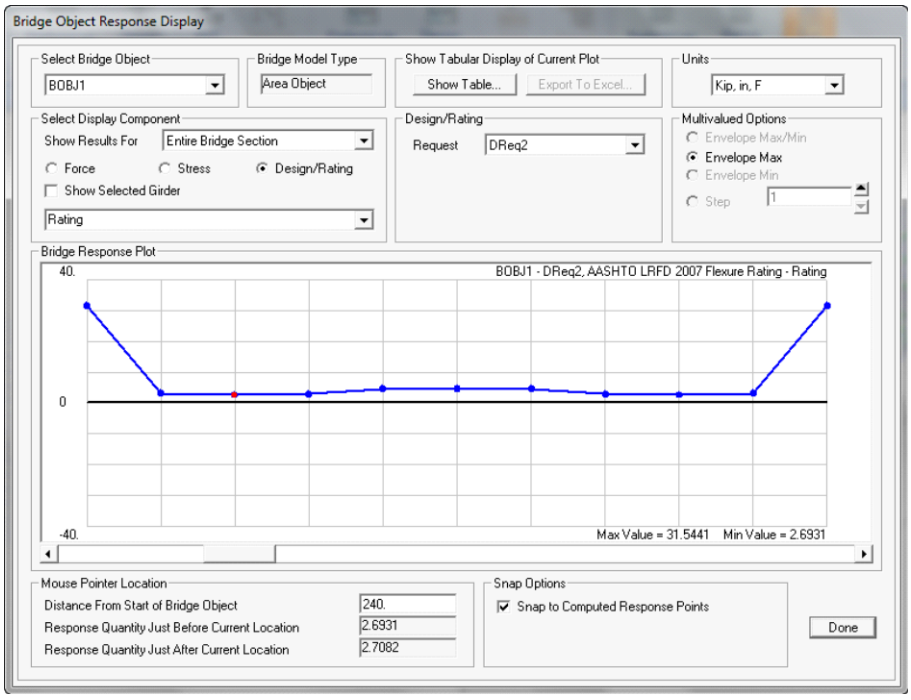


9.4.3 Load Rating > Run Rating – Screen Capture

These commands display these forms:
Design/Rating > Load Rating > Run Rating



> Design Now

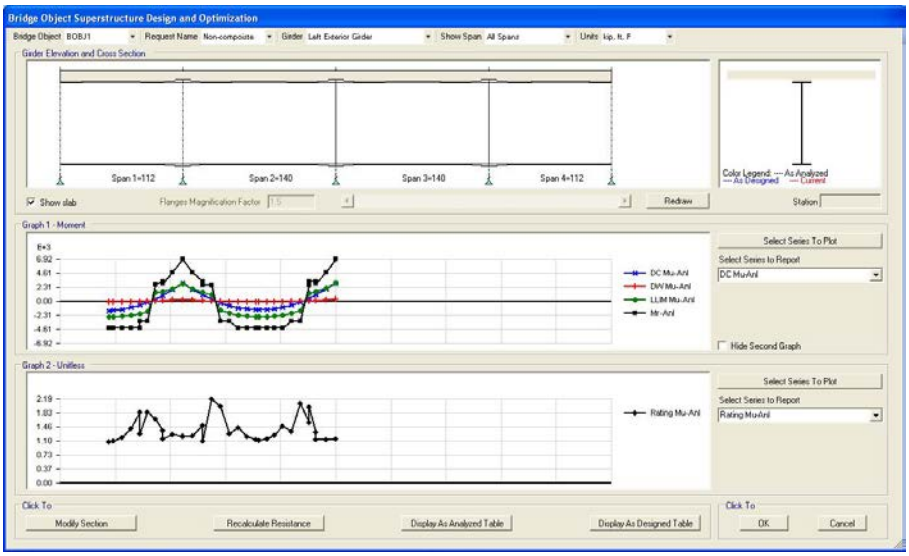


Plot of the Load Rating Response – Note that the rating factor of 2.7082 at station point 240 denotes that the bridge is adequate to support

a vehicle weighing 2.7082 times greater than the vehicle used to load the bridge.

9.4.4 Load Rating > Optimize – Screen Capture

For a steel bridge after an analysis and design has been run, this command displays this form: **Design/Rating > Load Rating > Optimize**



CHAPTER 10 Advanced

The **Advanced** tab consists of the commands that can be used to edit selected objects; add definitions; draw objects (e.g., areas, special joints, frames, cables and tendons); assign definitions to joints, frames, areas, cables, tendons, solids, links/supports; assign loads to selected joints, frames, areas, cables, tendons, solids, link/supports; complete steel and concrete frame design, overwrite frame design procedures, specify lateral bracing; and add plug ins. The majority of these commands must be used when the model is unlocked. These commands can be used irrespective of how the bridge model was initiated: the using Quick Bridge template, the ***Bridge Wizard***, or the Blank option (i.e., the model is being built "from scratch" or by importing model data). A bridge model can be created, analyzed, and designed in CSiBridge without using any of the commands on this tab.

The **Advanced** tab has eight panels: *Edit*, *Define*, *Assign*, *Assign Loads*, *Analyze*, *Frame Design*, and *Tools*. Tables 10-1 through 10-7 briefly describe the commands on the various panels.

10.1 Advanced > Edit

Table 10-1 describes the commands on the *Edit* panel of the **Advanced** tab.

Table 10-1 Advanced > Edit

Command	Description
> Points	> **Add Grid at Selected Points** – Use to add grid lines in the current coordinate/grid system.
	> **Merge Joints** – Use to automatically merge selected joints with the default auto merge tolerance of one another.
	> **Align Points** – Use to align selected points with the specified X, Y, or Z coordinates or to the nearest line.
	> **Disconnect** – All objects connected to each other normally share a common joint. Use this command to break off the objects from the joint and will add duplicate joints to each of those objects.
	> **Connect** – Use in conjunction with the **Disconnect** command to connect all selected objects that have been disconnected from each other or that were connected to independent joints. When objects are reconnected, the independent joints are collapsed into a common joint that is shared by all the combined objects.
> Lines	> **Divide Frames** – Use to divide a selected frame into user-specified segments of user-defined size, or break a frame at intersections with selected frames and joints.
	> **Join Frames** – Use to immediately join the selected frame objects into a single object and remove unused joints remaining after the joining process.
	> **Trim/Extend Frames** – Use to lengthen frame elements that are too short, or shorten frame element that are too long.
	> **Edit Curved Frame Geometry** – Use to edit selected curved frame objects. Only one object can be edited at a time.
	> **Edit Cable Geometry** – Use to edit selected cable objects. Only one object can be edited at a time.
	> **Edit Tendon Profile** – Use to edit a line object drawn as a tendon.
> Areas	> **Divide Areas** – CSiBridge allows area objects to be divided into smaller objects in the object-based model, and meshed into elements in the analysis model. Dividing the object is accomplished using this command. Meshing an area object is accomplished using the **Advanced > Assign > Areas > Automatic Area Mesh** command.

