USER INTERFACE SPECIFICATION

FOR

QCONBRIDGE II

VERSION 1.0

SEPTEMBER 20, 2000

RICHARD BRICE, PE
WSDOT

RICHARD PICKINGS, PE
BRIDGESIGHT SOFTWARE

User Interface Specification.doc

REVISION CHART

Version	Primary Author(s)	Description of Version	Date Completed
1.0 - Draft 1	RAB, RDP	Initial draft created for distribution and review comments	
1.0	RAB, RDP	Initial Version. Distributed to review committee.	9/20/00

CONTENTS

LIST OF FIGURES

1 INTRODUCTION

1.1 Purpose

The purpose of this document is to provide a detailed specification of the QConBridge II user interface. These requirements will detail the outwardly observable behavior of the program. The user interface provides the means for you, the engineer, to interact with the program.

This first version of the User Interface Specification is intended to convey the general idea for the user interface design and the operational concept for the software. Many details have been omitted for both clarity and because they have not been addressed yet. This document will be updated with additional detail as our analysis and design activities progress.

1.2 Overview

The QConBridge II program allows you to perform a plane frame structural analysis on an idealized bridge that satisfies the requirements of LRFD 4.6.2, Simplified Analysis. The goal of this analysis is to determine moments, shears, reactions, and displacements in the superstructure and piers; due to a myriad of loading conditions including live load and dead load. The key to performing this analysis is modeling. QConBridge will analyze bridges using mathematical models.

We use computer models to approximate and study that which we wish to understand. If money is of no concern, and pinpoint accuracy is essential, every bridge would be constructed to exacting standards in a laboratory and thoroughly tested before being constructed in the field and opened to traffic. Obviously, this is not practical. The next best thing is to create models of the bridge so that we can approximate its behavior. QConBridge II uses two types of models: Analytical Models and Product Models.

Analytical models are used to simulate the behavior of the bridge. Analytical models are generally very abstract. In the case of plane frame bridge models, the description of a bridge is abstracted down to span lengths, section properties (Area and Moment of Inertia), material properties (Young's Modulus and Density), live load distribution factors, and idealized loads. QConBridge will support analytical models in much the same way QConBridge Version 1 does. This provides the engineer with the most flexibility and control over the analysis of a structure. However, generic analysis models are tedious to construct and are not directly linked to the real bridge structure.

To make modeling of common bridge types more efficient, QConBridge will support product modeling of Prestressed/Precast Concrete I Girder Bridges, Rolled Steel Beam Bridges, and Build-up Steel I-beam Bridges. Product models are used to capture the physical attributes of real bridges. Generally, product models are described with real-world attributes, such as pier location, girder type, or slab dimensions. QConBridge can then use the product model of a structure to create the necessary analysis models and perform the required structural analysis.

The first decision when creating a new QConBridge project is whether the bridge should be represented as a Product Model or an Analysis Model. This choice will put QConBridge into Product Model Mode or Analysis Model Mode. In Product Model Mode, the bridge is described in terms of its real world attributes. QConBridge will automatically generate an analysis model, and allow limited editing capabilities for it. In Analysis Model Mode, the bridge is described in terms of its geometric and structural properties. You will have the responsibility of creating all the analysis models and loading conditions.

QConBridge will support conversion of a Product Model to an Analysis Model. This allows users who feel the need to tweak the automatically generated models the freedom to do so. Once a Product Model is converted to an Analysis Model, it loses all connections back to its original real world description.

QConBridge II uses two main user interface features to simplify use of the program and to cut down on the amount of data that engineers need to type in. Firstly, much of the detailed data needed by the QConBridge II program is stored in libraries that can be reused and shared among workgroups. Libraries also help to reduce input errors by allowing complex data to be input once and then be referred to by name. The second feature is the Bridge Contractor. The Bridge Contractor guides you step-by-step through the project creation process. The Bridge Contractor helps beginning users by leading them through the input process, and it helps experienced users insure that all data pertaining to a project was entered completely and correctly. Together, these two features make the job of creating new bridge projects very easy.

1.3 Basic Goals

Often, the goals for a program are confused with the tasks the program is to perform. This results in programs that are good at performing individual tasks, but may not integrate well into office workflow, or are very difficult and cumbersome to learn and use. Examples of task-oriented programs are GTStrudl and BDS.
QConBridge II will be designed with your goals in mind rather than just tasks. The basic goals for the user interface of QConBridge II are as follows:

1. **Don't make our users look stupid**. This can be achieved by:
    - Not making users guess at what the program is doing. Provide good documentation on internal algorithms and report all intermediate calculations and data used in the algorithms
    - Not making users guess what input data means. Provide good documentation and context-sensitive help on all items to be input to the program
    - Providing well-designed reports that engineers will be proud to place in their calculation workbooks. Allow users to configure what goes into the reports
    - Allowing engineers to engineer. Make it possible to tweak analyses and allow engineers to insert their judgment where possible
2. **Don't allow users to make big mistakes**. This can be achieved by:
    - Providing detailed feedback and visual results so engineers can easily verify program results
    - Reducing manual data input as much as possible
    - Providing comprehensive data validation at the user-interface level and highlighting suspicious input data
    - Allowing users to configure units for input and output of all data
3. **Help our users get an adequate amount of work done**. This can be achieved by:
    - Providing good performance
    - Providing shortcut keys to access commonly-used features
    - Providing integration with other programs on the desktop so engineers spend as little time as possible cutting, pasting, hand calculating, and inputting data
4. **Let users have fun (or at least not get too bored)**. This can be done by:
    - Reporting results in an educational manner
    - Providing interactive features that allow users to visually explore design possibilities

Most of the above goals are difficult, if not impossible to measure directly. However, it is important to keep them in mind as we design and review our user interface.

1.4 Using the Program – The Big Picture

The QConBridge II program solves two basic types of engineering problems: the description and visualization of a bridge structure, and a detailed dead load and live load analysis of a given girder line or pier. A context diagram for direct use of the system by a user (designer) is given in Figure 1.

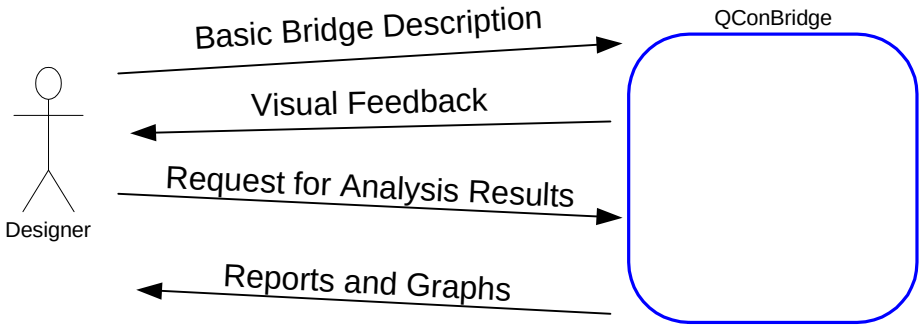


Figure 1 Context Diagram for QConBridge II

We need to face reality. The context diagram above shows only a very small part of the of the bridge design process. In practice, input for this program will come from other sources (hand calcs, plans, other programs), and results from this program will be used in yet more processes until the design is complete. It is also likely that the program will be run many times as the design is refined.
In most scenarios, this means that there is a significant of time spent copying data between processes used in the design. This is an inefficient, mundane, and error-prone way for an engineer to spend his valuable time. Figure 2 shows a slightly expanded view of the bridge design process. As we can see, the designer's job is to control an iterative design loop, making decisions along the way. In reality, much of his work involves manually copying data from one process to another – not exactly the dream of a highly skilled professional, but this is how most engineers perform their work today.

Figure 2 illustrates how most users will use QConBridge II and how it fits in the office process. However, the program will be designed so that advanced users and programmers can better fit the program into the workflow.

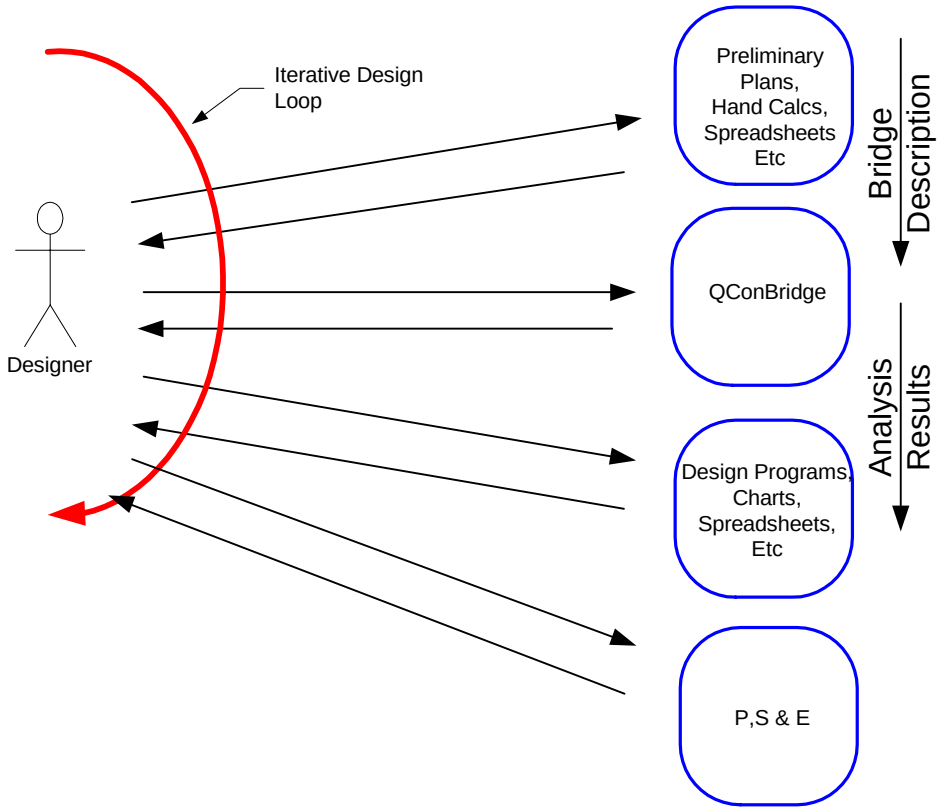


Figure 2 Expanded View of the Bridge Design Process (User manually copying data)

Our solution, which helps to eliminate some of the tedium of copying data, is shown in Figure 3. In this scenario, Designers and application developers create custom programs, or scripts to shuttle data between applications. The advantage to this scenario is that data transfer between applications is automated, and there is less risk for data to be copied incorrectly. The engineer can provide as little or as much automation as necessary or desired. The disadvantage to this scenario is it requires programming skills, or at least a basic understanding of script programming.

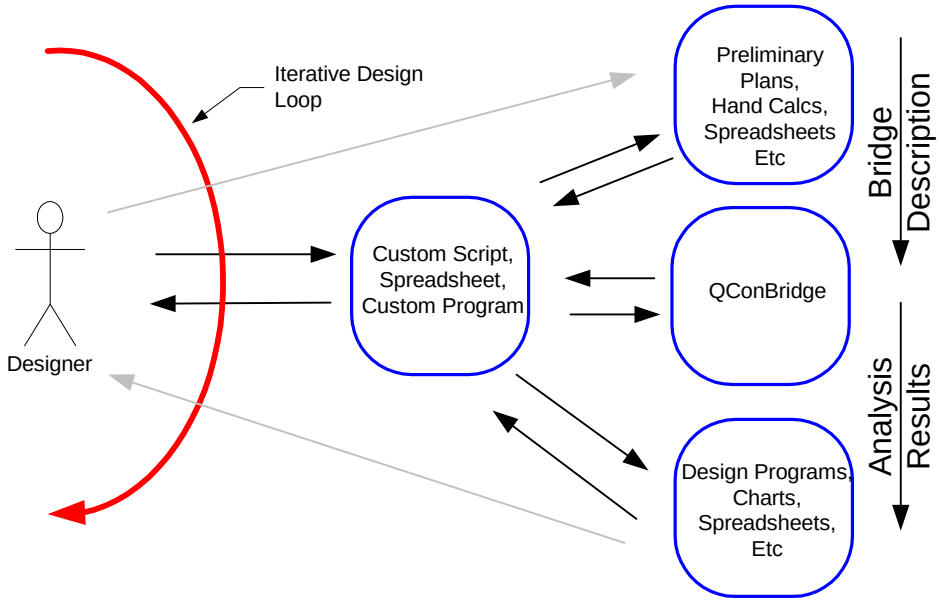


Figure 3 Use of Automation to Streamline Design Process

Note that scripting is optional: users are free to interact with programs individually as they did before. The main purpose of the scripts is to transfer data between applications. Of course, it is possible for scripts or spreadsheets to perform intermediate calculations. This enables engineers to better leverage the standard desktop tools like spreadsheets, word processors, and CADD.

1.5 Avoiding "Interface Schizophrenia"

As elaborated in the System Design Document, QConBridge II allows you to edit idealized bridge analysis models and three different types of product models. It's easy to see how the program could become very confusing to use if the user interface is not designed very carefully.

In order to avoid this type of confusion, the program will behave as if it were five similar, but different programs depending on the type of project being edited. For example, if a precast I girder bridge is being edited, there will be no signs in the user interface of rolled steel or built up plate girder bridges, only input items and interface elements for precast I girder bridges will be visible.

In fact, the only indication that the program supports different bridge types in the user interface will be when you create a new file (and in the Help system). The first question that the Bridge Contractor will ask is "what type of bridge project would you like to create? (Figure 13) After this question is answered, the program will appear to be custom-designed for that type.

Although it is important that the program act as four different programs depending on the project type, it is also very important to remember that this is still a single program. This means that the user interface should provide consistency throughout.

1.6 Common Scenarios

This section describes some common scenarios that the QConBridge II program is designed for.

1.6.1 Analysis of a "Standard" Bridge

A "Standard" bridge is a bridge that can be described by one of QConBridge II's product model descriptions. In this case, there is very little input required to run an analysis. The steps would be as follows for a typical product model bridge:

1. Create a new project file and start off in step 1 of the Bridge Contractor. Select bridge type, number of spans, display units, and specification.
2. Define analysis settings in step 2 of the Bridge Contractor. In most cases, the defaults will suffice.
3. In Step 3 of the Bridge Contractor, define the alignment.
4. In Step 4 of the Bridge Contractor, layout the pier locations and skews. Hold off in defining piers in detail until after superstructure forces are analyzed for the simple support case.
5. In Step 5 of the Bridge Contractor, describe the girder spacing and girder type.
6. In Step 6 of the Bridge Contractor, describe the slab dimensions and barriers. This completes preliminary entry of bridge information for simple supported case.
7. Open an Analysis Results View and view moment and shear diagrams for desired limit states. Check these demands against tabulated capacities for the current girder type. Perform for all girders.
8. Make adjustments in superstructure for girder types and spacing until girder and slab design is adequate.
9. Open Framing Plan Editor and describe pier cap beam and column dimensions.
10. In Analysis Results View, view cap beam and column shears and moments for desired limit states. Compare these demands against hand-calculated capacities. Perform analysis for all piers.
11. Make adjustments in pier dimensions until design is adequate.
12. Run superstructure analysis a final time to insure that flexibility effects from more accurate pier description does not affect superstructure design.
13. Print graphs and reports for calculation package.

1.6.2 Preliminary Force Analysis (Quick & Dirty)

In this case, the alignment and span lengths have been determined, but little else. The preliminary designer wants to get a handle on the basic live load forces to be expected on the bridge. A BAM model is ideal for this use:

1. Create a new project file and start off in step 1 of the Bridge Contractor. Select Bridge Analysis Model for type, number of spans, display units, and specification.
2. Define analysis settings in step 2 of the Bridge Contractor. In most cases, the defaults will suffice.
3. In Step 3 of the Bridge Contractor, define a single stage – this is the default.
4. In Step 4 of the Bridge Contractor, describe the bridge alignment.
5. In Step 5 of the Bridge Contractor, layout the pier locations and skews. Treat piers as simple supports (this is the default).
6. In Step 6 of the Bridge Contractor, define bridge to have a single girder line. Default section properties will have unit values for area, moment of inertia, E, and distribution factors.
7. In Step 7 of the Bridge Contractor, accept default values for deck width. This is not needed because we are not performing a transverse pier analysis. This completes entry of bridge information.
8. Open an Analysis Results View and view moment and shear diagrams for desired limit states.
9. Make adjustments in superstructure for span lengths until desired effects are achieved.
10. Print graphs and reports for calculation package.
11. Can now complete analysis model description by entering properties directly, or start a new bridge project to define a product model with desired attributes.

1.6.3 Other Possible Scenerios

Here are some other scenarios that can be explored:

1. Analysis of a non-standard pier. This could involve exporting a TBAM project from and existing product model or BAM project.
2. Sensitivity Studies. QConBridge can be used to generate graphs for all types of sensitivity studies. This is a case where having an automation interface could save huge amounts of time.

2 SYSTEM OVERVIEW

This section describes the basic concepts of the QConBridge II system. The features described here apply to all project types.

2.1 Projects

As mentioned earlier, QConBridge II is really five programs in one. You will experience QConBridge II in one of its five forms based on the type of project you are working with. When a new QConBridge II project is started, you must select the project tpee. Once the project type is selected, it cannot be changed.
Interface Model
QConBridge II is a Multiple Document Interface (MDI) application that is constrained to have only one project open at a time. QConBridge II has many views for editing bridge models, viewing analysis results, and displaying reports. The MDI Model supports the display of multiple views within the main window of the application. This is exactly what we need with QConBridge II.
Because many views can be open simultaneously for a single project, it will become confusing if more than one project were open at a time. The confusion would be amplified if the open projects were of different types. For this reason, QConBridge II is constrained to have only one project open at a time.
Multiple projects can be viewed at the same time by simultaneously running more than one instance of QConBridge II. When a user attempts to open a project that is currently open, QConBridge automatically "jumps" to the previous instance of the program.