Table 10-1 Advanced > Edit

Command	Description
> Areas (continued)	> **Merge Areas** – Merge selected area objects that are in the same plane and share a common edge or with overlapping area edges.
	> **Expand/Shrink Areas** – Use to specify offsets for area edges to make areas larger or smaller.
	> **Add Point to Area Edge** – Add collinear points to the edges of area objects.
	> **Remove Points from Area** – Remove selected points from the edges of area objects.
> More	> **Undo/Redo** – Use the **Undo** command to reverse an action performed during the modeling process back to the last time the model was saved. If the command is used for too many steps, the steps will begin to be "redone." Use the **Redo** command to restore an action performed during the modeling process.
	> **Cut, Copy, Paste** – Work similar to standard, cut, copy, and paste Windows commands. The entire structure or any selected part of it may be cut or copied and then pasted back into the model at any user-specified location.
	> **Delete** – Deletes selected objects.
	> **Add to Model From Template** – Add objects to a model using a template.
	> **Interactive Database Editing** – Use to create or edit model definition data in a tabular format rather than in graphical mode or using the Draw or Assign menu forms. Only model definition data (not analysis results or design data) can be edited in this way. The model must be unlocked to use this feature.
	> **Replicate** – Replicate is a very powerful way of generating a large model from a small model when the objects and/or joints form a linear or radial pattern or are symmetrical about a plane. When joints or objects are replicated, the assignments on those joints and objects are also replicated (for example, member section assignments, member loads, joint loads and joint restraints). This is a major benefit of using **Replicate** rather than the **Cut**, **Copy** and **Paste** commands, which can be used to cut, copy, or paste lines, areas and joints but not their assignments or loads.

Table 10-1 Advanced > Edit

Command	Description
> More (continued)	> **Extrude** – Use to sweep selected objects through space to create new objects of higher dimension or to convert lines to areas and areas to solids with the same thickness as the original line or area object. The process of extrusion increases the dimensional space of an existing object by one. In other words, line objects are of one dimension that can be generated from a dimensionless object, the point object. In a similar manner, a two-dimensional object, area or plate/shell can be generated from a one-dimensional object, the line object. This feature is especially suited to creating solid objects from plate/shells, plate/shell objects from beams and beams/columns from points/nodes.
	> **Move** – Use to move selected parts of a structure to a new location in the model.
	> **Divide Solids** – Use to divide selected solid objects on all faces.
	> **Show Duplicates** – Use to verify that duplicates are included in the model and, if so, where they are located. Duplicate objects can be defined as objects of the same type (joints, frames, shells, asolids, solids and so forth) that have the same coordinates in the model. If duplicate objects are unnecessary, this command is useful in locating them so that they can be deleted or merged. Duplicates can not be drawn or created; they can result when data is imported from other programs, such as AutoCAD, CIS/2 Step and so on.
	> **Merge Duplicates** – Use in conjunction with the **Show Duplicates** command to combine duplicate objects into one object.
	> **Change Labels** – By default, the names (labels) of the objects and all defined entities (loads, properties, and so on) in the model are sequentially designated by the program using alphanumeric characters. Use this command to access the Interactive Name Change form and change the name of any of the named items.

10.2 Advanced > Define

Table 10-2 describes the command on the *Define* panel of the **Advanced** tab.

Table 10-2 Advanced > Define

Command	Description
> Section Properties	Use the subcommands to add area section, solid property, frequency dependent property, and hinge property definitions.
> Mass Source	Specify how CSiBridge calculates mass for the model. In CSiBridge, mass and weight serve different functions. Mass is used for the inertia in dynamic analyses, and for calculating the built-in acceleration loads. Weight is a load that users define and assign in one or more loads (see Assign - Area Loads, Frame Loads, Cable Loads, Tendon Loads, Joint Loads, Link Loads, and Solid Loads), which can then be applied in one or more load cases.
> Coordinate Systems/ Grids	Coordinate systems are defined by specifying their origins and orientations with respect to the fixed Global coordinate system. They are used to define the location and orientation of the associated grid system. Thus, coordinate systems are always defined as part of a coordinate/grid system pair. However, when only a coordinate system is needed, such as when viewing the model, defining the model (object local axes, loads, and the like), and reporting results, a coordinate/grid system can be defined with no grid lines. A grid system is a set of intersecting lines used to aid in drawing the model. Grid systems may be Cartesian (rectangular), cylindrical, or general. Each grid system is defined with respect to its own coordinate system, so, as indicated previously, grid and coordinate systems are defined together as a single entity. An unlimited number of coordinate/grid systems can be defined, each with a unique name. A regular system is any coordinate/grid system having a Cartesian (rectangular) or cylindrical grid system. A general system is a system comprised of arbitrarily defined grid lines.
> Joint Constraints	Add, modify, or delete constraint definitions. Rigid-type constraints are those when all joints in the constraint move together as some type of rigid body. Rotational and translational degrees of freedom may be coupled together**.** Equal-type constraints are those when individual degrees of freedom of different joints are identical. These are usually used for connection and for symmetry conditions. Interpolation constraints are those when degrees of freedom at one joint are interpolated from the degrees of freedom at some other joints. These are usually used to connect incompatible meshes.

Table 10-2 Advanced > Define

Command	Description
> Joint Patterns	Use to specify the name of a joint pattern, which is a set of scalar values defined at the joints. The specified named joint pattern can be referenced when assigning temperature or pressure loads to objects. This feature allows description of more complex distributions of temperature and pressure over the structure. Joint patterns by themselves create no loads on the structure.
> Groups	The concept of groups is the backbone of some powerful tools in CSiBridge. A group is a collection of objects that is assigned a unique name. Groups may be used for many different purposes. Each object may be part of as many groups as needed. All objects are part of the built-in group named "ALL."
> Section Cuts	Use to obtain resultant forces acting at section cuts through a model. Section cuts can be defined before or after an analysis has been run; however, it is safest to wait until after the analysis has been run. Typically do not define section cuts, and more importantly, the groups used in the section cut definition, until all manual meshing of the model (if any) has been completed (see Frame - Automatic Frame Mesh, Area - Automatic Area Mesh or Solid - Automatic Solid Mesh for more information about meshing). If the groups are defined before manual meshing, some of the point objects that should be in the group may not yet be created
> Generalized Displacements	A generalized displacement is a named displacement measure that is user defined. It is simply a linear combination of displacement degrees of freedom from one or more joints. For example, a defined generalized displacement named "DRIFTX" could be the difference of the UX displacements at two joints on different stories of a building. Another defined generalized displacement named AVGRZ could be the sum of three rotations about the Z axis, each scaled by 1/3. Generalized displacements are primarily used for output purposes, except that a generalized displacement also can be used to monitor a displacement-controlled nonlinear static analysis.
> Functions	> **Steady State** – Use to add, modify, or delete a steady-state function. A steady-state function is a list of function values versus frequency of excitation. The function values in a steady-state function may be normalized ground acceleration values or they may be multipliers for specified (force or displacement) load patterns.

Table 10-2 Advanced > Define

Command	Description
> Functions (continued)	> **Power Spectral Density** – Use to add, modify, or delete power spectral density functions. A power spectral density function is a list of function values versus frequency of excitation. The function values in a power-spectral-density function may refer to normalized ground acceleration values or to multipliers for specified (force or displacement) load patterns. This is a probabilistic type of function. Function values are specified as the **square** of the input value **per unit of frequency**. In other words, this function should be specified so that the square-root of the integral of the function over the frequency range gives the expected RMS (root-mean-square) magnitude of loading. Function values may not be negative. The program defines a default unit constant power spectral density function.
> Named Property Sets	> **Frame Modifiers, Area Modifiers** – A named set of property modifiers can be applied to a frame or area object during staged construction to change the property modifiers that were previously assigned or applied to the object. The effect of property modifiers in a named set is exactly the same as the effect of those modifiers that can be assigned directly to frame and area objects using the **Advanced > Assign > Frame > Property Modifiers** command or the **Advanced > Assign > Area > Area Stiffness Modifiers** command. By default, all objects use the property modifiers that are assigned to them unless explicitly changed by applying a named set during staged construction, in which case they replace the previously assigned or applied property modifiers for all subsequent loading in the analysis. Named sets of property modifiers affect only the element values of the modifiers, not those that are defined as part of the frame/area section properties.
	> **Frame Releases** – A named set of frame releases can be applied to a frame object during staged construction to change the releases that were previously assigned or applied to the object. The effect of frame releases in a named set is exactly the same as the effect of those releases that can be assigned directly to selected frame objects using the **Advanced > Assign > Frame > Releases/Partial Fixity** command. By default, all frame objects use the releases that are assigned to them unless explicitly changed by applying a named set during staged construction, in which case they replace the previously assigned or applied releases for all subsequent loading in the analysis.

Table 10-2 Advanced > Define

Command	Description
> Pushover Parameters Sets	Specify the parameters for displaying pushover curves. The parameters are saved as a named definition so they can be recalled and applied quickly.
> Named Sets	Save the options selected on the various forms used to generate output (tables, curves, and the like) as a definition known as a Named Set. The options can then be recalled and applied quickly.

10.3 Advanced > Draw

Table 10-3 describes the command on the *Draw* panel of the **Advanced** tab.

Table 10-3 Advanced > Draw

Command	Description
> Set Select Mode	CSiBridge has two modes of mouse-cursor operation in a display window: Draw mode and Select mode. The program can be in one mode at a time only. After a new model has been started, by default, the program is in Select mode. The mouse cursor is used to select objects before performing certain editing, assignment, display, and output operations. When the model is locked, the program is always in Select mode.
> Set Reshape Object Mode	In reshape mode, click on an area, line or point object and modify it in one of several ways. If you have difficulties using any of the methods, try using the method in a different view. A Properties of Object form will display with a number of Drawing Control options. The controls are assumed to be self-explanatory.
> Draw One Joint Link	Add a joint link when the model is displayed in a 2-D view.
> Draw Two Joint Link	Draw a two joint link when the model is displayed in a2-D view.
> Draw Frame/ Cable/Tendon	Draw a straight or curved frame object, a cable, or a tendon.

Table 10-3 Advanced > Draw

Command	Description
> Quick Draw Frame/Cable/ Tendon	Draw a straight or curved frame object, a cable, or a tendon using grid lines.
> Quick Draw Braces	Click in a grid space, bounded by four grid lines, to draw a quick brace.
> Quick Draw Secondary Beams	Click in a grid space bounded by four grid lines to draw quick secondary beams.
> Draw Poly Area	Click on a grid intersection, a previously define joint, or any point in a plane in clockwise or counterclockwise direction, collinear or not, to draw the poly area.
> Draw Rectangular Area	Click on a grid intersection, a previously define joint, or any point in a plane to draw one corner of the rectangular area; click again on the opposite corner of the shape.
> Quick Draw Areas	Click in a grid space, bounded by four grid lines, to draw a quick single area object.
> Draw Special Joint	In creating a CSiBridge model, it is not necessary to predefine joints. The joints are automatically added to the ends or corners of objects. Special Joints are those joints added by the user. Adding joints may be necessary in rare cases, such as at one end of an NLLink Element - an end where there is no other object present and hence no automatically generated joint. Click on a grid intersection or any other point in the plane to add a joint.
> More	> **Draw Solid** – Use to draw a solid object; works only in a 3D view that already contains a structural model that has three dimensions. Click at the first corner point of the solid. Move the mouse to a second and third or fourth point to complete the outline of the area of the solid. Then drag the mouse down or up to specify the depth of the solid. > **Quick Draw Link** – Draw a link using a grid line.

Table 10-3 Advanced > Draw

Command	Description
> More (continued)	> **Draw Section Cut** – Use to display integrated forces along a specified section cut. With a deformed shape showing in the active window, move the mouse pointer/cursor to the starting point of the section cut to be drawn on the deformed shape and click the left mouse button. Drag the mouse to the ending point for the section cut.
	> **Draw Developed Elevation Definition** – Define a developed elevation graphically. Name the elevation and then click the left mouse button at a point representing the beginning point for the developed elevation. Continue selecting points to define the location of the developed elevation.
	> **Draw Reference Point** – Draw individual reference points in a plan view for facilitating the placement of other objects
	> **Draw/Edit General Reference Line** – Add, change or delete a general reference line. Objects can be extruded relative to a general reference line, and bridge layout lines can be specified using a general reference line as long as the reference line complies with the restrictions applicable to layout lines with respect to vertical and horizontal alignment.
	> **New Labels** – The labels in CSiBridge are alphanumeric. By default the program automatically assigns a numeric numbering scheme to the joints and objects. However, it is possible to assign an alphanumeric labeling scheme by giving an alpha prefix and a starting numeric sequence. All objects added after the scheme is initialized will be affected by the scheme.

10.4 Advanced > Assign

Table 10-4 describes the command on the *Assign* panel of the **Advanced** tab.

Table 10-4 Advanced > Assign

Command	Description
> Joints	> **Restraints** – A joint restraint is the same as a joint support. It is a rigid connection of the joint to the ground. Restraints are specified independently for each degree of freedom at a joint. Restraints are always applied in the joint local coordinate system. By default, this is the same as the Global coordinate system, i.e., joint local axes 1, 2, and 3 are the same as the global axes X, Y, and Z, respectively. If necessary, use the **Advanced > Assign > Joint > Local Axes** command to specify the local axes for selected joints. To impose a specified displacement at a joint degree of freedom, first restrain that degree of freedom using this command. This is because displacement loads are actually specified for the ground, and the restraint will force the joint to move with the ground.
	> **Constraints** – A constraint is a group of joints that are connected. When a constraint is assigned to a joint, the joint becomes a part of the constraint and connects to other joints in the constraint.
	> **Springs** – Springs are flexible connections to ground and are always linear elastic. Assigning a spring to a joint is only meaningful if structural objects are connected to the joint. Otherwise, the spring will support air, so to speak; that is, it will not support anything. Use this command to assign to selected joints springs that are oriented in the global axes directions. Both translational and rotational springs can be assigned to a joint
	> **Panel Zones** – A panel zone assignment to a point object allows differential rotation and in some cases differential translation at beams-to-other-objects, braces-to-other-objects, or from column connections. Multiple panel zones can not be assigned to the same point object.
	> **Masses** – Every object contributes mass to the structure from the mass density of its material. Use this command to assign additional joint mass to a joint. Note that the additional joint mass is considered by CSiBridge only if the mass source has been specified to be based on element masses and additional masses, not from a specified load combination.