2.2 Editing Goals

QConBridge II has three primary goals for editing input data: 1) At the completion of any editing operation, the project must be in a state that is ready for analysis, 2) Edits can happen in any order, and 3) the user can back out of any edit.

2.2.1 Undo and Redo

Users generally do not make mistakes. They take actions that will lead towards accomplishing their goals. From time to time (actually, quite frequently because it is human nature), a user will discover that his actions are not providing the intended results and he wants to backup to a known place. This is where an Undo/Redo feature comes into play. A user can undo previous actions, step by step, to arrive at a previous state.
QConBridge II implements a multiple explanatory undo/redo with milestones. Each action that the user takes can be undone sequentially. The menu item and tool tip for the undo command provides a explanation of what is about to be undone or redone. For example, if a span is deleted, the tool tip for the undo button might say "Undo: Delete Span".
Users often experiment with software to discover the commands needed to accomplish their goals. Sometimes the experiments are unsuccessful and the user wants to go back in time to the moment when the experiment started. QConBridge supports a Milestone feature to allow a user to quickly jump back to known point in time. A user can undo all actions back to a milestone.

2.3 Units

2.3.1 Display Units

QConBridge II supports a flexible unit system. For display units, users can define *display modes* in the library. These display modes define the unit of measure and the display accuracy for various types of physical quantities. For example, dimensions of length that represent cross section dimensions could be displayed in inches.

2.3.2 LRFD Specification

Some intermediate calculations, such as live load distribution factors, depend upon equations and constants defined in the LRFD Specifications. These equations and constants differ between the SI and US unit version of the specification. Through the Specification Library Entry, the user can select which version of the LRFD specification calculations will be based on.

2.4 Structural Analysis

The primary purpose for QConBridge II is to analyze a bridge structure for dead and live loads. The non-linear editing capability of QConBridge lends itself to continuous processing of the input data. Users can input some data and results displayed on the screen are automatically updated. On slower computers, or for large models, this continuous processing might create unacceptable lag times between edits (i.e. the user has to wait too long for processing to complete before making the next edit). For this reason, QConBridge II allows the user to turn off the automatic processing.

2.4.1 AutoCalc Mode

When AutoCalc Mode is enabled, the structural analysis results are continuous updated. When it is disabled, the user must request that the displays be updated.

2.4.2 License Plate Mode

If a model is changed and AutoCalc is off, results views will contain out of date information. A license plate mode scheme, identical to that used in PGSuper will be used to inform the user that results data is out of date.

2.4.3 Progress Dialog

During analysis calculations, the program shall display a modal progress dialog. This dialog shall contain a progress widget that displays the progress of the structural analysis.

2.4.3.1 Canceling An Analysis

The progress dialog shall have a Cancel button that will cancel the current calculations and return the user back to editing mode. If the program has AutoCalc on and Cancel is selected, the program shall change AutoCalc mode to Off. Any results windows (graphs or reports) shall be put into license plate mode and display the message “Results not Calculated – Press Update to Perform Analysis”.

2.5 Model Editing Issues

This section addresses issues that are common when editing all model types.

2.5.1 Resolving Conflicts Caused by Component Length Edits

Editing span, cap beam, or column lengths can cause inconsistent input data for those items such as loads, cross section data, and hinge locations. This can happen if the item is located using absolute measure and the span length becomes less than the location of the load or hinge. The basic strategy for resolving this conflict is to delete the offending input, indicate a potential problem in the Status Center (Section 7 below), and ignore the data during analysis. If the Status Item is opened, let the user pick one of four different options: Delete Load/Hinge/Section Properties, Convert to Fractional Measure, Move to end of span (loads only), or Move to next span (loads only, not available for loads that go beyond the last span).

The Load Conflict dialog is shown in Figure 4. The Hinge and Section Property Conflict dialogs are similar. This dialog is displayed when a conflict is encountered. Since changing a span length could cause several conflicts, the resolution can be applied to all loads or hinges by selecting the Apply to All check box.

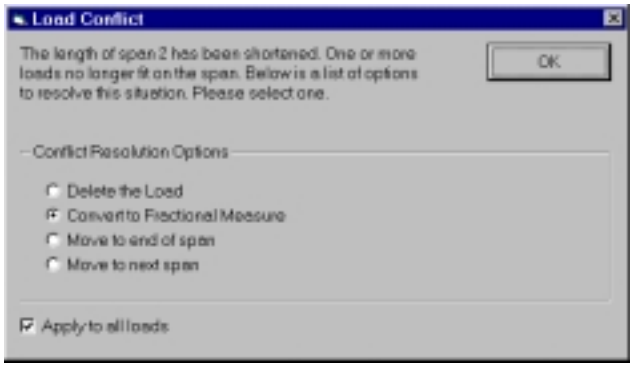


Figure 4 – Load Conflict

2.6 Load Editing

This section addresses load editing issues that are common to all model types.

2.6.1 Editing Load Groups

Load Groups are used to categorize user-defined and built-in loads (in the case of product models). All user-defined loads must be assigned to a load group. Load Groups are either predefined by the Analysis Criteria library Entry, or can be created on the fly for the current project. Figure 5 shows the dialog activated when "Create New Load Group…" is selected from one of the load editing dialogs.

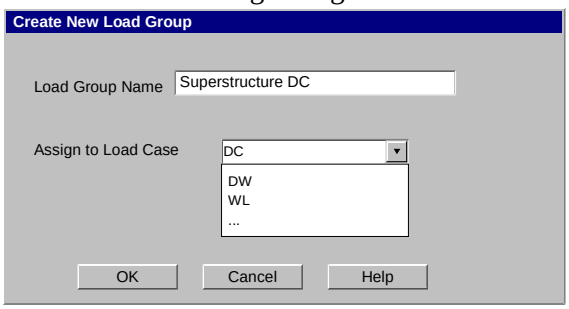


Figure 5 - Create New Load Group

Load Groups can be managed through the Edit Load Groups dialog shown in Figure 6. Note that a load group may not be deleted if it has loads in it. This dialog only lists Load Groups that are local to the current project. Load Groups from the Library Entry are not shown here, but do appear in load editing dialogs. Duplicate Load Groups are not allowed.

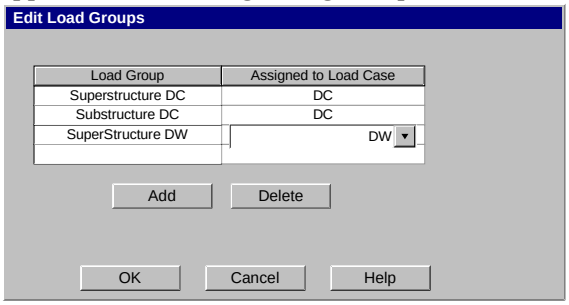


Figure 6 - Edit Load Groups

2.6.1.1 Load Group Conflicts

It is possible that a project-specific load group can conflict with a load group defined in the library. Here are the cases where this can happen:

2.6.1.1.1 New Load Group From Library Conflicts With Existing Load Group in Project

This can happen if the library is changed. In this case, the load group takes on the properties of the library version, and a message is sent to the Status Center stating what happened.

2.6.1.1.2 Library is Changed and Load Group is Orphaned

In this case, the load group becomes local to the project and retains its properties. The load group is retained only if it contains loads.

2.6.2 Load Display

Loads are displayed with a schematic or scaled representation as defined by the view settings. A color scheme is used to show membership in the various load groups. When loads are present in a view, a legend is displayed to define the color scheme.

If two or more loads occupy the same position on the screen, the second and subsequent loads are displayed slightly above and to the right of the first load. A small glyph is also placed on the span at the point of application. While it will be impossible to select a particular load with the mouse in this case, the mouse can be used to pick the first load and the arrow keys can be used to select the rest. The display of multiple loads is shown below.

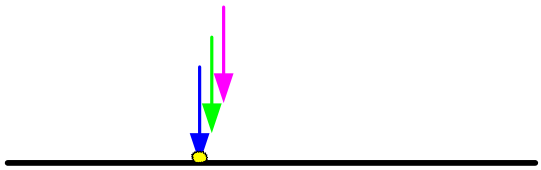


Figure 7 - Display of Overlapping Loads

2.7 Consistent Object Selection

The system shall provide a mechanism so that all objects (spans, piers, poi's, etc.) are selected in a consistent manner. Selected items are drawn using a heavier line weight than when they are unselected. This will make the item look "bold".

2.8 Linkages (References)

Linkages are used throughout the QconBridge II user interface. Linkages allow you to use references to other structures (girderlines and piers) to insure sameness. This reduces the amount of input required by the system. However there are some bugaboos:

2.8.1 Changing a Linkage from Linked to Unique.

When a linked item is changed to become a unique item, it becomes a copy of the original item.

2.8.2 Changing Linkage from Unique to Linked.

Unique items can only become changed to linked items if others do not link to them – chained linkages are not allowed.

2.8.3 Deleting Unique Items.

Unique items may only be deleted if they are not referenced. (could also give option to turn all linked items into copies).

2.8.4 Visual Representation of Linkages

TBD
We need to provide a visual cue at the place where linkages are established, that a link cannot be made. If we don't allow the user to make the link and we provide a visual cue as to why, we don't have to ding them with an error message. We should also disable the delete option for linked items.

2.9 Compatibility with QConBridge I

QConBridge II will read QConBridge I files and convert the project into a Bridge Analysis Model project. Most users will want to upgrade their project files to the new software so there is no sense bothering them with this detail. QConBridge II will automatically save project files in the new format and create a backup file of the old format named *filename*_QConBridge1.qcb. A message will be sent to the Status Center when this is done.

2.10 Context Sensitive Help

Context sensitive help will be provided for all views and dialogs in the program.

2.11 Clipboard Operations

The program shall support clipboard operations on program actors within and between views in the current application. Standard Copy, Cut, and Paste commands shall work similar to industry standard software. Copy, Cut, Paste operations between multiple instances of QConBridge will be supported.

3 PRIMARY MODEL TYPES FOR EDITING

QConBridge II supports three primary model types for analyzing bridge structures. These are Product Models, Bridge Analysis Models (BAM's), and Transverse Bridge Analysis Models (TBAM's). These modes are, in fact, hierarchical: Product Model bridges make use of BAM's and TBAMs', and BAM's make use of TBAM's. This section discusses the user interface for editing and interacting with each model type in detail.

3.1 Product Model Editing

Through it's BAM Editor, QConBridge II can be used to analyze any type of bridge that can be modeled using the "Simplified Method" outlined in LRFD 4.6.2. However, the BAM Editor requires you to describe the bridge using abstract data like section properties and distribution factors. This is cumbersome and error-prone because it requires pre-calculation and manual input of these values.

In order to make it easier to input common WSDOT bridges, QConBridge II provides Product Model Editors for three bridge types: Precast I Girder Bridges, Rolled Steel I Girder Bridges, and Built-Up Plate Girder Bridges. Product Model Editors make it easier to use the program because data is entered in bridge terms, not structural analysis terms. This document contains a dedicated section for each type of bridge.

3.1.1 Product Model Editor Actors

Each Product Model Editor is integrated into the QConBridge II user interface in a similar manner. All Product Model Editors will consist of similar views and deal with the following UI Actors:

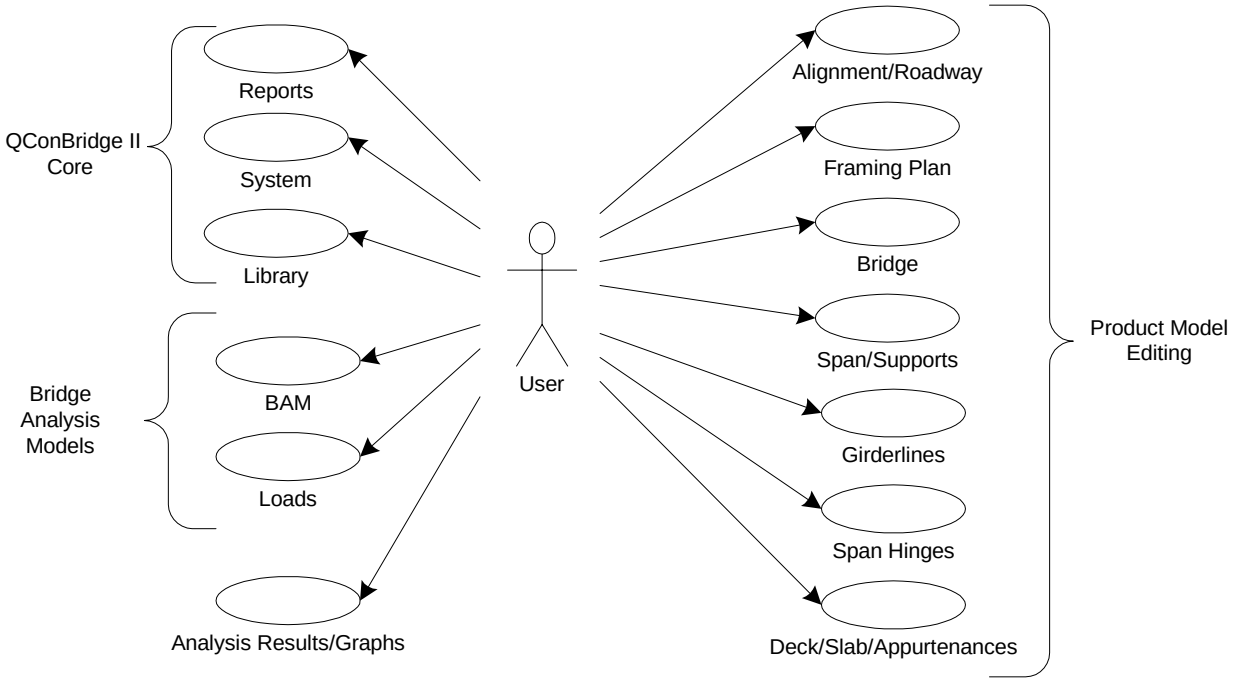


Figure 8 - Product Model Editor UI Actors

3.1.2 Product Model Editing Views

This section discusses the main views for product model editing and the primary interactions in these views.

3.1.2.1 Framing Plan View

The Framing Plan View is a plan view that displays the alignment, pier layout, overall superstructure layout, and is the focal point of a product model-based project. Figure 9 is a schematic of the Framing Plan View.

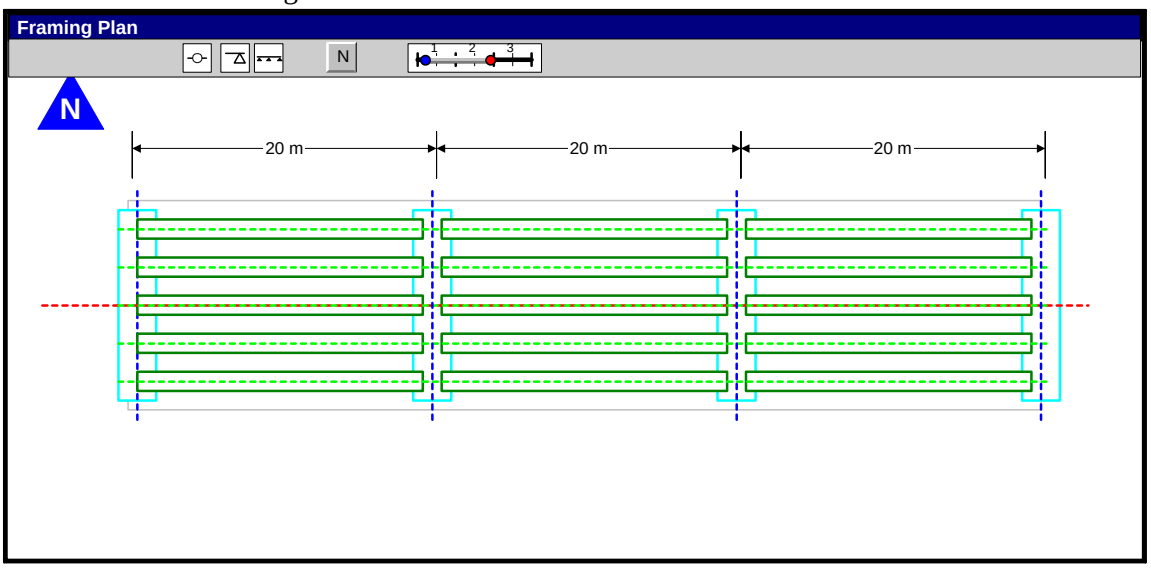


Figure 9 - Product Model Framing Plan View

3.1.2.1.1 KEYBOARD SELECTION

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys select the slab, piers, spans, hinges, and pier/span pairs. The selection sequence using the right arrow key goes as follows, slab → pier 1/span 1 → hinges in span 1 → span 1/ pier 2 → pier 2/span 2 → … → span n/pier n+1 → slab. The order is reversed if the left arrow key is used. Pressing the CTL key makes multiple selections, but the selection sequence changes so that only like items are selected. For example, pier 2 is selected and the CTL key and right arrow key are pressed, pier 3, pier 4, etc, are then selected. The next right arrow key press selects the next item in the sequence after the last pier selected.

Girder lines are selected by using the up and down arrow keys. The up arrow key moves the selection to the left and the down arrow key moves the selection to the right. Holding down the CTL key causes multiple girder lines to be selected.

3.1.2.1.2 MOUSE OPERATIONS

The Framing Plan View displays a plan view of the bridge and supports mouse interaction with many of the product model interface actors as specified in the following table.