Table 10-4 Advanced > Assign

Command	Description
> Joints (continued)	> **Local Axes** – By default, the joint local 1-2-3 coordinate system is identical to the global X-Y-Z coordinate system. However, it may be necessary to use different local coordinate systems at some or all joints in the following cases: skewed restraints (supports) are present; constraints are used to impose rotational symmetry; constraints are used to impose symmetry about a plane that is not parallel to a global coordinate plane; the principal axes for the joint mass (translational or rotational) are not aligned with the global axes; and joint displacement and force output is desired in another coordinate system. Joint local coordinate systems need only be defined for the affected joints. The global system is used for all joints for which no local coordinate system is explicitly specified. Advanced methods are available to define a joint local coordinate system. These may be used separately or together. Local coordinate axes may be defined to be parallel to arbitrary coordinate directions in an arbitrary coordinate system or to vectors between pairs of joints. In addition, the joint local coordinate system may be specified by a set of three joint coordinate angles. Use this command to assign the same local axes to one or more selected joints.
	> **Merge Number** – Assign merge numbers to selected points. When the analysis model is created, points that occur in the same location and have the same merge number will be merged into a single point. Assigning different merge numbers to points in the same location that have been disconnected in the object based model assures that those points will remain disconnected in the analysis model.
>	**Joint Patterns** – Use to assign a previously defined joint pattern to a selected joint. A joint pattern is a set of scalar values defined at the joints. The specified named joint pattern can be referenced when assigning temperature or pressure loads to objects. This feature allows description of more complex distributions of temperature and pressure over the structure. Joint patterns by themselves create no loads on the structure.
> Frames	> **Sections** – Use to assign frame section properties to frame objects.

Table 10-4 Advanced > Assign

Command	Description
> Frames (continued)	> **Property Modifiers** – Modification factors can be defined as part of frame section properties **and** assigned directly to frame objects. Note that when modification factors are assigned directly to a frame that also has modification factors defined as part of its frame section properties, the two factors are multiplied. Therefore, it is intended that modification factors be specified using a frame section property definition **or** a frame assignment, not both. The definition of modification factors as part of frame section properties allows application of modification factors whenever a particular property definition is used. The assignment made using this command allows application of modification factors on a "line-object-by-line-object" basis (although multiple objects can be selected). This may be needed to incorporate the effect of cracking of sections or the enhancement in a particular property of the section because of the presence of other members that may, for some reason, not have been modeled. CSiBridge allows these properties to be changed for frame objects.
	> **Material Property Overwrites** – By default, the program uses the material properties associated with the section assigned to the frames. Use this command to specify that a previously defined material property be assigned to a selected object(s).
	> **Release/Partial Fixity** – Use to assign end releases and partial fixity to selected frame objects.
	> **Local Axes** – The program automatically determines the local axis for frames from their connectivity and orientation. This command can be used to modify those axes when necessary.
>	> **Reverse Connectivity** – When a frame object is drawn, the first end point created is the "End I" or the "Start" of the object, and the second end point is the "End J" or the "End" of the object. The local 1 axis of the object runs from End I to End J. The local axes affect the assignments of some properties and loads, as well as the interpretation of results. Make sure that the local 1 axis of each frame object is in the direction that makes the most sense for the model. This command can be used to switch, or reverse, the local I and J Ends of a frame object. This reversing of ends results in a change in the orientation of the object's local axes, allowing the user to change the local axes to be consistent with other members, if so desired.

Table 10-4 Advanced > Assign

Command	Description
> Frames (continued)	> **End (Length) Offsets** – Use to assign frame end offsets to selected frame objects.
	> **Insertion Point** – Assign both the cardinal point and any joint offsets to selected frame/cable objects. This feature is useful, as an example, for modeling beams and columns when the beams do not frame into the center of the column. It will not generally be needed for modeling cables. Joint offsets are specified along with the cardinal point as part of the insertion point assignment, even though they are independent features. Joint offsets are used first to calculate the object axis and therefore the local coordinate system, then the cardinal point is located in the resulting local 2-3 plane.
	> **End Skews** – Use to assign skews to selected frame objects.
	> **Fireproofing** – Specify the type and density of fireproofing applied to selected frames. The program automatically add the weight of the fireproofing to all load cases specified to include self weight.
	> **Output Stations** – Assign output stations to selected frame objects. For each load pattern and load combination, the frame object internal forces and moments are computed and reported at each output station on the frame. The spacing of the stations may be based on a maximum spacing size or a minimum number of stations that are then spaced equally along the frame.
	> **P-Delta Force** – This is a specialized option intended primarily as a substitute for a true P-delta analysis. We recommend that you use nonlinear static analysis to calculate P-delta effects.
	> **Lane** – Assign frame objects to defined lanes as part of a bridge analysis process.
	> **Tension/Compression Limits** – An upper limit on the amount of tension and compression force supported by a frame object can be assigned. The behavior modeled is nonlinear but elastic. For example, assume a compression limit of zero has been set. If the object tries to go into axial compression, it will shorten without any stiffness. If the load reverses, it will recover its shortening with no stiffness, then engage with full stiffness when it reaches its original length.
	> **Hinges** – Assign hinge definitions to selected frame objects.

Table 10-4 Advanced > Assign

Command	Description
> Frames (continued)	> **Hinge Overwrites** – Assign an auto subdivision at the selected frame or modify the hinge behavior so that the hinges cannot drop load.
	> **Line Springs** – Assign line springs to frame objects. Line springs can be assigned in any of the local axes directions of a frame object. CSiBridge distributes the springs associated with the frame object to all of the nodes associated with the internal-to-CSiBridge (analysis model) representation of the line object. Note that internally CSiBridge may mesh (break up) a line object into several elements with associated points between each element.
	> **Frames > Line Mass** – Assign line mass to frame objects. Every object contributes mass to the structure from the mass density of its material. Use this command to assign additional line mass to a frame. Any additional mass assigned to a frame object is added to the object mass to give the total mass of the structure. The additional mass might be used to account for partitions, cladding, and the like.
	> **Material Temperature** – Assign a material temperature to selected frame objects. The material temperature is the temperature at which temperature-dependent properties are evaluated for the object. The properties at this fixed temperature are used for all analyses, regardless of any temperature changes experienced by the object during loading. The object material temperatures are (a) interpolated over an object from values given over joints, or (b) considered as uniform.
	> **Automatic Frame Mesh** – Assign automatic meshing to selected straight or curved frame objects.
> Areas	> **Sections** – Assign area section definition to selected areas.