When drag operations are not supported, the software will enter drag mode, but the mouse icon will be the ⊗ cursor.

Table 1 – Mouse Actions for Framing Plan View

Actor	Action	Result
Alignment	Single Click	Select Alignment
	CTL/SHIFT-Click	Select Alignment (only one to select)
	Double Click	Activate alignment editing dialog (See Section 3.1.4.3)

	Right Click	Context menu – Properties only active selection
	Drag	⊗
	Ctl-Drag	⊗
Girderline	Single Click	Select Girder line
	CTL/SHIFT-Click	Select multiple girder lines
	Double Click	Activate Girderline Elevation View for selected girderline. See Section 3.1.2.2
	Right Click	Context menu
	Drag	Moves girderline to the location of the girderline nearest drop location. Girderlines in between the source and target girderline are shifted toward the source to fill the void. Current girder spacing is retained.
	Ctl-Drag	Copies source girder into location between girders at drop location. Spacing on either side of new girder is same as spacing at drop location. If girder is dropped outside of girderlines, it becomes an exterior girder with the same spacing as the previous exterior girder. Widens the cap beams and roadway slab to accommodate new girderline. Overhang dimensions with respect to the exterior girder are maintained.
Span/Pier	Single Click	Select Span/Pier – hot spot is empty space between girders and pier lines
	CTL/SHIFT-Click	Multiple select to allow multi-span cut, copy, paste, and delete.
	Double Click	On Span: Activate Span Editing dialog On Pier: Activate Pier View for selected pier.
	Right Click	Context menu
	Drag	Move span/pier to another location – animated mouse feedback. If the user attempts to drag multiple span/Piers, the "no" cursor is used and a tool tip follows the mouse that tells the user that multiple span/piers cannot be dragged. If the span/pier is dropped on the source span/pier, nothing happens. Figure 10 shows the scenarios.
	Ctl-Drag	Copy span/pier to another location – animated mouse feedback. If the user attempts to drag multiple span/piers, the "no" cursor is used and a tool tip follows the mouse that tells the user that multiple span/piers cannot be dragged. If the span/pier is dropped on the source span/pier, a copy of that span/pier is inserted into the project. Figure 10 shows the scenarios.
Span Hinge	Single Click	Select
	CTL/SHIFT-Click	Multiple Select - For Delete only
	Double Click	Activate Span Hinge Editing dialog
	Right Click	Context menu

	Drag	Move hinge to another location. The distance from the hinge to its reference pier is maintained. The orientation of the hinge is changed to match the skew angle of the new reference pier. The left/right location of the hinge, with respect to the reference pier, is defined by where the hinge is dropped. Example: a hinge that is 10ft left of pier 1 is dropped to the right of pier 3 will be positioned 10ft to the right of pier 3, parallel to pier 3. If the hinge falls in the same location as an existing hinge, the hinges are merged into one. A hinge cannot be dropped in a span that contains two hinges. Cannot drag if multiple select
	Ctl-Drag	Copy span hinge to another location. See Drag for details
Slab	Single Click	Select
	CTL/SHIFT-Click	Select – there is only one slab to select
	Double Click	Activate Slab Editing dialog
	Right Click	Context menu – Properties only option
	Drag	⊗
	Ctl-Drag	⊗

3.1.2.1.2.1 Span/Pier Drag & Drop

This section describes the scenario when a span/pier is dragged or Ctrl-dragged using the mouse. Figure 10 shows the scenario when a left pier span/pier is selected. Right pier scenario is similar.

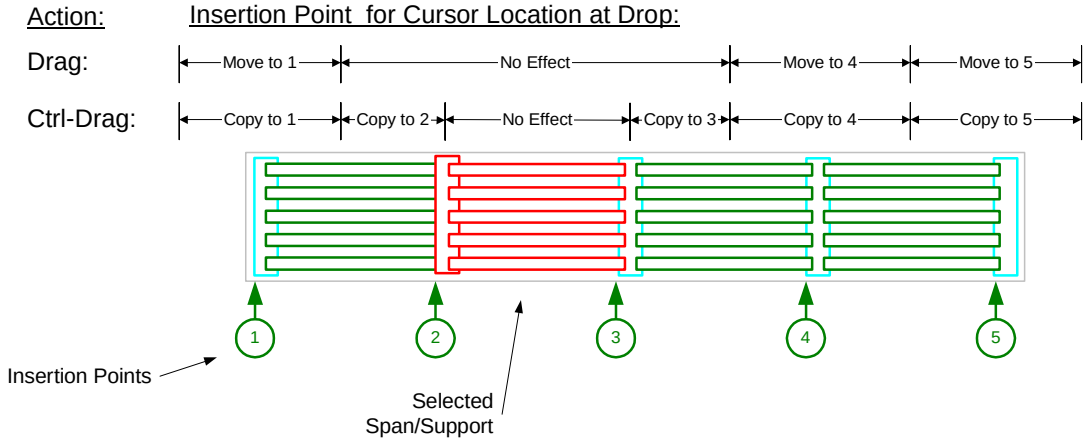


Figure 10
Drag Scenarios for Span/Pier Drag & Drop

3.1.2.1.3 COPY AND PASTE OPERATIONS

Some of the actors in the Framing Plan View also support Delete, Cut, Copy and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. In the table below, if a paste target is not listed, it means that a paste has no effect on the actor type. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled. .

Table 2 - Copy and Paste Operations for Framing Plan View

Actor	Action	Paste target	Result

Alignment	NA		
Girderline	Copy		Copy to paste buffer
	Cut		Remove girderline and copy to paste buffer - Can't remove last girderline
	Paste	Girderline	Copy properties to paste target without changing the girder spacing
		Span/ Pier	Paste a new girderline right of the right exterior girder, using the right exterior girder spacing. The new girderline is initialized with the properties of the right girderline. The width of the slab is extended to accommodate the new girderline. The girder/slab edge overhang is maintained. If the girderline is beyond the cap beam, the cap beam is extended to accommodate. The girder/cap beam overhang is maintained.
		Nothing Selected	Same as for Span/Pier
		Pier	Same as for Span/Pier.
	Paste Special	Same as Paste	Option given to mirror girderline properties about its centerline.
	Delete		Delete girderline, because girder spacing is constrained to be equal for product models, a gap is not left. - Can't delete last girderline
Span/Pier	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer – can't remove last
	Paste	Span/ Pier	Copy properties to paste target
		Girderline	Paste span/pier after the last span. Orientation of the paste source is maintained (that is, if the pier is left off the span, the span/pier is pasted to the left of the last pier, otherwise it is pasted to the right of the last pier). The new span/pier is initialized with the properties of the last span and pier.
		Pier	Place span/pier on the right side of the pier, retaining the relative locations of the span and pier in the paste buffer. The new span/pier is initialized with the properties of the adjacent span and pier.
		Nothing Selected	Same as for Pier
	Paste Special	Same as Paste	• Paste span only • Paste span only - Mirror properties • Paste pier only
	Delete		Delete (can't delete if only one span)
Span Hinge	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer

	Paste - If the span hinge is pasted on top of an existing span hinge, the two hinges are automatically merged together. Pasting is not allowed if a span contains two hinges.	Span Hinge	Replace paste target with paste source.
		Span/ Pier	Paste hinge onto span retaining the distance from reference pier and reference pier location. Example: A hinge is 50ft to the right of a pier is in the paste buffer, when applied to the paste target, it is placed 50ft to the right of the start pier. If the hinge was left of a pier, then it would have been pasted left of the end pier. If target span is too short, paste fractional
		Girderline	⊗
		Pier	Paste hinge to the pier, retaining the relative distance and orientation of the referenced pier.
	Delete		Delete

3.1.2.1.4 IN-PLACE EDITING

The Framing Plan View shows dimensions for span lengths. These dimensions may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing of the span length. For in-place editing, the station of Pier 1 is fixed. All the other piers move in response to span length changes.

3.1.2.1.5 CONTEXT EDITING

These commands will become available on the context menu and main Edit menu if an applicable item is selected.

3.1.2.1.5.1 Mirror Properties

This command will mirror the stiffness properties if a span/pier or girderline is selected. Can also mirror the entire bridge if all are selected.

3.1.2.1.6 TOOLBOX

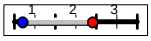
The Toolbox is a palette of icons across the top of the Framing Plan View window. The purpose of the toolbox is to provide an interactive means for editing the bridge model. Icons are dragged from the toolbox and dropped onto the model. Buttons are pressed to invoke a mode or perform an action. Editing features include addition of span/piers, span hinges, girderlines, and orienting North in the window.

The  icon is used to add new span/piers to the model. Drag this icon from the toolbox to the location where you want to add a new span/pier. The mouse cursor will indicate the orientation of the span/pier when it is dropped (pier on left or right of span). The new span and pier will take on the properties of the drop target.

The  icon is used to add hinges to a span. Span hinges can be dropped onto any span as long as structure remains stable. Once dropped, the Span Hinge dialog is displayed so the user can position the hinge exactly. The dialog will be initialized with data based on the drop location. Span hinges are only available for built-up steel girder bridges.

The  icon is used to drop new girderlines onto the model. New girderlines take on the properties and spacing of the girderline below and adjacent to the drop. If dropped to the above of the left-most girder line, the properties for the left-most girder line are used.

The  drop down button is used to orient North. The first option is a "best fit" orientation of north. QConBridge II attempts to align North such that the bridge fits best in the view. The second option allows the user to define North by two mouse hits in the view window. The third option makes North straight up on the screen.

The  tool is used to zoom in on the bridge model and to select the spans for viewing.

3.1.2.2 Elevation View

The Elevation view is a two-pane view displaying an elevation section cut for a given girderline. The upper pane shows a section cut of the product model, and the lower pane shows the Longitudinal Analysis Model for that girderline. A schematic is shown in Figure 11.

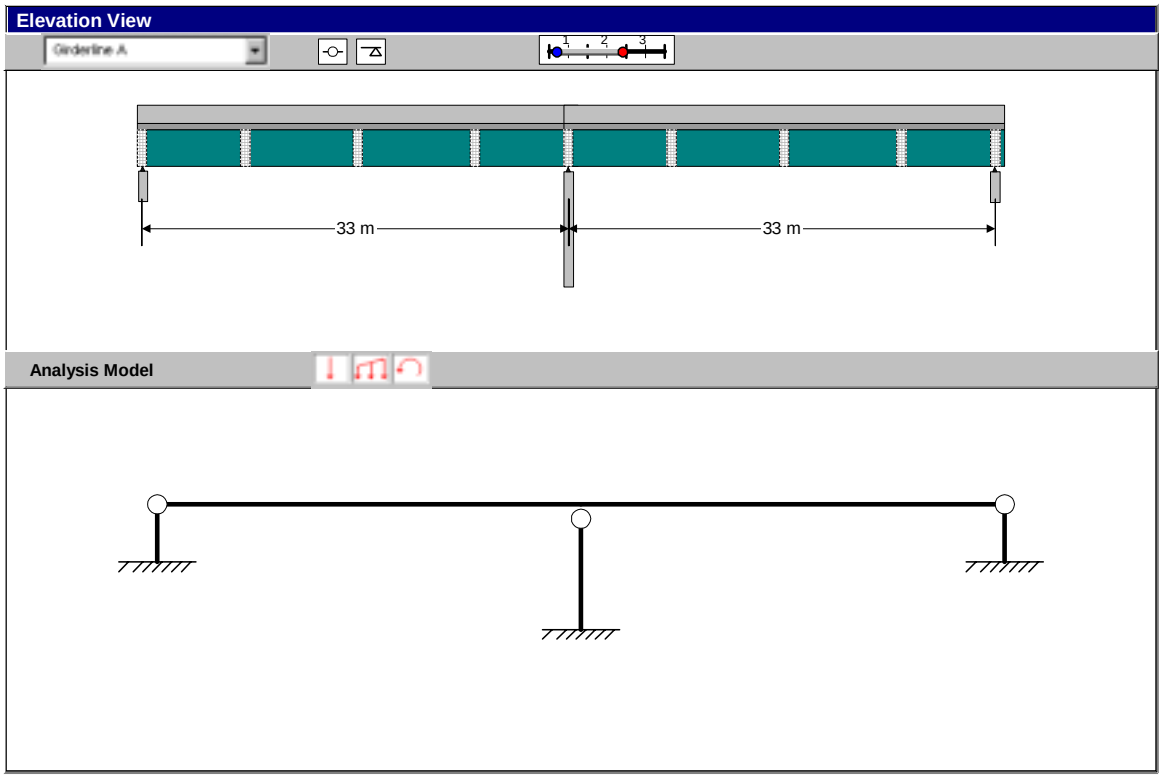


Figure 11 – Product Model Elevation View

3.1.2.2.1 KEYBOARD SELECTIONS

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys are used to select the slab, span/piers, and hinges. The selection sequence using the right arrow key goes as follows: slab → pier 1/span 1 → hinges in span 1 → span 1/pier 2 → pier 2/span 2 → … → span n/pier n+1 → slab. The order is reversed if the left arrow key is used. The selections in both panes are synchronized.

In the analysis model pane, the up arrow will jump the selection to user-defined loads. This will cause all items in the product model pane to become unselected. After the loads selection mode is enabled, the left and right arrow keys are used to change the selection. The down arrow key is used to jump into span/pier selection mode.

3.1.2.2.2 MOUSE OPERATIONS

The Elevation View displays an elevation view of the bridge cut at the selected girderline and supports mouse interaction with many of the product model interface actors as specified in the following table. Interaction with the LBAM (bottom pane) is limited to editing of user-defined loads and viewing LBAM dialogs in read-only mode. Refer to the BAM Model Editor reference for dialogs and actors used in the BAM model.

Table 3 – Mouse Actions for Elevation View (Product Pane Only)

Actor	Action	Result
Span/Pier	Single Click	Select Span/Pier
	CTL/SHIFT-Click	Multiple select
	Double Click on Pier Double Click on Span	Activate Pier Editing Dialog. See Section 3.1.4.8 below Activate Span Edit dialog. See Section 3.1.4.5.2 below
	Right Click	Context menu – Edit Pier – Edit Framing Plan
	Drag	Move span/pier to another location – animated mouse feedback - Same semantics as the Framing Plan
	Ctl-Drag	Copy span/pier to another location - Same semantics as the Framing Plan
Span Hinge	Single Click	Select
	CTL/SHIFT-Click	Multiple select (for delete)
	Double Click	Activate Span Hinge Editing dialog
	Right Click	Context menu
	Drag	Move hinge to another location. The distance from the hinge to its reference pier is maintained. The orientation of the piers is changed to match the skew angle of the new reference pier. The left/right location of hinge, with respect to the reference pier, is defined by where the hinge is dropped. Example: a hinge that is 10ft left of pier 1 is dropped to the right of pier 3 will be positioned 10ft to the right of pier 3, parallel to pier 3. If the hinge falls in the same location as an existing hinge, the hinges are merged into one. A hinge cannot be dropped in a span if it causes the structure to become unstable.
	Ctl-Drag	Copy span hinge to another location
Slab	Single Click	Select
	CTL/SHIFT-Click	Select – no multiple
	Double Click	Activate Slab Editing dialog
	Right Click	Context menu – Properties only option
	Drag	⊗
	Ctl-Drag	⊗

3.1.2.2.3 COPY AND PASTE OPERATIONS

Some of the actors in the Elevation View also support Delete, Cut, Copy and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled.

Table 4 - Copy and Paste Operations for Elevation View

Actor	Action	Paste target	Result
Span/Pier	Copy		Copy to paste buffer

Actor	Action	Paste target	Result
	Cut		Remove and copy to paste buffer. (Can't cut if only one span)
	Paste	Span/ Pier	Copy properties to paste target
		Span Hinge Slab Nothing	⊗ ⊗ Add a Span/Pier to the end of the bridge
	Paste Special	Same as Paste	• Paste span only • Paste span only - Mirror properties • Paste pier only
	Delete		Delete (can't delete if only one span)
Span Hinge	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste	Span Hinge	⊗
		Span/ Pier	Same as for Plan View. See Section 3.1.2.1.3
		Slab	⊗
	Delete		Delete

3.1.2.2.4 IN-PLACE EDITING

The Elevation View shows dimensions for span lengths. These dimensions in the product model pane may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing of the span length. For in-place editing, the stations of left-most piers are fixed.

3.1.2.2.5 CONTEXT EDITING

These commands will become available on the context menu and main Edit menu if an applicable item is selected.

3.1.2.2.5.1 Mirror Properties

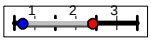
This command will mirror the stiffness properties if a span/pier or girderline is selected. Can also mirror the entire bridge if all are selected.

3.1.2.2.6 TOOLBOX

Across the top of the Elevation View window is a palette of icons (toolbox) applicable to the product model view only. The purpose of the toolbox is to provide an interactive means for editing the bridge model. Editing features includes addition of span/piers and span hinges.
The combo-box is used to select the girder line that is displayed in the Elevation View.

The  icon is used to add new span/piers to the model. Drag this icon from the toolbox to the location where you want to add a new span/pier. The mouse cursor will indicate the orientation of the span/pier when it is dropped (pier on left or right of span). The new span and pier will take on the properties of the drop target. This icon cannot be dropped on the analysis model pane.

The  icon is used to add hinges to a span. Span hinges cannot be dropped if it causes the structure to become unstable. Once dropped, the Span Hinge dialog is displayed so the user can position the hinge exactly. The dialog will be initialized with data based on the drop location. This icon cannot be dropped on the analysis model pane.

The 

tool is used to zoom in on the bridge model and to select the spans for viewing. The zoom tool affects both the product model and the analysis model panes.
The toolbox on the Analysis Model pane is used for applying loads to the analysis model. Details are provided in Section 3.2.1.2.2.5.

3.1.2.3 Pier View

The Pier View is a two-pane view showing the product model of the pier in the left pane and the analysis model of the pier in the right. The editing operations available on the product model are presented in this section.

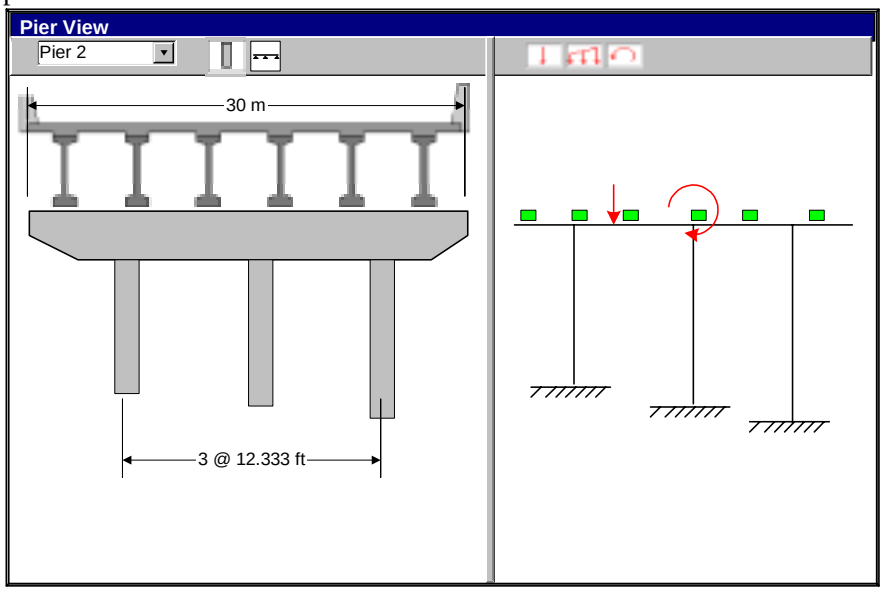


Figure 12 - Pier View

3.1.2.3.1 KEYBOARD SELECTIONS

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys are used to select the cap beam and columns. The selection sequence using the right arrow key goes as follows: cap beam → column 1 → column 2 → … → column n → slab → girder line 1 → … → girder line m → cap beam. The order is reversed if the left arrow key is used. The selections in both panes are synchronized when cap beam, column, or girder lines (bearings are selected in the TBAM pane) are selected.

In the analysis model pane, the up arrow will jump the selection to user-defined loads. This will cause all items in the product model pane to become unselected. After the loads selection mode is enabled, the left and right arrow keys are used to change the selection. The down arrow key is used to jump into member selection mode.

3.1.2.3.2 MOUSE OPERATIONS

The Pier View displays an elevation view of the selected pier and supports mouse interaction with many of the product model interface actors as specified in the following table. Interaction with the TBAM (right pane) is limited to editing of user-defined loads and viewing TBAM dialogs in read-only mode. Refer to the TBAM Model Editor reference for dialogs and actors used in the TBAM model.

Table 5 – Mouse Actions for Framing Plan View (Product Pane Only)

Actor	Action	Result
Pier Cap	Single Click	Select Pier Cap
	CTL/SHIFT-Click	Select cap – only one to select
	Double Click	Activate Cap editing dialog
	Right Click	Context menu
	Drag	⊗
	Ctl-Drag	⊗

Girderline	Single Click	Select Girderline
	CTL/SHIFT-Click	Multiple Select
	Double Click	Activate Elevation View for girderline analysis model
	Right Click	Context menu
	Drag	Same behavior as in Plan View. See Section 3.1.2.1.2
	Ctl-Drag	Same behavior as in Plan View. See Section 3.1.2.1.2
Column	Single Click	Select
	CTL/SHIFT-Click	Select multiple
	Double Click	Activate Column Editing dialog
	Right Click	Context menu
	Drag	Moves column to the location of the column nearest drop location. Columns in between the source and target column are shifted toward the source to fill the void. Current spacing is retained.
	Ctl-Drag	Copies source column into location between columns at drop location. Spacing on either side of new column is same as spacing at drop location. If column is dropped outside of exterior column, it becomes an exterior column with the same spacing as the previous exterior column. Widens the cap beams to accommodate new column spacing. Overhang dimensions with respect to the exterior columns are maintained.
Slab/ Appurtenances	Single Click	Select
	CTL/SHIFT-Click	Select – only one to select
	Double Click	Activate Slab Editing dialog
	Right Click	Context menu – Properties only option
	Drag	⊗
	Ctl-Drag	⊗

3.1.2.3.3 COPY AND PASTE OPERATIONS

Some of the actors in the Pier View also support Delete, Cut, Copy, and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled.

Table 6 - Copy and Paste Operations for Framing Plan View

Actor	Action	Paste target	Result
Pier Cap	Copy		Copy cap to paste buffer
	Cut		⊗
	Paste	Pier Cap	Copy properties to paste target
	Delete		⊗
Girderline	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer (can't remove if at minimum # girders)
	Paste	Girderline Nothing	Copy properties to paste target Puts a girderline to the right of the right-most girderline

	Paste Special	Girderline	Give option to paste mirror image of properties
	Delete		Delete (can't delete if at minimum # girders)
Column	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer (can't remove if it is the only column)
	Paste	Column Nothing	Paste properties to paste target Puts a column to the right of the right-most column
		Pier Cap	Paste a new column on the right hand side of the pier using the right-most column spacing. If the new column is beyond the end of the cap beam, the cap beam is extended, maintaining the current column/cap overhang.
	Paste Special	Same as Paste	Provide option to mirror properties
	Delete		Delete (can't delete if it is the only column). Column spacing is adjusted to maintain cap/column overhangs.

3.1.2.3.4 IN-PLACE EDITING

The Pier View shows dimensions for cap length, number of columns and column spacing. These dimensions may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing. The cap width is adjusted for new column spacing.

3.1.2.3.5 CONTEXT EDITING

These commands will become available on the context menu and main Edit menu if an applicable item is selected.

3.1.2.3.5.1 Mirror Properties

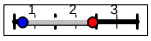
This command will mirror the stiffness properties if a cap beam or a column is selected. Can also mirror the entire pier if all are selected.

3.1.2.3.6 TOOLBOX

Across the top of the Pier View window is a palette (toolbox) for icons and widgets applicable to the pier view only. The purpose of the toolbox is to provide an interactive means for editing the pier product and analysis models. The combo box allows selection of the current pier. Editing features include addition of columns to the product model and loads to the analysis model.

The  icon is used to drop a new column on the pier. New columns are given the properties and spacing of the left adjacent column to the drop point. The cap beam is extended to accommodate the new column configuration. The column/cap overhang is maintained.

The  icon is used to drop new girderlines onto the superstructure cross section. New girderlines take on the properties and spacing of the right adjacent girderline. If dropped to the left of the left-most girder line, the properties for the left-most girder line are used.

The 

tool is used to zoom in on the bridge model and to select the spans for viewing. The zoom tool effects both the product model and the analysis model panes.

3.1.2.4 Viewing Tools

3.1.2.4.1 DYNAMIC CURSOR

In QConBridge II, the mouse cursor changes dynamically depending on the program state and what the mouse is hovering over.

Table 7 - Dynamic Mouse Icons

Mouse Icon	Mouse is Hovering over:
	Normal – over white space
	Loaded cursor – cursor is over a selected item
	In drag mode – cannot drop
	Alignment
N	North arrow
	Girderline
 or 	Span/pier – direction of icon depends on cursor location
	Span Hinge
	Slab/appurtenances
	Load
	Cap Beam
	Columns

3.1.3 Bridge Contractor

The Bridge Contractor is a wizard used to edit bridge and create QConBridge II projects. The Bridge Contractor is reentrant for all Product Model project types.

3.1.3.1 Step 1

Step 1 is seen for all types of projects. It allows you to choose the project type and system-level properties for a project.

Figure 13 - Bridge Contractor Step 1

Project type can't be changed on the second and subsequent passes through the wizard. Once you pick a project type, you are stuck with it.
To the left of the Next button, we might want to have a "Goto" button that drops down a menu of Wizard Steps. This will allow the user to quickly jump to a particular step.

3.1.3.2 Step 2 – Analysis Settings

In this step we describe how the structure is to be analyzed. This step is similar to Figure 20 - Analysis Settings.

3.1.3.3 Step 3 – Alignment

In step 3 we define the alignment. This step is similar to Figure 21 - Alignment and Roadway.

3.1.3.4 Step 4 – Pier Layout

This step in the Bridge Contractor is used to define the Pier layout and to edit piers. Default piers are 2D zero-height with simple support connections.

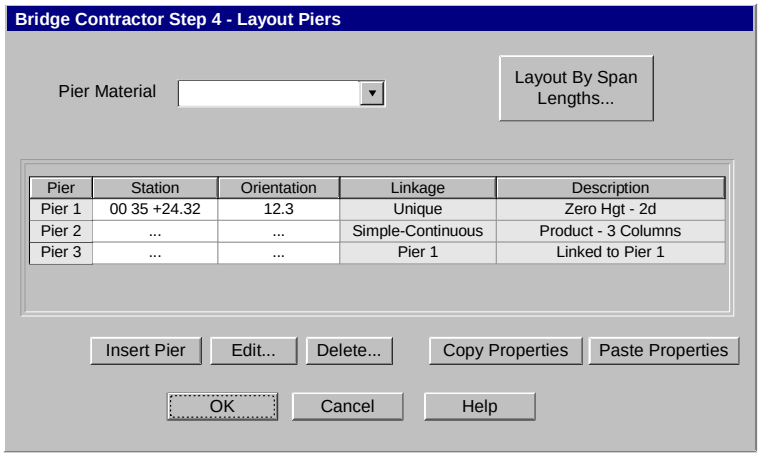


Figure 14 - Bridge Contractor Step 4

When the Insert Pier button is pressed, the Edit Pier Dialog is displayed. You must then select the pier location and pier properties. When a pier is inserted between two existing piers, the span definition is fit between the left pier and the new pier[1]. The new span created between the new pier and the right pier is a copy of the span that is left of the new pier. When a pier is inserted before or after ends of the bridge, the superstructure is extended using the same girders and spacing as the previous outer span.
When the Delete button is pressed, a dialog will come up asking which span will be deleted with the pier.
Piers must be in order. If piers are not in order, all piers that are not will be shown in red.

3.1.3.5 Step 5 – Superstructure Layout

This step is used to define the girderlines and girder properties for the superstructure. The top portion, as shown in Figure 15 below, is the same for all bridge types. The bottom portion of the dialog varies each for bridge type as shown in the paragraphs in this section.

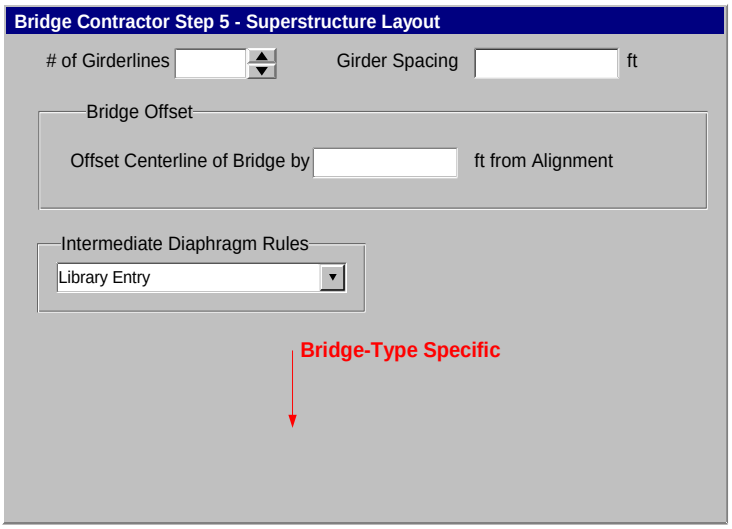


Figure 15 - Bridge Contractor Step 5 – Generic Superstructure Layout

3.1.3.5.1 PRECAST BRIDGES

Precast bridges only add one option: the girder type

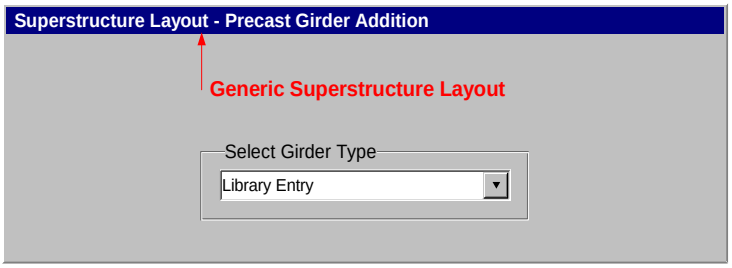


Figure 16 - Bridge Contractor Step 5 - Precast Girders

3.1.3.5.2 ROLLED STEEL BRIDGES

Rolled steel bridges require the addition of girder type information and top and bottom flange cover plate schedules. Plate Schedules can be unique (defined explicitly for that girder) or linked (takes on values of another, unique girder).

[1] Any of the span definition that does not physically fit between the new pier arrange is retained in the span definition, but is marked as ignored.

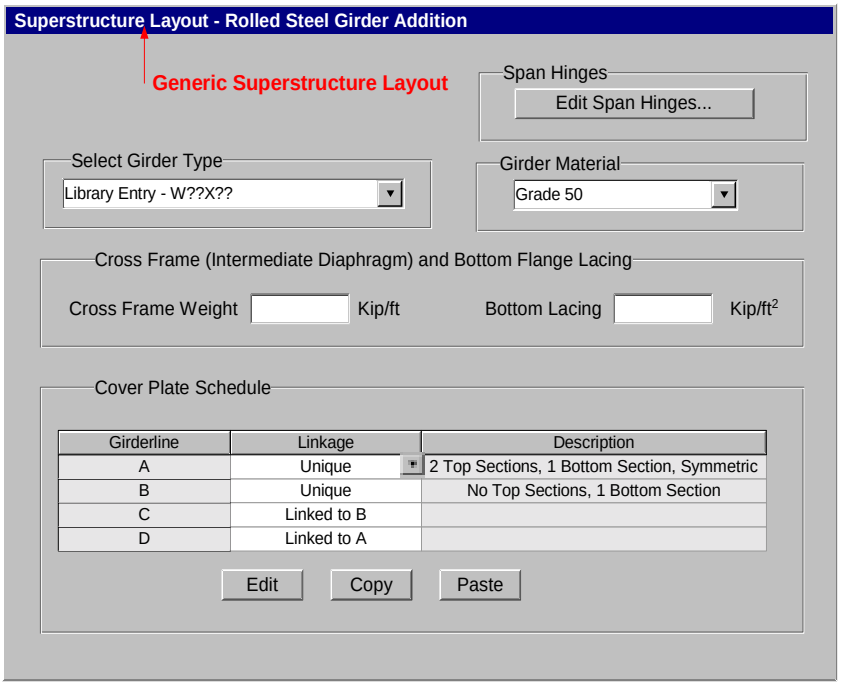


Figure 17 - Bridge Contractor Step 5 - Rolled Steel Girders

Each girderline's cover plate schedules may be edited separately or linked together. The editing dialog is shown in Section 3.1.4.6.1 below

3.1.3.5.3 BUILT-UP PLATE GIRDER BRIDGES

Built-up plate girders are fully defined by their total depth, material type, and plate schedules.

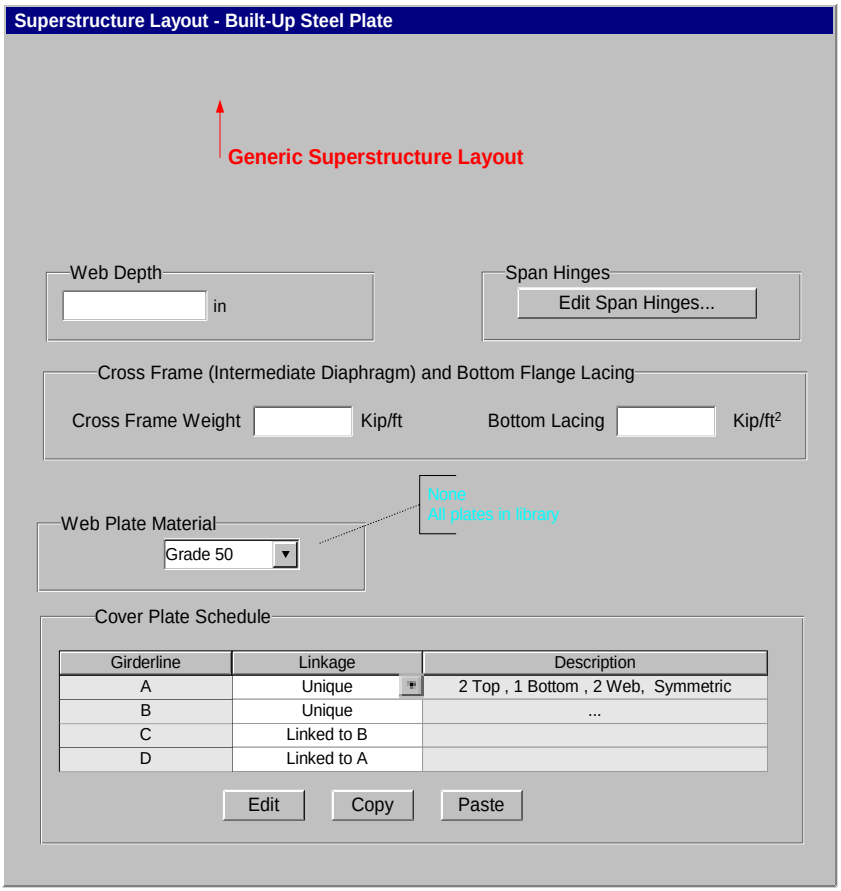


Figure 18 - Bridge Contractor Step 5 - Built-up Plate Girders

Each girderline's cover plate schedules may be edited separately or linked together. The editing dialog is shown in Section 3.1.4.6.2 below.