Table 10-4 Advanced > Assign

Command	Description
Areas (continued)	> **Stiffness Modifiers** – Modification factors can be defined as part of area section properties **and** assigned directly to area objects. Note that when modification factors are assigned directly to an area object that also has modification factors defined as part of its area section properties, the two factors are multiplied. Therefore, it is intended that you specify modification factors using area section property definition **or** area object assignment, not both. The definition of modification factors as part of area section properties allows application of modification factors on the basis of section type. The area object assignment allows application of modification factors on an "object-by-object" basis (although multiple objects can be selected) regardless of section type. Note that the modification factors affect only the ***analysis*** properties. They do not affect the ***design*** properties. The CSiBridge default for all modification factors is 1.
	> **Material Property Overwrites** – Specify that a previously defined material property be assigned to a selected object(s).
	> **Thickness Overwrites (Shells)** – Use to modify the thickness of an area object.
	> **Local Axes** – Each Shell object (and other types of area objects) has its own object local coordinate system used to define Material properties, loads and output. The axes of this local system are denoted 1, 2, and 3. The first two axes lie in the plane of the object with a user-specified orientation; the third axis is normal. It is important to understand the definition of the object local 1-2-3 coordinate system and its relationship to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems. The user defines the local systems that simplify data input and interpretation of results. In most structures the definition of the object local coordinate system is extremely simple. The program also provides advanced options to define the orientation of the tangential local 1 and 2 axes, with respect to an arbitrary reference vector when the object coordinate angle, ang, is zero. If ang is different from zero, it is the angle through which the local 1 and 2 axes are rotated about the positive local 3 axis from the orientation determined by the reference vector. The local 3 axis is always normal to the plane of the object. Use this command to rotate the local axis 2 of an object around the local axis 3.

Table 10-4 Advanced > Assign

Command	Description
> Areas (continued)	> **Reverse Local 3 Axes Direction** – The orientation of the area local 3 axis (which is perpendicular to the plane of the area) is very important. The pressure load and other assignments are all made in conjunction with the local 3 axis. The local 3 axis for area objects is assigned automatically by the program according to the node numbering and at times may not be as desired. Make sure that the area local 3 axis is in the desired direction, otherwise all assignments will be reversed on the area object. Use this command to reverse the orientation of the Local 3 axis.
	> **Area Springs** – Assign area springs to area objects. CSiBridge distributes the springs associated with the area object to all of the nodes associated with the internal-to-CSiBridge (analysis model) representation of the area object. Note that in some cases, internally CSiBridge may mesh (break up) an area object into several elements with associated joints between each element.
	> **Area Mass** – Every object contributes mass to the structure from the mass density of its material. Use this command to assign additional area mass to an area object. Note that the additional area mass is considered by CSiBridge only if the mass source is based on object masses and additional masses, not from a specified load combination. The additional area mass is applied only in the three translational degrees of freedom.
	> **Material Temperature** – Use this command to assign selected area objects an object material temperature. This is the temperature at which temperature dependent properties are evaluated for the object. Temperature dependent properties are internal to the program.
	> **Automatic Area Mesh** – CSiBridge allows area objects to be divided into smaller objects in the object-based model, and meshed into elements in the analysis model. Dividing the object is accomplished using the **Advanced > Edit > Areas > Divide Areas** command. Meshing an area object is accomplished using the **Assign** menu **> Area > Automatic Area Mesh** command.

Table 10-4 Advanced > Assign

Command	Description
> Areas (continued)	> **Generate Edge Constraints** – Add or remove edge constraints from area objects in the model. This command can be useful to model transitions where two incompatible meshes meet along an edge, for example where a finer mesh in one area has elements half as wide as elements in the coarser mesh in the other area.
More > Cable >	> **Cable Properties** – Assign a cable property to line objects that have been drawn as cables.
	> **Property Modifiers** – Modification factors can be defined as part of cable properties **and** assigned directly to cable objects. Note that when modification factors are assigned directly to a cable that also has modification factors defined as part of its cable properties, the two factors are multiplied. Therefore, it is intended that modification factors be specified using a cable property definition **or** a cable assignment, not both. The definition of modification factors as part of cable properties allows application of modification factors whenever a particular cable property definition is used. The property modifier assignment allows application of the modification factors on a "cable-object-by-cable-object" basis (although multiple cables can be selected). CSiBridge allows you to change these properties for frame, cable, and area objects.
	> **Material Property Overwrites** – By default, the program uses the material properties associated with the section assigned to the cables. Use this command to specify that a previously defined material property be assigned to a selected object(s).
	> **Reverse Connectivity** – When a cable object is drawn, the first end point created is the "End I" or the "Start" of the object, and the second end point is the "End J" or the "End" of the object. The local 1 axis of the object runs from End I to End J. The local axes affect the assignments of some properties and loads, as well as the interpretation of results. Make sure that the local 1 axis of each cable object is in the direction that makes the most sense. Use this command to switch, or reverse, the local I and J Ends of a cable object. This reversing of ends results in a change in the orientation of the object's local axes, allowing the user to change the local axes to be consistent with other members, if so desired.
	> **Insertion Point** – Assign any joint offsets to selected cable objects. Joint offsets are used to calculate the object axis.

Table 10-4 Advanced > Assign

Command	Description
> More > Cable (continued)	> **Output Stations** – For each load pattern and load combination, the cable object internal forces and moments are computed and reported at each output station on the cable. The spacing of the stations may be based on a maximum spacing size or a minimum number of stations that are then spaced equally along the cable.
>	> **Line Mass** – Every object contributes mass to the structure from the mass density of its material. Use this command to assign additional line mass to a cable. Any additional mass assigned to a cable is added to the object mass to give the total mass of the building. The additional mass might be used to account for cable coatings and the like. The additional line mass is considered by CSiBridge only if the mass source is to be based on object masses and additional masses, not from a specified load combination. Also, the additional line mass is applied only in the three translational degrees of freedom
	> **Material Temperature** – The material temperature is the temperature at which temperature-dependent properties are evaluated for the object. The properties at this fixed temperature are used for all analyses, regardless of any temperature changes experienced by the object during loading. The object material temperatures are (a) interpolated over an object from values given over joints, or (b) considered as uniform.
More > Tendon	> **Tendon Properties** – Assign a tendon section to a line object that was drawn as a tendon.
	> **Local Axes** – The program automatically determines the local axes for tendons from their connectivity and orientation. This command can be used to modify those axes when necessary.