3.1.3.6 Step 6 – Slab

This step is used to define the slab and appurtenances. The step is similar to Figure 32.

3.1.4 Detailed Editing for Actors

This section describes the primary dialogs for editing each of the major UI actors in QConBridge II. Access to these dialogs is either through context menus from the main views, or by menu selections.

3.1.4.1 Bridge

The Bridge is the entire product model. There are some actions that can be carried out on the entire bridge as follows:

3.1.4.1.1 MOVE BRIDGE

This command translates the entire bridge along the alignment a given distance, or moves the bridge such that the location of a selected pier is at the desired station.

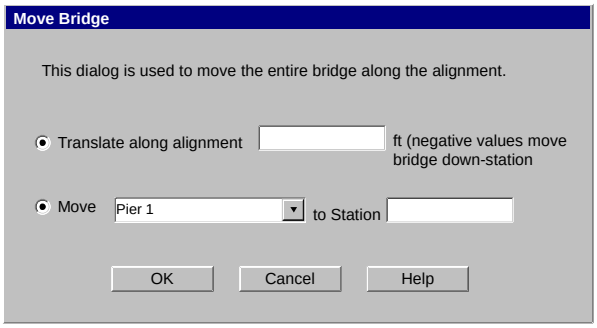


Figure 19 - Move Bridge

3.1.4.1.2 MIRROR

If the bridge is selected, the Mirror command will effectively flip the entire bridge around and keep the current stationing.

3.1.4.2 Settings

Settings affect the way the structure is analyzed.

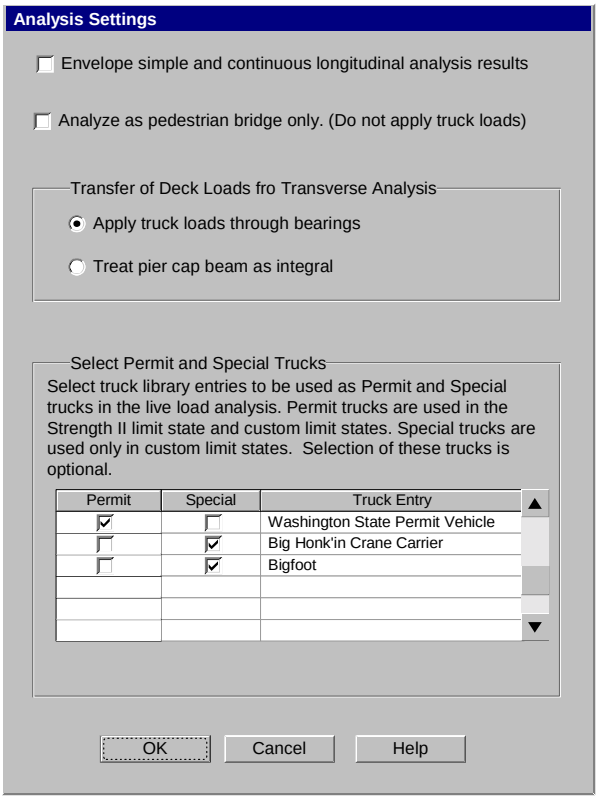


Figure 20 - Analysis Settings

3.1.4.3 Alignment/Roadway

The Alignment defines the horizontal and vertical orientation of the bridge roadway.

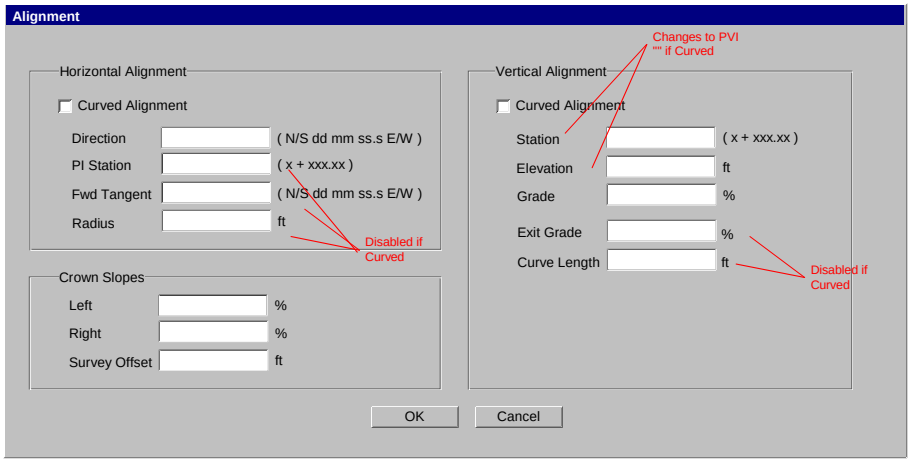


Figure 21 - Alignment and Roadway

3.1.4.4 Framing Plan Editor

The Framing Plan Editor is a dialog-based interface element that provides a compact and complete way of editing the bridge product model's pier layout and superstructure layout. The Framing Plan Editor is a multi-tab dialog that consists of similar dialogs given in Bridge Contractor Steps 4-5 above.

3.1.4.5 Span/Piers

The primary purpose for Span/pier actors in QConBridge II is to provide a clean way to add, move, and delete spans in from the Framing Plane and Elevation Views. Dialog editing for Span/Piers is restricted to editing either the length of the selected span, or editing the selected pier.

3.1.4.5.1 SPAN/PIER SELECTION

Just like in QConBridge I, span/piers can be selected with either the mouse or the keyboard. Span/Pier selection is supported in the Framing Plan and Elevation Views. In the Framing Plan View, span/piers are selected by clicking in the hollow space between girderlines. In Elevation view, span/piers are selected by clicking on the girderline or any pier. The cursor will dynamically display the ⊐ icon whenever the mouse is over a span/pier.
QConBridge II will eliminate the idea of Selection Mode. To enforce the requirement that a valid analysis model must exist after every edit, spans and piers are operated on in pairs. In QconBridge I, the selection mode defined which pier was selected when a span was selected (See page 3-3 of the QConBridge Reference Manual for a complete discussion of Selection Mode). QConBridge II will use a more natural means of selecting span/pier pairs. When the user clicks on a span to select it, the pier nearest the click point is selected. To change the selected pier, the user can either click on the desired pier or click on the other half of the span. If the user clicks on a pier that is not adjacent to the selected span, the new selection will have the same span/pier selection configuration as the previous selection (there may be exceptions to this if the first or last pier is selected). The dynamic cursor displays either ⊐ or ⊏ to confirm the selection orientation.

3.1.4.5.2 EDIT SPAN

This dialog is shown when Edit Span is selected from the context menu if a Span/Pier is selected. Note that the Girder type is only shown for Precast and rolled steel product models.

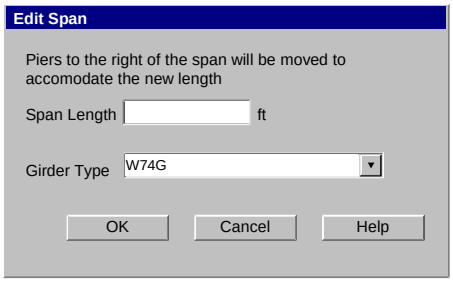


Figure 22 - Edit Span

3.1.4.5.3 ADD SPAN/PIER

New span/piers can be added to the model in by using the  icon in the toolbox or by selecting Bridge|Add Span/Pier from the main menu. Both operations work in a similar manner:
The toolbox icon can only be dropped onto span/piers in the model. When it is dropped onto a span/pier, it makes a copy of the target span/pier and places it to the right of the target. All piers to the right of the target are moved to the right.
The Result of the menu operation is dependent on the current selection. If a span/pier is selected, the operation makes a copy of the selected pair and inserts it to the right of the selection. Again, all piers to the right of the selection are adjusted to the right. If no span/supports are selected, the Add Span/Pier command makes a copy of the rightmost span/pier and pastes it to the right end of the bridge.

3.1.4.6 Girderlines

The primary purpose of girderline actors is to allow gui interaction to add/delete/move girderlines, and to provide schedule-based editing for built-up plate girders and for rolled steel top and bottom plates.

3.1.4.6.1 SCHEDULE-BASED EDITING FOR ROLLED STEEL FLANGE COVER PLATES

This dialog allows addition of top and bottom cover plates to rolled steel girders. . Individual plates are edited using the dialog in Figure 27.

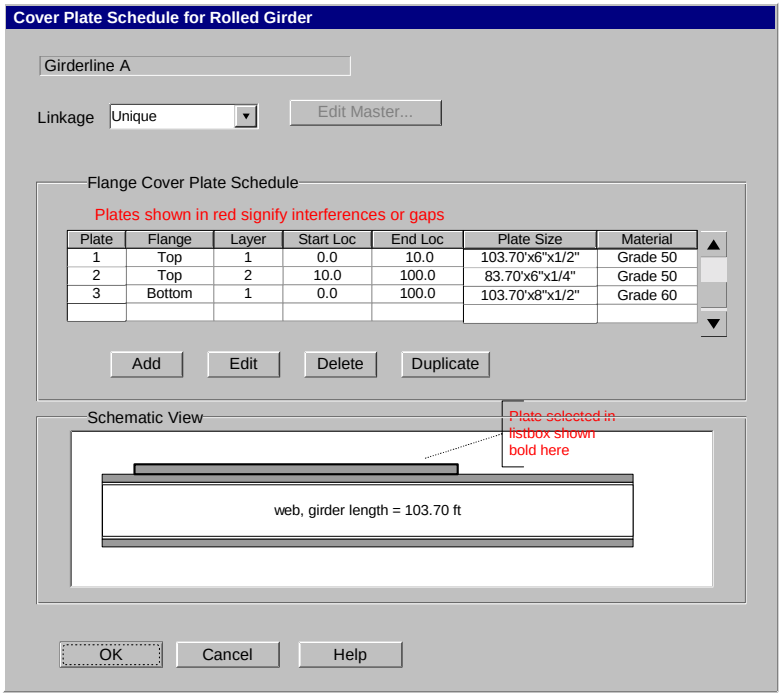


Figure 23 - Plate Schedule for Rolled Steel Girders

3.1.4.6.2 SCHEDULE-BASED EDITING FOR BUILT-UP STEEL GIRDERS

This three-tab dialog describes plate girders for a given girderline. Measurements can be taken either along the entire girderline or on a span-by-span basis. Tab 1 is used to define the linkage to another girderline, and the web plates. Note that the web and flange plates must extend along the entire girderline.

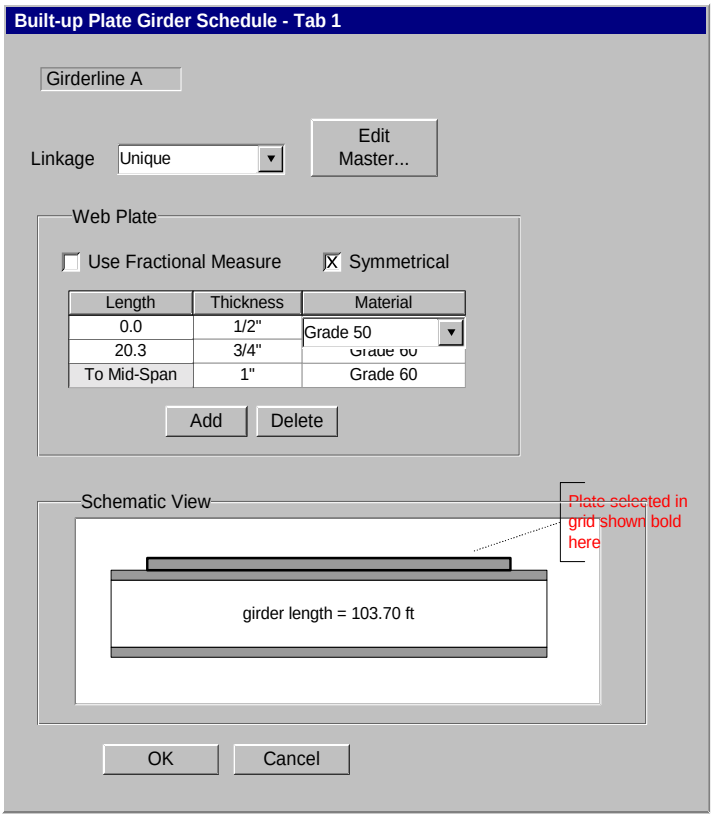


Figure 24 - Built-up Steel Girder Web Plate Schedule – Tab 1

Tab 2 describes the top and bottom flange plates for a built-up plate girder.

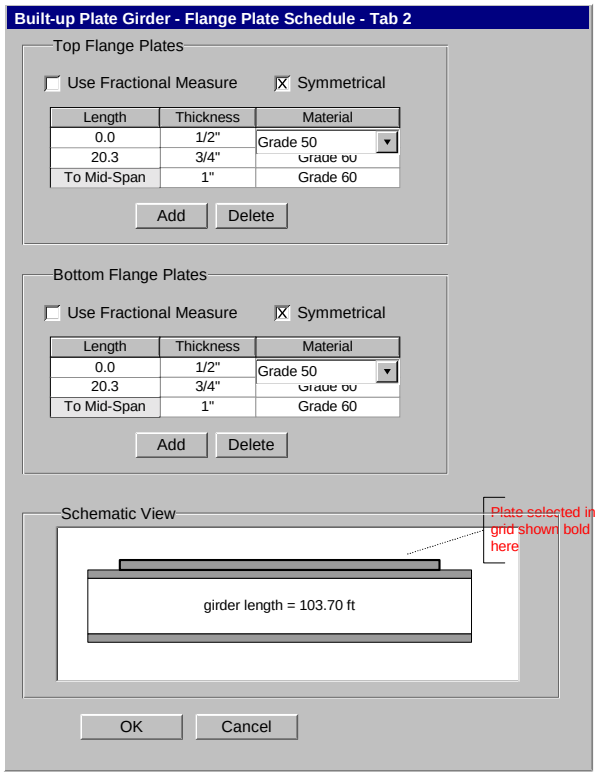


Figure 25 – Built-up Steel Girder Flange Plates - Tab 2

Tab 3 is used to edit the top and bottom flange cover plates for a built-up steel girder. Individual plates are edited using the dialog in Figure 27.

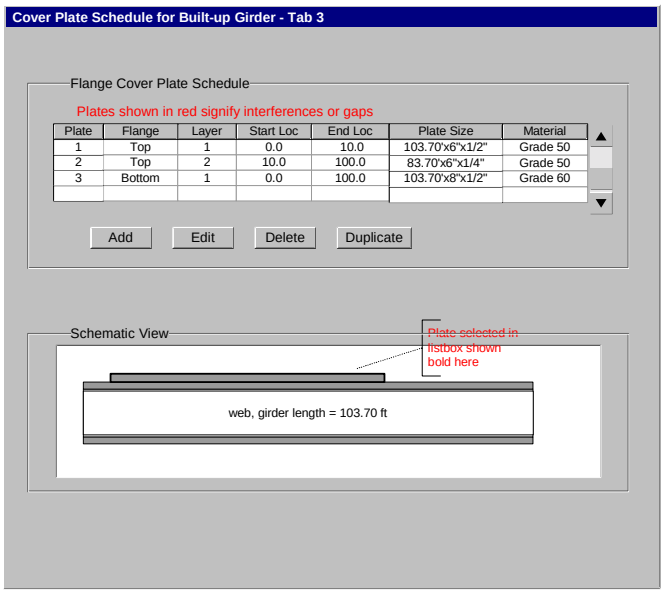


Figure 26 – Built-up Steel Girder Cover Plates - Tab 3

3.1.4.6.3 COVER PLATE EDITING DIALOG

This dialog is used to edit individual cover plates for rolled and built-up product models.

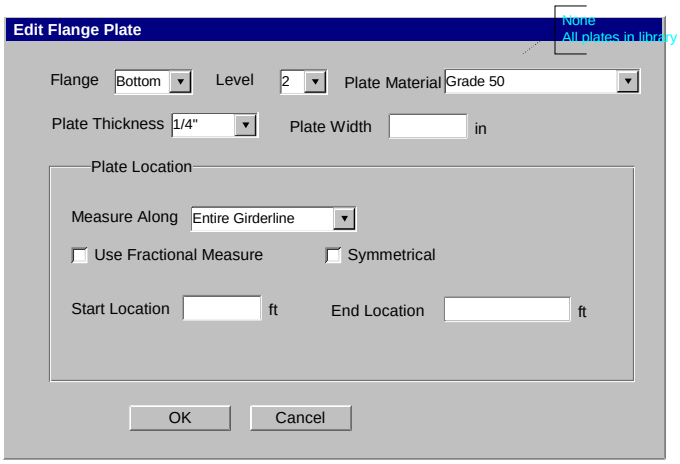


Figure 27 - Edit Cover Plate

3.1.4.6.4 General Notes About Schedule-Based Editing

Plate Schedules for steel involve the description of plates, which make up the web or flanges of an I-beam. Web plates are described in segments along the length of the beam using a From-To format. The From-To values can either be specified from the beginning to end of the given girderline, or from the centerline of the piers at the beginning and end of a given span. Flange plates are described as discrete plates that are located along the length of the girderline or span.

3.1.4.6.4.1 Exact or Fractional Locations

Lengths can be defined using exact distances (in the current units), or by describing decimal fractional values.