Table 10-4 Advanced > Assign

Command	Description
> More > Tendon (continued)	> **Tension/Compression Limits** – An upper limit on the amount of tension and compression force supported by a tendon object can be assigned. The behavior modeled is nonlinear but elastic. For example, assume a compression limit of zero has been set. If the object tries to go into axial compression, it will shorten without any stiffness. If the load reverses, it will recover its shortening with no stiffness, then engage with full stiffness when it reaches its original length. This feature is useful for modeling cables and braces that can reasonably be represented by a single straight object when the analysis is focused more on the effect of the cable/brace on the structure than on the detailed behavior of the cable/brace itself. To model the deformation of the cable or brace in detail, break the cable/brace into several sub-objects, and use large-displacements analysis without compression limits. Under compression, the cable/brace will buckle out of the way in a more realistic representation of the true behavior
>	> **Material Temperature** – The material temperature is the temperature at which temperature-dependent properties are evaluated for the object. The properties at this fixed temperature are used for all analyses, regardless of any temperature changes experienced by the object during loading. The object material temperatures are (a) interpolated over an object from values given over joints, or (b) considered as uniform. Use this command to assign a material temperature as a uniform temperature or as an average of the temperature for the joints as applied using a joint pattern.
> More > Solid	> **Properties** – Assign previously define solid properties to selected solid objects.
	> **Local Axes** – Specify the local axes for the selected solid object(s).
	> **Surface Spring** – Assign solid surface springs to selected solid objects. Solid surface springs can be assigned on any face of the solid object
	> **Material Temperature** – Each solid object can be assigned an element material temperature. This is the temperature at which temperature-dependent properties are evaluated for the element.
	> **Switch Faces** – Switch the faces of a solid object.

Table 10-4 Advanced > Assign

Command	Description
> More > Solid (continued)	> **Automatic Solid Mesh** – Use the options on the form that displays when this command is used to specify how CSiBridge™ internally subdivides the selected solid object(s).
> More > Link/Support	> **Link/Support Properties** – Assign link properties to a link object drawn as a one joint link or a two joint link, or that was drawn using the **Advanced > Draw > Quick Draw** Link command.
	> **Frequency Dep. Link Properties** – Assign a frequency dependent property to an existing link or add, modify or delete the definition of frequency dependent link properties. Frequency dependent link properties are used in steady state and power spectral density analysis.
	> **Local Axes** – Each link/support object has its own element local coordinate system used to define force deformation properties and output. The axes of this local system are denoted 1, 2, and 3. The first axis is directed along the length of the object and corresponds to extensional deformation. The remaining two axes lie in the plane perpendicular to the element and have a user-specified orientation; those directions correspond to shear deformation. It is important to clearly understand the definition of the element local 1- 2-3 coordinate system and its relationship to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems. It is up to the user to define local systems that simplify data input and interpretation of results.
	> **Reverse Connectivity** – Switch the Start Joint (I-end) and End Joint (J-end).
> More	> **Assign to Group** -- Select the objects to be included in a group. When making a selection of objects to perform an operation, consider if those same objects are likely to be selected again, potentially for another operation. If so, assigning those objects to a group can save time.
	> **Update All Generated Hinge Properties** – Update auto hinge properties. It is not necessary to select any objects in the model before using this command.

Table 10-4 Advanced > Assign

Command	Description
> More (continued)	> **Clear Display of Assigns** – After performing an assignment operation, the active display window shows all assignments for that type of assignment. For example, when force loads are assigned to joints, the active window displays on the model the location, direction, and a value for the force that has been applied for the specified load pattern. Use this command to remove the display of assignments.
	> **Copy Assigns / Paste Assigns** – Copy assignments from one object and then paste them to another object of the same type (i.e., point to point, line to line, area to area, solid to solid, link to link).

10.5 Advanced > Assign Loads

Table 10-5 describes the command on the *Assign Loads* panel of the **Advanced** tab.

Table 10-5 Advanced >Assign Loads

Command	Description
> Joints	> **Forces** – The force load is used to apply concentrated forces and moments at the joints. Values may be specified in a fixed coordinate system (global or user-defined system) or the joint local coordinate system. All forces and moments at a joint are transformed to the joint local coordinate system and added together. Forces and moments applied along restrained degrees of freedom add to the corresponding reaction, but do not otherwise affect the structure. Use this command to assign a force load to a selected joint(s).
	> **Displacements** – Assign displacement loads to selected joints. Displacement should be applied to restrained joints only. The joint restraint should be in the degree of freedom in which the load is placed.
	> **Vehicle Response Components** – Overwrite the program assumed components and their values that contribute to the reaction or moment of a joint as a result of vehicle loading.

Table 10-5 Advanced >Assign Loads

Command	Description
> Frames	> **Gravity** – Add the factored self weight of the members as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the static load pattern.
	> **Point** – Assign concentrated forces and moments along the length of frame objects. The loads may be as simple or as complicated as required.
	> **Distributed** – Assign distributed (Uniform and Trapezoidal) forces and moments along the length of frame objects. The loads may be as simple or as complicated as required.
	> **Temperature** – Assign a temperature load to one or more selected frames. The temperature load creates thermal strain in the frame object. This strain is given by the product of the material coefficient of thermal expansion and the temperature change of the object, which is the temperature being specified here.
	> **Strain** – Specify a strain load on the selected frames.
	> **Deformation** – Assign deformation loads to selected frames.
	> **Target Force** – Assign a target force load to one or more selected frames.
	> **Auto Wave Loading Parameters** – Specify overwrites for offshore wave load analyses. Wave load overwrites are basic properties that apply only to the frame objects to which they are specifically assigned. Default values are provided for all wave load overwrite items. Thus, specifying overwrites it is not required. However, at least review the default values for the overwrite items and change them if necessary to make sure they are acceptable.
	> **Open Structure Wind Parameters** – Specify overwrites for wind loads on open structures. Wind load overwrites are basic properties that apply only to the frame objects to which they are specifically assigned. Default values are provided for all wind load overwrite items. Thus, it is not required that you specify or change any of the overwrites. However, at least review the default values for the overwrite items to make sure they are acceptable.

Table 10-5 Advanced >Assign Loads

Command	Description
> Frames (continued)	> **Vehicle Response Components** – Overwrite the program assumed components and their values that contribute to the reaction or moment of a frame as a result of vehicle loading. Assign vehicle response overwrites to selected frame objects.
> Areas	> **Gravity (All)** – Add a factored self weight of the members as a force in any of the global directions. The self-weight load itself acts equally on all objects of the structure and always in the global Z direction. It is recommended that the actual self weight of the structure be included in the definition of the static load patterns.
	> **Uniform (Shell)** – Apply uniformly distributed forces to the mid surfaces of area objects. The direction of the loading may be specified in a fixed coordinate system (global or user-defined system) or in the object local coordinate system.
	> **Uniform to Frame (Shell)** – Apply uniformly distributed forces to the frames of area objects. The direction of the loading may be specified in a fixed coordinate system (global or user-defined system) or in the object local coordinate system.
	> **Surface Pressure (All)** – Surface pressure always acts normal to the area object face. Positive pressures are directed towards the interior of the object. The pressure may be constant over the face or interpolated from values given at the joints. The values at the joints are obtained from joint patterns and need not be the same for different faces. Joint patterns can be used to easily apply hydrostatic pressure.
	> **Pore Pressure (Plane, Asolid)** – Assign pore pressure to an area object(s). Pore pressure loads are used to model the drag and buoyancy effects of a fluid within a solid medium.
	> **Temperature (All)** – Assign temperature load to one or more selected area objects. The temperature load creates thermal strain in an area object. This strain is given by the product of the material coefficient of thermal expansion and the temperature of the object.
	> **Strain (All)** – Specify strain load on selected area objects.