3.1.4.6.4.2 Dealing With Problems

Since flange plate schedules are described by placing plates along a girderline, two problems can come up: 1) Interferences where two or more plates can share the same space and, 2) Gaps where a plate at an elevated level is “left hanging”.
The problems are equally problematic and lead to the analysis of a structure that does not reflect reality. When either of these conditions happens, the affected flange plate(s) shall be shown in red in the list box and in the schematic view. Also, a Level 3 status item that describes the will be logged to the Status Center (see Section 7 below).

3.1.4.6.4.3 Resolution When a Span Length Is Changed

Redefinition of span lengths can cause problems to plate schedules. This happens when lengths are shortened and plates no longer fit in the increment for which they were defined. In this case, the segment is simply cut off of the end (or middle if a symmetrical girder). When this happens, the status center is used to relay the message.

3.1.4.6.4.4 Resolution of Linked and Pasted Plate Schedules.

A problem occurs if an explicit splice location from a longer girder is pasted or linked to a shorter girder such that the splice is located beyond the end of the shorter girder. In this case, the segment is simply cut off of the end (or middle if a symmetrical girder). The data isn't actually lost, it just isn't used.

In the case where the master girder is shorter than the linked girder, the last plate is lengthened to the end of the linked girder. If the girder is symmetric, the mid section is lengthened.

3.1.4.7 Span Hinges

This dialog is used to add, delete and edit span hinge properties.

Note that span hinges are not available for Precast I and Rolled Steel bridges.

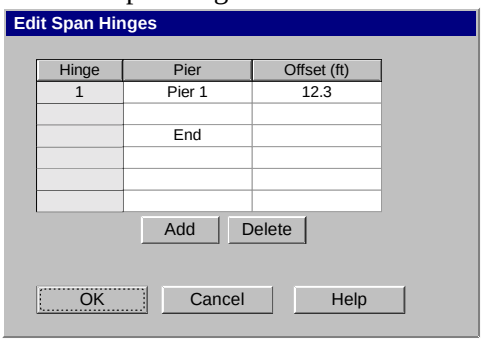


Figure 28 - Edit Span Hinges

3.1.4.8 Piers

The Piers dialog is a two-tab dialog used to describe piers. Piers are described with progressively more detail with each iteration of an engineer's design. Descriptions can start with a simple span/pier and evolve to a full product model description.

3.1.4.8.1 TAB 1 - PIER LOCATION AND CAP

Tab 1 allows editing of the pier location and connection type. It also allows the user to choose whether to make the pier just a simple support or 3D, in which case the cap beam and columns must be described.

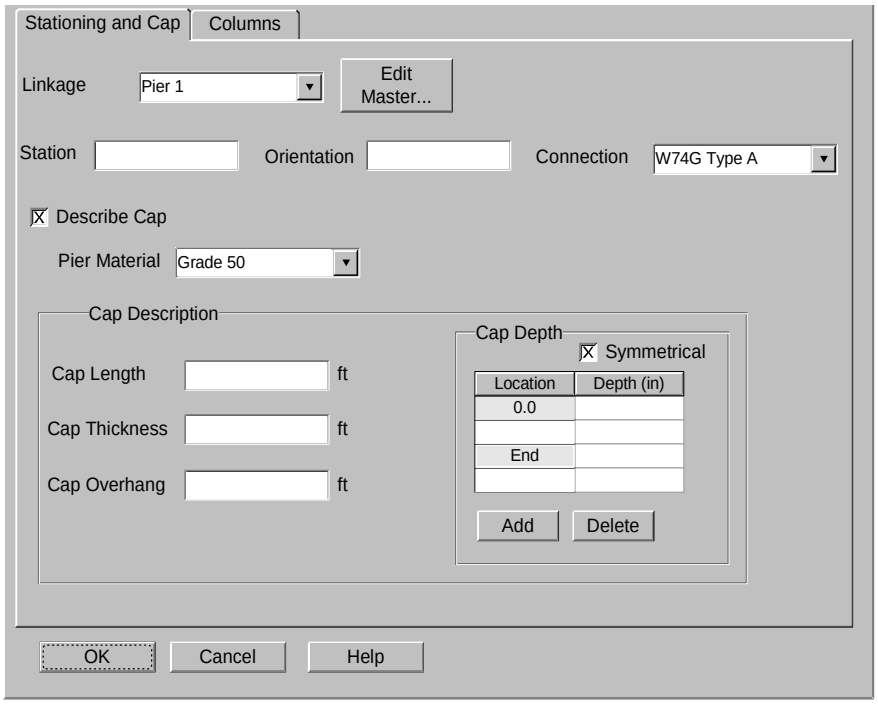


Figure 29 - Pier Definition and Cap Tab

3.1.4.8.2 TAB 2 – COLUMNS

Tab 2 is used to describe the pier columns. This tab changes appearance depending whether or not a cap is defined.

If a cap is described, columns are spaced evenly and are symmetric about the cap beam. Columns may be zero-length (Simple Support), or may have a round or rectangular cross section. All columns are of the same type and cross section dimensions. Column sections may taper linearly along the column length.
Figure 30 shows the Columns tab when a cap is fully defined.

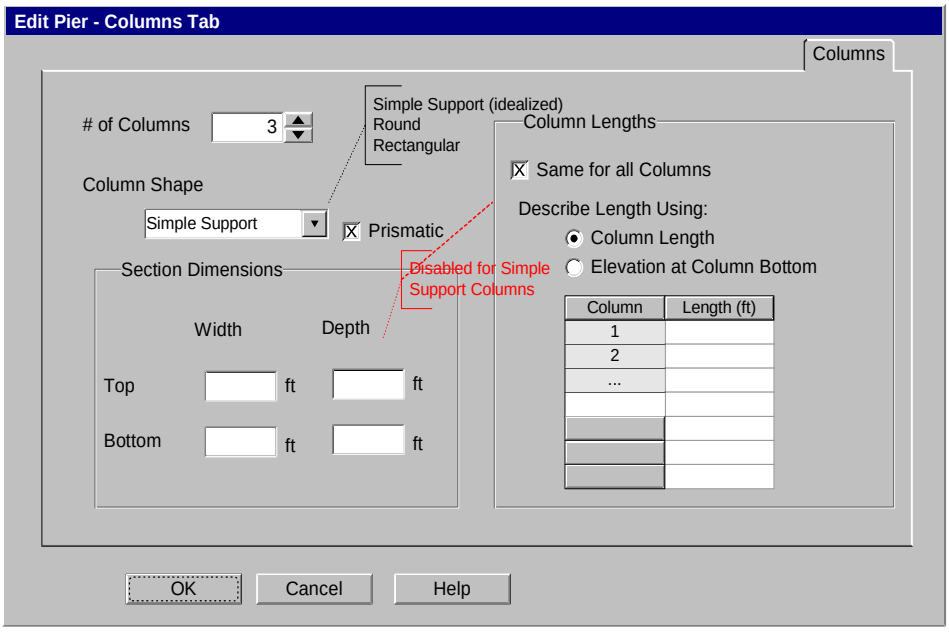


Figure 30 - Pier Columns Tab When a Cap is Defined

If a cap is not described, columns may be zero-length, or may be described as a single, fixed length column whose stiffness properties for the longitudinal analysis are divided evenly over all girderlines framing into the pier. Figure 31 shows the Columns Tab for this case.

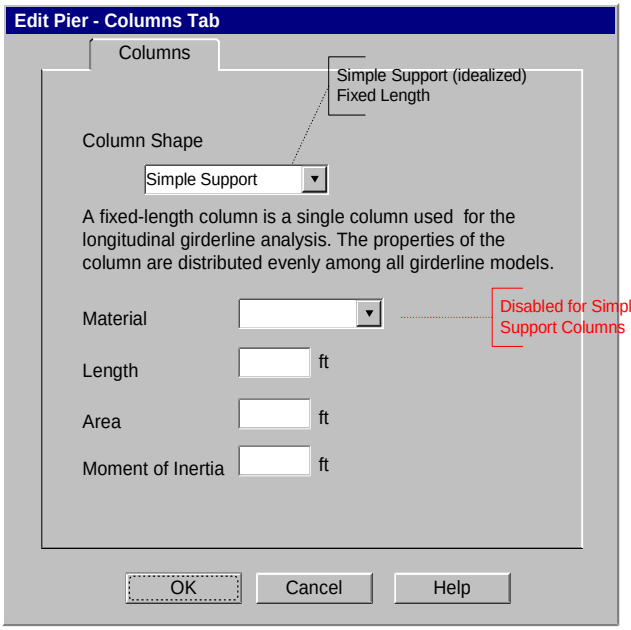


Figure 31 - Pier Columns Tab When No Cap is Defined

3.1.4.9 Slab/Appurtenances

The Slab and Appurtenances dialog defines the slab, the width of the deck, and sidewalks and barriers.

3.1.4.9.1 TAB 1 – SLAB

Dimensions for slab – Note that picture must be redone to reflect correct dimension descriptions

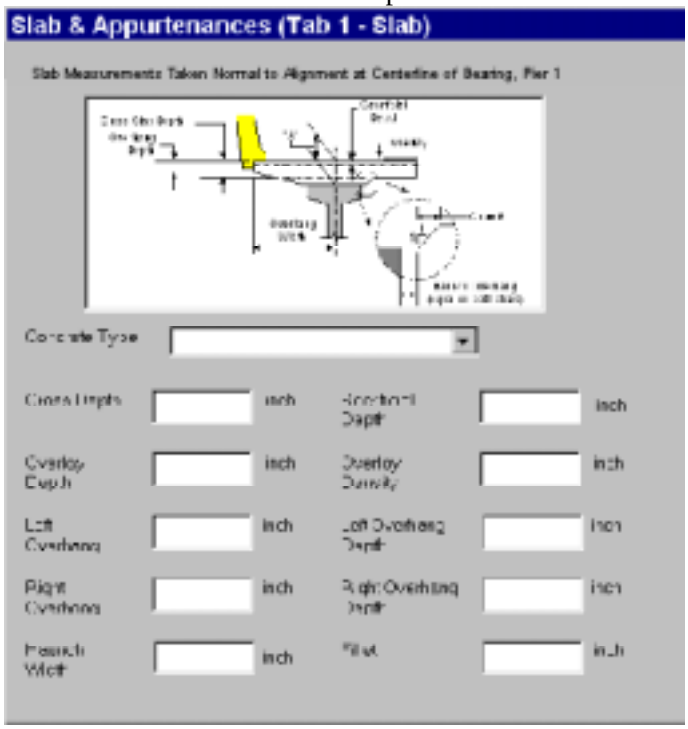


Figure 32 - Slab Tab 1

3.1.4.9.2 TAB 2 – APPURTENANCES

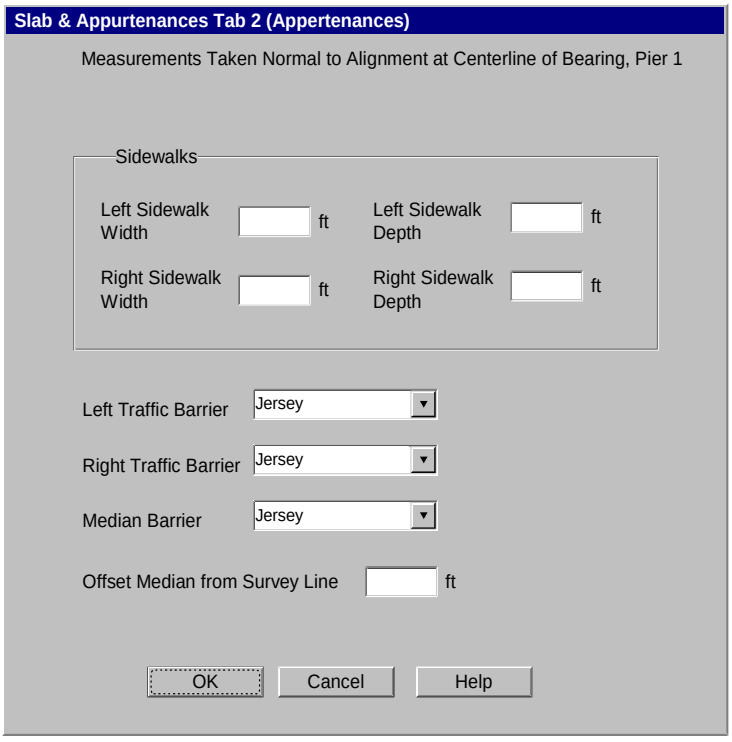


Figure 33 - Slab Tab 2

3.2 Editing of Analysis Models

QConBridge II has two types of analysis models that can be edited: Bridge Analysis Models (BAM's), and Transverse Bridge Analysis Models (TBAM's). For BAM projects, the Bridge Analysis Model Editor manages models of the bridge for both longitudinal analysis and transverse analysis. It ties the results and geometries of girderline analysis models and pier analysis models together for the user. TBAM's are integrated into BAM's, or can be a separate project type used solely to determine transverse forces on pier structures.

3.2.1 Bridge Analysis Model Editing

This section describes the various elements of the Analysis Model Editor. The Analysis Model Editor is used to display and edit the bridge analysis model. When in Product Model Mode, the analysis model(s) are automatically constructed by the software hence, the user is limited to viewing and adding externally-applied loads. When in Analysis Model Mode, the user is free to create and manipulate all aspects of analysis models.

3.2.1.1 Analysis Model Editor Actors

The Analysis Model Editor deals with the following UI Actors:

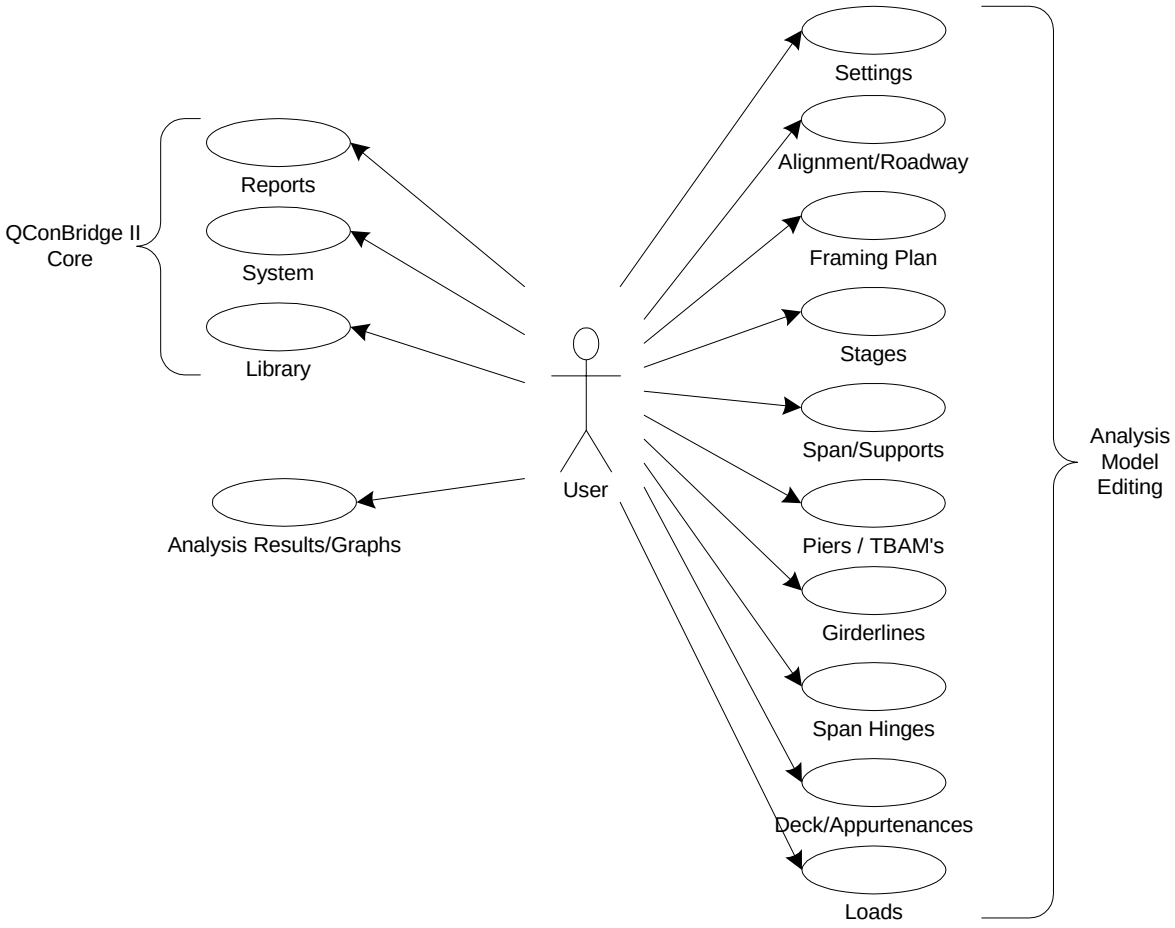


Figure 34 - Analysis Model Editor UI Actors

3.2.1.2 Analysis Model Editing Views

This section discusses the main views for analysis model editing and the primary interactions in these views.

3.2.1.2.1 FRAMING PLAN VIEW

The Framing Plan View is a plan view that displays the alignment, pier layout, overall superstructure layout and is the focal point of an analysis model-based project. Figure 35 is a schematic of the Framing Plan View.