Table 10-5 Advanced >Assign Loads

Command	Description
> Areas (continued)	> **Rotate (Asolid)** – Apply centrifugal force to an object. Each object is assumed to rotate about its own axis of symmetry at a constant angular velocity. The angular velocity creates a load on the object that is proportional to its mass, distance from the axis of rotation, and the square of the angular velocity. This load acts in the positive radial direction and is apportioned to each joint of the object. No Rotate Load will be produced by an object with zero mass density.
	> **Wind Pressure Coefficient (All)** – Apply pressure coefficients to selected area objects.
	> **Vehicle Response Components (All)** – Overwrite the program assumed components and their values that contribute to the reaction or moment of an area object as a result of vehicle loading. Assign vehicle response overwrites to selected area objects.
> More > Cable Loads	> **Gravity** – Add the factored self weight of the members as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the static load pattern.
	> **Distributed** – Assign distributed (Uniform) forces and moments along the length of cable objects. The loads may be as simple or as complicated as required. Loads are specified in force-per-length or moment-per-length units.
	> **Temperature** – The temperature load creates thermal strain in the cable object. This strain is given by the product of the material coefficient of thermal expansion and the temperature change of the object, which is the temperature being specified here.
	> **Strain** – Specify the strain load on the selected cable(s).
	> **Deformation** – Assign deformation load to selected cables.
	> **Target Force** – Assign target force to selected cables.
	> **Vehicle Response Component** – Overwrite the program assumed components and their values that contribute to the reaction or moment of a cable as a result of vehicle loading. Assign vehicle response overwrites to selected cables.

Table 10-5 Advanced >Assign Loads

Command	Description
More > Tendon Loads	> **Gravity** – Add the factored self weight of the members as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the static load pattern.
	> **Tendon Force Stress** – Assign tension force/stress on a selected tendon.
	> **Temperature** – Assign temperature load to a selected tendon. The temperature load creates thermal strain in a tendon object. This strain is given by the product of the material coefficient of thermal expansion and the temperature change of the object, which is the temperature being specified here.
	> **Strain** – Specify the strain load on selected tendons.
	> **Deformation** – Assign deformation load to one or more selected tendons.
	> **Target Force** – Assign target force load to one or more selected tendons.
	> **Vehicle Response Components** – Overwrite the program assumed components and their values that contribute to the reaction or moment of a selected tendon as a result of vehicle loading.
More > Solid Loads	> **Gravity** – Add the factored self weight of the objects as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the static load pattern. The load is calculated based on the mass density of the material used to define the material assigned to the corresponding section.
	> **Surface Pressure** – Surface pressure always acts normal to the element face. Positive pressures are directed towards the interior of the element. The pressure may be constant over the face or interpolated from values given at the joints. The values at the joints are obtained from Joint Patterns, and need not be the same for different faces. Joint Patterns can be used to easily apply hydrostatic pressure.
	> **Pore Pressure** – Assign a pore pressure load to selected solid objects. Pore pressure loads are used to model the drag and buoyancy effects of a fluid within a solid medium.

Table 10-5 Advanced >Assign Loads

Command	Description
> More > Solid Loads (continued)	> **Temperature** – Assign a temperature load to one or more selected solid objects. The temperature load creates thermal strain in the solid element. This strain is given by the product of the material coefficient of thermal expansion and the temperature change of the object, which is the temperature being specified here.
	> **Strain** – Specify the strain load on selected solid objects.
	> **Vehicle Response Components** – Overwrite the program assumed components and their values that contribute to the reaction or moment of a solid object as a result of vehicle loading.
> More > Link/Support Loads	> **Gravity** – Add the factored self weight of the members as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the static load pattern.
	> **Vehicle Response Components** – Overwrite the program assumed components and their values that contribute to the reaction or moment of a selected link as a result of vehicle loading.

10.6 Advanced > Analyze

Table 10-6 describes the command on the *Analyze* panel of the **Advanced** tab.

Table 10-6 Advanced > Analyze

Command	Description
> Frame Design > Steel	> **View/Revise Preferences** – Preferences are basic settings that control design parameters, including the design code. Default values that generally reflect the design code are provided so that specification of the individual parameters is not necessary; however, it is advisable to review the preferences to ensure that they are acceptable, and where needed, to change them.

Table 10-6 Advanced > Analyze

Command	Description
> Frame Design > Steel (continued)	> **View/Revise Overwrites** – Overwrites are basic properties that apply only to the frame/cable elements to which they are specifically assigned. Default values are provided for all overwrite items. Thus, it is not required that overwrites be specified. However, at least review the default values for the overwrite items and change them if necessary to make sure they are acceptable. Some of the default overwrite values are based on preferences. Thus define the preferences before defining the overwrites (and before designing or checking any frame members). When changes are made to overwrite items, the program applies the changes only to the elements to which they are specifically assigned; that is, to the elements that are selected when the overwrites are changed.
	> **Select Design Groups** – Elements can be grouped for steel frame design. When a group is specified for design, all elements in the group are given the same section.
	> **Select Design Combos** – It is not necessary to run the analysis or select an object before using the command. Use the command to review and modify the design load combinations used during design.
	> **Set Displacement Targets** – For displacement optimization, CSiBridge predicts which members should be increased in size to control the displacements based on the energy per unit volume in the members. The members with more energy per unit volume are increased in size a larger percentage than those with smaller energies per unit volume. Some members with small energy per unit volume may be decreased in size if they are still acceptable for strength considerations. Auto select lists must be assigned to the frame members for the displacement optimization to work. When CSiBridge increases or decreases a section size, it uses the available sizes in the auto select list. If the appropriate sections are not included in the auto select lists, the desired displacement target may never be reached no matter how many times the analysis and design are rerun.

Table 10-6 Advanced > Analyze

Command	Description
> Frame Design > Steel (continued)	> **Set Time Period Targets** – For time period targets, specify a mode and a target period for that mode. Then, during design, CSiBridge automatically optimizes the structure to meet the target period using energy considerations. During the optimization process, members with large energy per unit volume are changed more than those with small energy per unit volume. Auto select section lists must be assigned to frame members for the time period optimization to work. When CSiBridge increases or decreases a section size, it uses the available sizes in the auto select section list. If the appropriate sections are not included in the auto select lists, the displacement target may never be met, no matter how many times the analysis and design are rerun.
	> **Start Design/Check of Structure** – Starts the steel frame design process.
	> **Interactive Steel Frame Design** – Use to review the design results for any frame element and to interactively change the design overwrites and immediately view the results.
	> **Display Design Information** – Display design input and output parameters in the active window.
	> **Make Auto Select Section Null** – Replace the Auto Select section assignments for the selected members with the current design section previously chosen from the auto select group. This is an irreversible action, i.e. once the Auto Select sections are turned off they will no longer be available for optimal design. After using this command, Auto Select sections would need to be reassigned if they were to once again be included in the design selection process.
	> **Change Design Section** – Because design is an iterative process, after a steel frame design has been run, it may be necessary to change the design section property assigned to one or more elements. CSiBridge allows multiple design iteration runs without rerunning the analysis, which can save computing time. Use this command to change the design section; then, redesign the structure without rerunning the analysis.
	> **Reset Design Section to Last Analysis** – In some instances design sections may change several times (before the analysis is rerun) and then the design section for one or more frame elements may need to be restored to match the last used analysis section. This command is a quick and easy way to reset the design section to match the last analysis section.