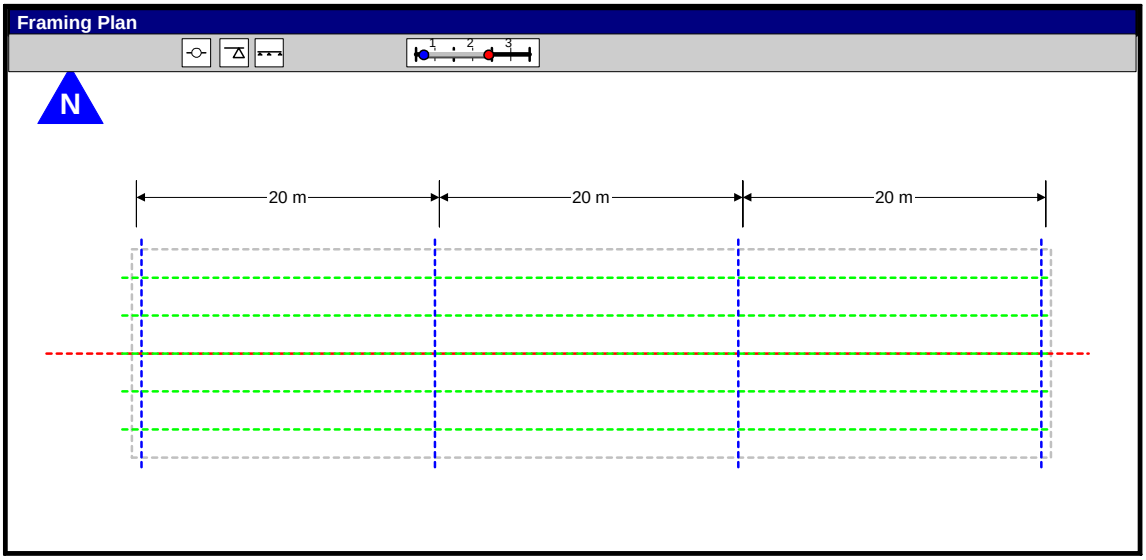


Figure 35 - Analysis Model Framing Plan View

3.2.1.2.1.1 Keyboard Selection

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys select the piers, spans, hinges, and pier/span pairs. The selection sequence using the right arrow key goes as follows, pier 1/span 1 → hinges in span 1 → span 1/ pier 2 → pier 2/span 2 → … → span n/pier n+1. The order is reversed if the left arrow key is used.
Pressing the CTL key makes multiple selections, but the selection sequence changes so that only like items are selected. For example, pier 2/Span 1 is selected and the CTL key and right arrow key are pressed, pier 3/Span 3, pier 4/Span 4, etc, are then selected. The next right arrow key press selects the next item in the sequence after the last pier/span selected.
Girder lines are selected by using the up and down arrow keys. The up arrow key moves the selection to the left and the down arrow key moves the selection to the right. Holding down the CTL key causes multiple girder lines to be selected.

3.2.1.2.1.2 Mouse Operations

The Framing Plan View displays a plan view of the bridge and supports mouse interaction with many of the product model interface actors as specified in the following table.

Table 8 – Mouse Actions for Framing Plan View

Actor	Action	Result
Alignment	Single Click	Select Alignment
	CTL/SHIFT-Click	Select – only one alignment to select
	Double Click	Activate alignment editing dialog
	Right Click	Context menu – Properties only active selection
	Drag	⊗
	Ctl-Drag	⊗
Girderline	Single Click	Select Girderline
	CTL/SHIFT-Click	Select multiple girder lines
	Double Click	Activate Girderline Elevation View for selected pier
	Right Click	Context menu

Actor	Action	Result
	Drag	Same as for Product Model View. See Section 3.1.2.1.2 above.
	Ctl-Drag	Same as for Product Model View. See Section 3.1.2.1.2 above.
Span/Support	Single Click	Select Span/Support – hot spot is empty space between girders
	CTL/SHIFT-Click	Multiple select to allow multi-span cut, copy, paste, and delete.
	Double Click	If on Span: Activate Span Editing dialog If on Pier: Activate Pier Editing dialog.
	Right Click	Context menu: Edit Span | Edit Pier
	Drag	Same as for Product Model View. See Section 3.1.2.1.2 above.
	Ctl-Drag	Same as for Product Model View. See Section 3.1.2.1.2 above.
Span Hinge	Single Click	Select
	CTL/SHIFT-Click	Select – no multiple
	Double Click	Activate Span Hinge Editing dialog
	Right Click	Context menu
	Drag	Same as for Product Model View. See Section 3.1.2.1.2 above.
	Ctl-Drag	Same as for Product Model View. See Section 3.1.2.1.2 above.

3.2.1.2.1.3 Copy and Paste Operations

Some of the actors in the Framing Plan View also support Delete, Cut, Copy and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. In the table below, if a paste target is not listed, it means that a paste has no effect on the actor type. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled.

Table 9 - Copy and Paste Operations for Framing Plan View

Actor	Action	Paste target	Result
Alignment	NA		
Girderline	Copy		Copy to paste buffer
	Cut		Remove girderline and copy to paste buffer - can't remove last girderline
	Paste	Girderline	Copy properties to paste target without changing the girder spacing

Actor	Action	Paste target	Result
		Span/ Support	Paste a new girderline right of the right exterior girder, using the right exterior girder spacing. The new girderline is initialized with the properties of the pasted girderline. The width of the slab is extended to accommodate the new girderline. The girder/slab edge overhang is maintained. If the girderline is beyond the cap beam, the cap beam is extended to accommodate. The girder/cap beam overhang is maintained.
		Nothing	Pastes new girderline into model to the right of the right-most girderline
	Paste Special	Same as Paste	Give option to mirror properties
	Delete		Delete girderline because girder spacing is constrained to be equal for product models, a gap is not left. Can't delete last girderline
Span/Support	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste	Span/ Support	Copy properties to paste target
		Girderline	Paste span/support after the last span. Span support orientation of the paste source is maintained (that is, if the support is left of the span, the span/support is pasted to the left of the last pier, otherwise it is pasted to the right of the last pier). The new span/support is initialized with the properties of the pasted span and pier.
		Nothing	Pastes a span/support after the last span.
	Paste Special	Same as Paste	• Paste span only • Paste mirrored span • Paste pier only
	Delete		Delete (can't delete if only one span)
Span Hinge	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste - If the span hinge is pasted on top of an existing span hinge, the two hinges are automatically merged together. Pasting is not	Span Hinge	Paste distance from reference pier to paste target
		Span/ Support	Paste hinge onto span retaining the distance from reference pier and reference pier location. Example: A hinge is 50ft to the right of a pier is in the paste buffer, when applied to the paste target, it is placed 50ft to the right of the start support. If the hinge was left of a pier, then it would have been pasted left of the end support.

Actor	Action	Paste target	Result
	allowed if it causes instability.	Girderline	⊗
	Delete		Delete

3.2.1.2.1.4 In-Place Editing

The Framing Plan View shows dimensions for span lengths. These dimensions may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing of the span length. For in-place editing, the stations for piers to the left of the edited span are fixed. All the other piers move in response to span length changes.

3.2.1.2.1.5 Context Editing

These commands will become available on the context menu and main Edit menu if an applicable item is selected.

3.2.1.2.1.5.1 Mirror Properties

This command will mirror the stiffness properties if a girderline or span/support is selected. Can also mirror the entire bridge if all are selected.

3.2.1.2.1.6 Toolbox

Across the top of the Framing Plan View window is a palette of icons called the Toolbox. The purpose of the toolbox is to provide an interactive means for editing the bridge model. Icons are dragged from the toolbox and dropped onto the model. Buttons are pressed to invoke a mode or perform an action. Editing features include addition of span/supports, span hinges, girderlines, and orienting North in the window.

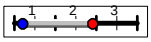
The  icon is used to add new span/supports to the model. Drag this icon from the toolbox to the location where you want to add a new span/support. The mouse cursor will indicate the orientation of the span/support when it is dropped (support on left or right of span). The new span and support will take on the properties of the drop target.

The  icon is used to add hinges to a span. Span hinges can be dropped onto any span with fewer than two hinges. Once dropped, the Span Hinge dialog is displayed so the user can position the hinge exactly. The dialog will be initialized with data based on the drop location.

The  icon is used to drop new girderlines onto the model. New girderlines take on the properties and spacing of the girderline below and adjacent to the drop. If dropped to the above of the left-most girder line, the properties for the left-most girder line are used.

The 

drop down button is used to orient North. The first option is a "best fit" orientation of north. QConBridge II attempts to align North such that the bridge fits best in the view. The second option allows the user to define North by two mouse hits in the view window. The third option makes North straight up on the screen.

The 

tool is used to zoom in on the bridge model and to select the spans for viewing.

3.2.1.2.2 ELEVATION VIEW

The Elevation view is a single-pane view displaying an elevation of the longitudinal analysis model for a selected girderline. A schematic is shown in Figure 36.

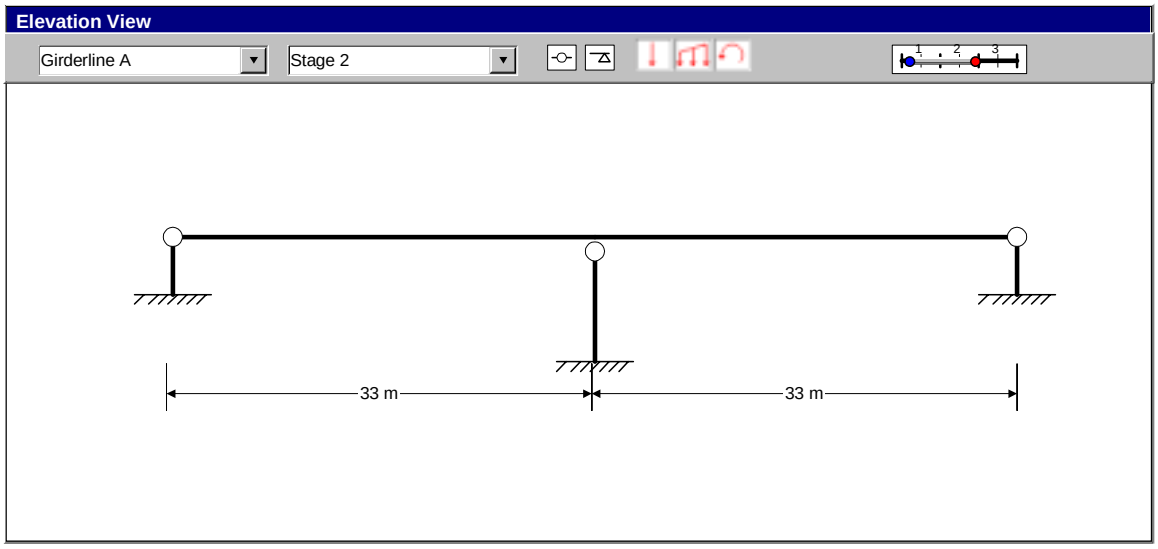


Figure 36 – Analysis Model Elevation View

3.2.1.2.2.1 Keyboard Selections

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys are used to select the span/supports, and hinges. The selection sequence using the right arrow key goes as follows: pier 1/span 1 → hinges in span 1 → span 1/pier 2 → pier 2/span 2 → … → span n/pier n+1 → pier 1/span 1. The order is reversed if the left arrow key is used. The selections in both panes are synchronized.
The up arrow will jump the selection to user-defined loads. After the load selection mode is enabled, the left and right arrow keys are used to change the selection. The down arrow key is used to jump into span/support selection mode.

3.2.1.2.2.2 Mouse Operations

The Elevation View displays an elevation view of the longitudinal analysis model for the selected girderline and supports mouse interaction with many of the interface actors as specified in the following table.

Table 10 – Mouse Actions for Elevation View

Actor	Action	Result
Span/Support	Single Click	Select Span/Support
	CTL/SHIFT-Click	Multiple select
	Double Click on Support Double Click on Span	Activate Support Editing Dialog. See Section 3.2.1.4.5 below. Activate Span Editing Dialog. See Section 3.2.1.4.5 below.
	Right Click	Context menu – Edit Pier – Edit Span – Edit Framing Plan
	Drag	Move span support to another location – animated mouse feedback - Same semantics as the Framing Plan
	Ctl-Drag	Copy span/support to another location - Same semantics as the Framing Plan

Actor	Action	Result
Span Hinge	Single Click	Select
	CTL/SHIFT-Click	Select – no multiple
	Double Click	Activate Span Hinge Editing dialog
	Right Click	Context menu
	Drag	Move hinge to another location. The distance from the hinge to its reference pier is maintained. The orientation of the piers is changed to match the skew angle of the new reference pier. The left/right location of hinge, with respect to the reference pier, is defined by where the hinge is dropped. Example: a hinge that is 10ft left of pier 1 is dropped to the right of pier 3 will be positioned 10ft to the right of pier 3, parallel to pier 3. If the hinge falls in the same location as an existing hinge, the hinges are merged into one. A hinge cannot be dropped in a span if it causes instability.
	Ctl-Drag	Copy span hinge to another location
Loads	Single Click	Select
	CTL/SHIFT-Click	Select multiple
	Double Click	Activate appropriate load editing dialog
	Right Click	Context menu
	Drag	Move load – animated mouse feedback. Load positioning data is maintained. Example: Load is dragged from span 1 to span 2, distance from start of span 1 is used as the distance from the start of span 2 after the drag. If span is too short to fit load, positioning is changed to fractional.
	Ctl-Drag	Moves a copy of the load - See Drag for details

3.2.1.2.2.3 Copy and Paste Operations

Some of the actors in the Elevation View also support Delete, Cut, Copy and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled.

Table 11 - Copy and Paste Operations for Elevation View

Actor	Action	Paste target	Result
Span/Support	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer (Can't cut if only one span)
	Paste	Span/ Support	Copy properties to paste target
		Span Hinge Slab	⊗ ⊗
		Nothing	Pastes span/support after last support
	Paste Special	Same as Paste	Option to mirror properties
	Delete		Delete (can't delete if only one span)

Actor	Action	Paste target	Result
Span Hinge	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste	Span Hinge	Paste distance from reference pier to paste target
		Span/ Support	Paste hinge onto span retaining the distance from reference pier and reference pier location. Example: A hinge is 50ft to the right of a pier is in the paste buffer, when applied to the paste target, it is placed 50ft to the right of the start support. If the hinge was left of a pier, then it would have been pasted left of the end support.
	Delete		Delete
Loads	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste	Span Hinge	Paste load at span hinge
		Span/ Support	Paste load onto span using location data for the load.
	Delete		Delete

3.2.1.2.2.4 In-Place Editing

The Elevation View shows dimensions for span lengths. These dimensions may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing of the span length. For in-place editing, the location of support 1 is fixed. All other supports move as required.

3.2.1.2.2.5 ToolBox

Across the top of the Elevation View window is a palette of icons (toolbox). The purpose of the toolbox is to provide an interactive means for editing the bridge model. Editing features include addition of span/supports and span hinges and applying loads to the analysis model.

3.2.1.2.3 PIER VIEW

The Pier View is used to view the 2D frame model transverse to the bridge for pier models with 3D attributes in the BAM. 2D Pier models have no dimension in this view, so when a 2D pier model is selected, the string "*2D Model – Nothing to View*" will be shown in the main portion of the window. Note that pier models are not stage-dependent in QConBridge II.

This view is nearly identical to the main view for Transverse Analysis Projects. The primary difference is the combo box that allows selection of the current pier to be edited. Also, results from the longitudinal analyses for all girderlines are transferred into the pier model. For BAMs with more than one LBAM , bearing locations cannot be edited because the framing plan defines the bearing locations. For BAMs with one LBAM, the bearing locations can be input and the distribution of the LBAM reaction is input. Load editing is limited when pier models are embedded into a BAM. Refer to the Transverse Analysis Modeling Editor reference for details on all interactions with this view. The editing operations available in BAM mode are presented in this section.

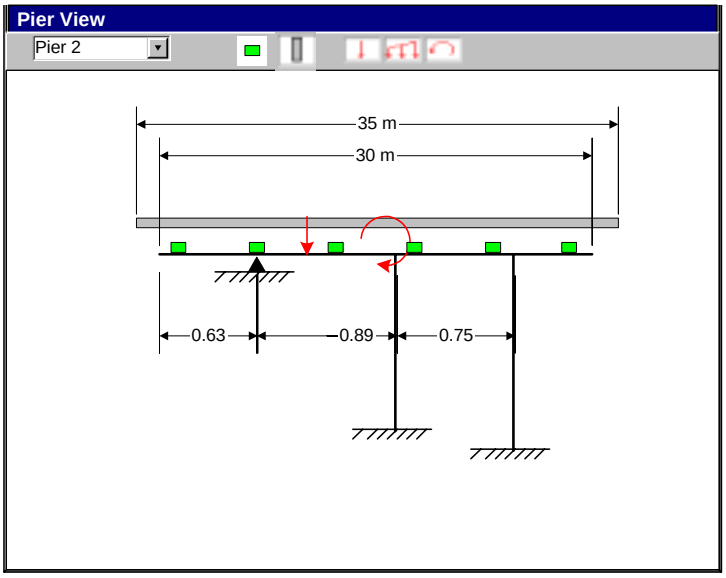


Figure 37 - Pier View

3.2.1.2.3.1 Keyboard Selections

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys are used to select the cap beam and columns. The selection sequence using the right arrow key goes as follows: cap beam → column 1 → column 2 →…→ column n → cap beam. The order is reversed if the left arrow key is used.
The up arrow will jump the selection to user-defined loads. After the loads selection mode is enabled, the left and right arrow keys are used to change the selection. The down arrow key is used to jump into member selection mode.

3.2.1.2.3.2 Mouse Operations

The Pier View displays an elevation view of the selected 3D pier analysis model and supports mouse interaction with many of the actors as specified in the following table.
Bearings are shown using ■ icons. Bearings cannot be manipulated in the model (except for models with a single LBAM), but point loads and moments can be applied to them.