Table 10-6 Advanced > Analyze

Command	Description
Frame Design > Steel (continued)	**Verify Analysis vs Design Section** – When the iterative design process is complete, the last used analysis section property for a frame element and the current design section property for that frame element should be the same. If this is not the case, the frame element may not have been designed for the correct forces. This command is useful for verifying that the last used analysis section and the current design section are the same for all steel frame elements in the model. When the command is used, CSiBridge identifies how many frame elements with the Steel Frame design procedure have different analysis and design sections and then selects those frame elements, if you ask it to. Typically this command would be used after the final design iteration to verify that the analysis and design properties used are consistent. It is not necessary to select any elements before using this command. This command automatically checks all frame sections with the Steel Frame design procedure.
	> **Verify All Members Passed** – Report if structural members passed the stress/capacity check.
	> **Reset All Steel Overwrites** – Resets the overwrites for all frame sections with the Steel Frame design procedure to their default values. It is not necessary to make a selection before using this command. This command automatically applies to all frame sections with the Steel Frame design procedure. The command can be used to reverse changes made using the **View/Review Overwrites** command.
	> **Delete Steel Design Results** – Deletes all of the steel frame design results. It is not necessary to make a selection before using this command. This command automatically applies to all frame sections with the Steel Frame design procedure.
Frame Design > Concrete	> **View/Revise Preferences** – Preferences are basic settings that control design parameters, including the design code. Default values that generally reflect the design code are provided so that specification of the individual parameters is not necessary; however, it is advisable to review the preferences to ensure that they are acceptable, and where needed, to change them.

Table 10-6 Advanced > Analyze

Command	Description
Frame Design > Concrete (continued)	> **View/Revise Overwrites** – Overwrites are basic properties that apply only to the frame/cable elements to which they are specifically assigned. Default values are provided for all overwrite items. Thus, it is not required that overwrites be specified. However, at least review the default values for the overwrite items and change them if necessary to make sure they are acceptable. Some of the default overwrite values are based on preferences. Thus define the preferences before defining the overwrites (and before designing or checking any frame members). When changes are made to overwrite items, the program applies the changes only to the elements to which they are specifically assigned; that is, to the elements that are selected when the overwrites are changed.
	> **Select Design Combos** – It is not necessary to run the analysis or select an object before using the command. Use the command to review and modify the design load combinations used during design.
	> **Start Design/Check of Structure** – Starts the steel frame design process.
	> **Interactive Concrete Frame Design** – Use to review the design results for any frame element and to interactively change the design overwrites and immediately view the results.
	> **Display Design Information** – Display design input and output parameters in the active window.
	> **Change Design Section** – Because design is an iterative process, after a concrete frame design has been run, it may be necessary to change the design section property assigned to one or more elements. CSiBridge allows multiple design iteration runs without rerunning the analysis, which can save computing time. Use this command to change the design section; then, redesign the structure without rerunning the analysis.
	> **Reset Design Section to Last Analysis** – In some instances design sections may change several times (before the analysis is rerun) and then the design section for one or more frame elements may need to be restored to match the last used analysis section. This command is a quick and easy way to reset the design section to match the last analysis section.

Table 10-6 Advanced > Analyze

Command	Description
Frame Design > Concrete (continued)	> **Verify Analysis vs Design Section** – When the iterative design process is complete, the last used analysis section property for a frame element and the current design section property for that frame element should be the same. If this is not the case, the frame element may not have been designed for the correct forces. This command is useful for verifying that the last used analysis section and the current design section are the same for all concrete frame elements in the model. When the command is used, CSiBridge identifies how many frame elements with the Concrete Frame design procedure have different analysis and design sections and then selects those frame elements, if you ask it to. Typically this command would be used after the final design iteration to verify that the analysis and design properties used are consistent. It is not necessary to select any elements before using this command. This command automatically checks all frame sections with the Concrete Frame design procedure.
	> **Verify All Members Passed** – Report if structural members passed the stress/capacity check.
	> **Reset All Concrete Overwrites** – Resets the overwrites for all frame sections with the Concrete Frame design procedure to their default values. It is not necessary to make a selection before using this command. This command automatically applies to all frame sections with the Concrete Frame design procedure. The command can be used to reverse changes made using the View/Review Overwrites command.
	> **Delete Concrete Design Results** – Deletes all of the concrete frame design results. It is not necessary to make a selection before using this command. This command automatically applies to all frame sections with the Concrete Frame design procedure.
More	> **Overwrite Frame Design Procedure** – Specify that a line object should not be designed. Line objects without a designated design procedure display **None** for the Design Procedure on the Line Information form.
	> **Lateral Bracing** – Specify design of lateral bracing for the selected frame object(s).

10.7 Advanced > Tools

Table 10-7 describes the command on the *Tools* panel of the **Advanced** tab.

Table 10-7 Advanced > Tools

Command	Description
Add/Show Plut Ins	A plug in is a software tool from an external source (i.e., not from CSi) that works inside CSiBridge to provide additional features to the program. For example, a plug in may add import/export capabilities, customized model-building templates, customized design or other post-processing of results, or it may perform parametric studies. Many other possibilities can be envisioned.
CSiLoad Optimizer	Use in determining an optimal set of loads, including cable tensioning, to achieve specified goals in a structural model. The loads to be optimized can be applied in any static load case, which can be of type linear, nonlinear, or staged-construction. The goals are specified as the values to be attained for response quantities such as joint displacements, generalized displacements, joint reactions, member forces or moments, and/or bridge superstructure forces and moments. The optimization operation consists of determining the scale factors of variable loads in the static load case linear to best meet these goals.

Bibliography

ACI, 2007. Building Code Requirements for Structural Concrete (ACI 318-08) and Commentary (ACI 318R-08), American Concrete Institute, P.O. Box 9094, Farmington Hills, Michigan.

AASHTO, 2009. AASHTO Guide Specifications for LRFD Seismic Bridge Design. American Association of Highway and Transportation Officials, 444 North Capital Street, NW Suite 249, Washington, DC 2001

Canadian Standards Association (CSA), 2006. Canadian Highway Bridge Design Code. Canadian Standards Association, 5060 Spectrum Way, Suite 100, Mississauga, Ontario, Canada, L4W 5N6. November.

CSI, 2011. Analysis Reference Manual. Computers and Structures, Inc., 1995 University Avenue, Berkeley, California, 94704.