Table 12 – Mouse Actions for Pier View

Actor	Action	Result
Pier Cap	Single Click	Select Pier Cap
	CTL/SHIFT-Click	Select cap – only one to select
	Double Click	Activate Transverse Pier Model editing dialog
	Right Click	Context menu
	Drag	⊗
	Ctl-Drag	⊗
Column	Single Click	Select
	CTL/SHIFT-Click	Select multiple
	Double Click	Activate Column Editing dialog

Actor	Action	Result
	Right Click	Context menu If multiple, adjacent Columns are selected, an “Equal Spacing Tool” is made available that displays a dialog to enter an equal spacing between the columns.
	Drag	Moves column (and the spacing to the right of the source column) to the right of the column adjacent to where it is dropped.
	Ctl-Drag	Moves a copy of the source column as described in Drag
Loads	Single Click	Select
	CTL/SHIFT-Click	Multiple Select
	Double Click	Activate Load Editing dialog
	Right Click	Context menu
	Drag	Move load to new location. Load is dragged along the cap or column based on mouse location. If multiple loads are dragged, the "No" icon is displayed and a tool tip telling the user that only single loads can be dragged is displayed
	Ctl-Drag	Moves a copy of the load. See Drag for details
Bearings	Single Click	Select
	CTL/SHIFT-Click	Multiple Select
	Double Click	Activate Bearing Editing dialog
	Right Click	Context menu
	Drag	Move load to new location. Bearing is dragged along the cap or column based on mouse location. If multiple bearings are dragged, the "No" icon is displayed and a tool tip telling the user that only single bearings can be dragged is displayed
	Ctl-Drag	Moves a copy of the bearing. See Drag for details

3.2.1.2.3.3 Copy and Paste Operations

Some of the actors in the Pier View also support Delete, Cut, Copy and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled.

Table 13 - Copy and Paste Operations for Pier View

Actor	Action	Paste target	Result
Loads	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste	CapBeam	Paste load using location data for the load.
		Bearings	Paste load to bearing
		Column	Paste load using location data for the load.
	Delete		Delete
Pier Cap	Copy		Copy to paste buffer
	Cut		Copy to paste buffer (can’t delete)
	Paste	Pier Cap	Paste properties to paste target
	Paste Special	Pier Cap	Option to mirror properties

Actor	Action	Paste target	Result
	Delete		can’t delete
Column	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer (can't cut last one)
	Paste	Column	Paste properties to paste target
		Pier Cap	Paste a new column on the right hand side of the pier using the right-most column spacing. If the new column is beyond the end of the cap beam, the cap beam is extended, maintaining the current column/cap overhang.
	Paste Special	Same as Paste	Option to mirror properties
	Delete		Delete (can’t delete last one)

3.2.1.2.3.4 In-Place Editing

The Pier View shows dimensions for cap length, roadway width, and column locations. These dimensions may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing.
When columns are selected, a dimension line showing the column length is then displayed. Users can click on the text in the dimension line to change the column length.

3.2.1.2.3.5 Context Editing

These commands will become available on the context menu and main Edit menu if an applicable item is selected.

3.2.1.2.3.5.1 Mirror Properties

This command will mirror the stiffness properties if a cap beam or a column is selected. Can also mirror the entire pier model if all are selected.

3.2.1.2.3.6 Toolbox

Across the top of the Pier View window is a palette (toolbox) for icons and widgets applicable to the pier view only. The purpose of the toolbox is to provide an interactive means for editing the pier analysis models. The combo boxes allow selection of the current pier and selection of analysis stages. Editing features include addition of columns, bearings and loads to the analysis model. Bearings ( ) may only be applied to models with a single girderline only.

3.2.1.2.4 VIEW SETTINGS

All views will have the capabilities to turn dimensions off and on, and to overlay member area or moment of inertia information in a graph-like style.

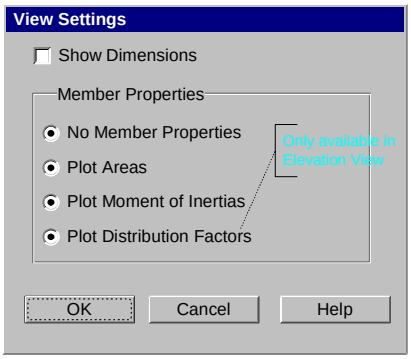


Figure 38 - View Settings

3.2.1.3 Bridge Contractor

The Bridge Contractor is a wizard used to edit bridge and create QConBridge II projects. The Bridge Contractor is reentrant for all BAM project types.

3.2.1.3.1 STEP 1

Step 1 is seen for all types of projects. It allows you to choose the project type and system-level properties for a project. If a new project type is selected, all data for the existing project is deleted. The Project Type can only be selected the first time through the wizard.

Figure 39 - Bridge Contractor Step 1

3.2.1.3.2 STEP 2 –ANALYSIS SETTINGS

This step is used to define special settings for the analysis. Truck library entries to be used for the "Permit" and "Special" cases are selected here.

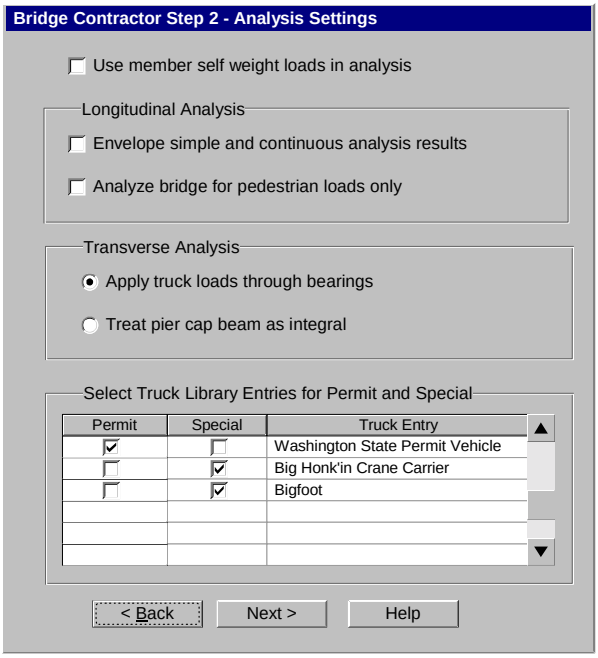


Figure 40 - Bridge Contractor Step 2 – Analysis Settings

3.2.1.3.3 STEP 3 – STAGES

In this step we define stages. This step is similar to Figure 54 - Edit Stages.

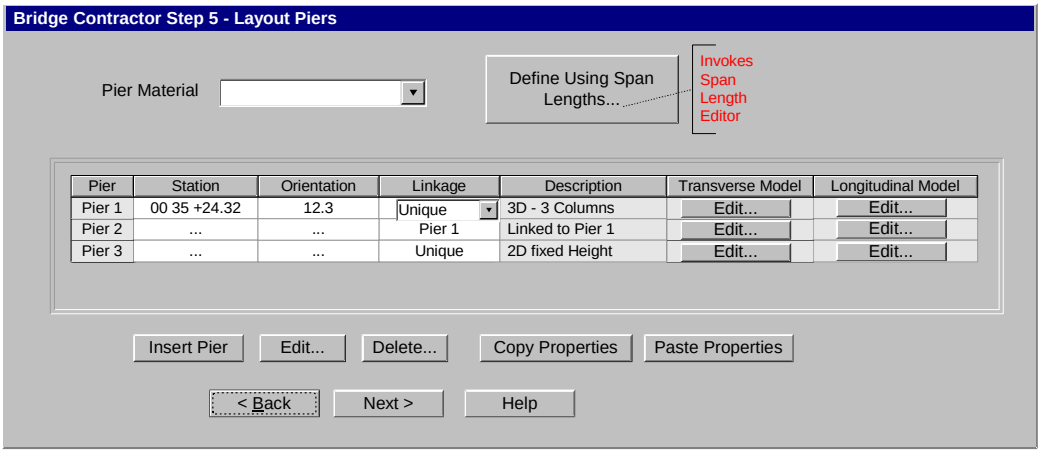
3.2.1.3.4 STEP 4 – ALIGNMENT

In step 4 we define the alignment. This step is similar to Figure 46.

3.2.1.3.5 STEP 5 – PIER LAYOUT

This step in the Bridge Contractor is used to define the Pier layout and to edit piers. Default piers are 2D zero-height with simple support connections.

Piers can have linkage. This means that piers can reference other piers for their properties. If a pier is linked to another, it's location and orientation can be unique; however, it inherits all other properties from the pier that it is linked to. If a linked pier is changed to a unique pier, it takes on a copy of all of the traits (except location and orientation) of the pier it was linked to. Properties for linked piers cannot be edited.



F**Figure 41 - Bridge Contractor Step 5**

The alignment offset is measured normal to the alignment at the pier station. Positive is right looking up station.

The transverse pier model data can be edited from the grid by selecting the Edit button. Transverse model editing is explained in Section 3.2.1.4.8.3 below.
Longitidinal pier model data is editing by pressing the Edit button. This dialog is shown in Figure 50.
Pressing the Delete button causes a dialog to come up which asks which span is to be deleted with the pier.
When the Insert Pier button is pressed, the Edit Pier Dialog is displayed. You must then select the pier location and pier properties. When a pier is inserted between two existing piers, the span definition is fit between the left pier and the new pier[2]. The new span created between the new pier and the right pier is a copy of the span that is left of the new pier. When a pier is inserted before or after ends of the bridge, the superstructure is extended using the same girders and spacing as the previous outer span.

3.2.1.3.6 STEP 6 – SUPERSTRUCTURE LAYOUT

This step is used to define the girderlines and girder properties for the superstructure.

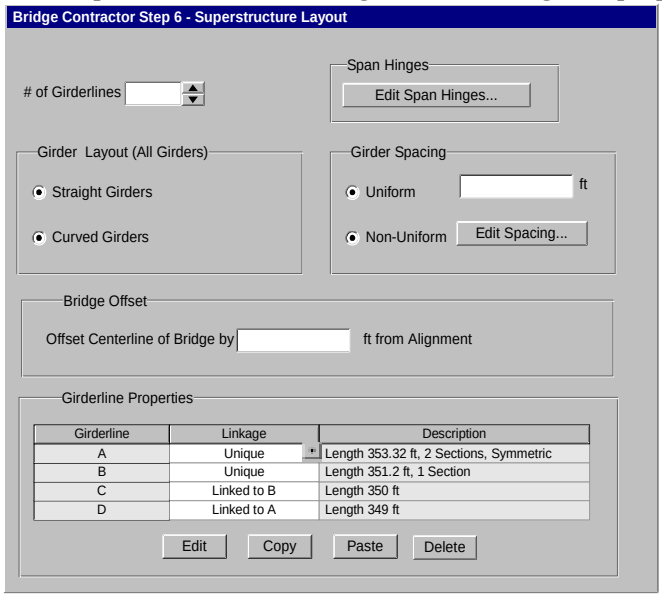


Figure 42 - Bridge Contractor Step 6 - Superstructure Layout

When the number of girderlines is increased from this dialog, the new girderline(s) are made unique and given the same properties and spacing as the last girderline in the group.

3.2.1.3.7 STEP 7 – ROADWAY

This step is used to define the width of the roadway. This step is similar to Figure 53 - Roadway Widths. This information is used for the transverse analysis.

3.2.1.4 Detailed Editing for Actors

This section describes the primary dialogs for editing each of the major UI actors for BAM editing. Access to these dialogs is either though context menus from the main views, or by menu selections.

3.2.1.4.1 ANALYSIS SETTINGS

Analysis Settings affect the way the model is analyzed. This dialog is similar to Figure 40 - Bridge Contractor Step 2.

[2] Any of the span definition that does not physically fit between the new pier arrange is retained in the span definition, but is marked as ignored.

3.2.1.4.2 BRIDGE

The Bridge is the entire model. There are some actions that can be carried out on the entire bridge as follows

3.2.1.4.2.1 Move Bridge

This command translates the entire bridge along the alignment a given distance, or moves the location of the first pier to the desired station.

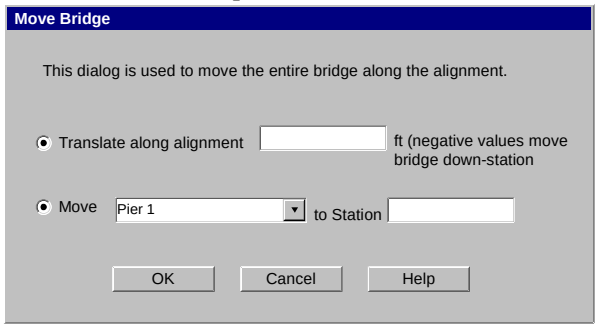


Figure 43 - Move Bridge

3.2.1.4.2.2 Edit Span Lengths

This dialog allows editing of span lengths directly in lieu of pier stationing.

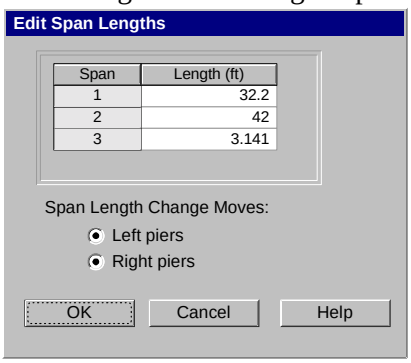


Figure 44 - Edit Span Lengths

3.2.1.4.2.3 Edit Girder Spacing

Girder Spacing can be non-uniform in the BAM. This dialog allows users to edit individual girder spaces.

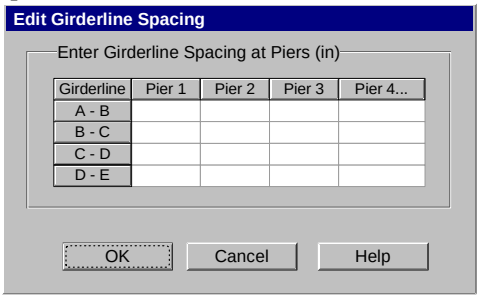


Figure 45 - Edit Girder Spacing

The grid control supports selection of multiple cells. If multiple cells are selected, a “edit multiple” dialog box appears that allows the user to specify girder spacing for all selected values.

3.2.1.4.3 ALIGNMENT/ROADWAY

The Alignment defines the horizontal orientation of the bridge. It also allows the definition of a datum elevation from which to measure the elevations of column tops and bottoms.

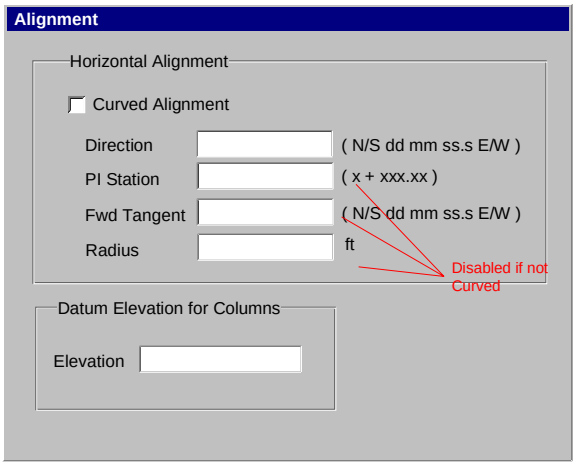


Figure 46 - Alignment

3.2.1.4.4 FRAMING PLAN

The Framing plan is a dialog-based actor that provides a compact and complete way of editing the bridge model's pier layout and superstructure layout. The Framing Plan is a multi-tab dialog that consists of the same dialogs given in Bridge Contractor Steps 5-6 above.

3.2.1.4.5 SPAN/SUPPORTS

The primary purpose for Span Support actors in QConBridge II is to provide a clean way to add, move, and delete spans in from the plan and elevation views. Dialog editing for Span/Supports is restricted to editing either the length of the selected span, or editing the selected pier.

3.2.1.4.5.1 Selection of Span/Supports

Selection of Span/Supports is identical to selection of span/piers in product models as discussed in Section 3.1.4.5.1 above.

3.2.1.4.5.2 Add Span/Support

Adding Span/Supports to the model is identical to adding of span/piers in product models as discussed in Section 3.1.4.5.3 above.

3.2.1.4.5.3 Edit Span

This dialog is shown when Edit Span is selected from the context menu if a Span/Support is selected. The correct icons must be shown depending on whether the pier in question is fixed-height or a simple support. Refer to Table 17 for guidelines.

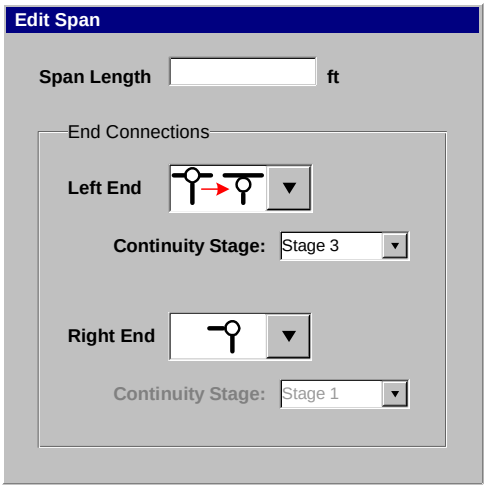


Figure 47 - Edit Span

3.2.1.4.6 GIRDERLINES

The primary purpose of girderline actors is to allow GUI interaction to add/delete/move girderlines, and to provide schedule-based editing for section properties.

3.2.1.4.6.1 Editing Section Property Schedules for Girderlines

This dialog allows editing of section properties along the length of a selected girderline. The longitudinal measurement can be from start to end of the entire girder line, or from start to end of a selected span. This dialog is accessed from the Framing Plan, or from the context menu when a girderline is selected.

Note that section properties are stage-dependent and properties for all stages must be entered. The stage selection tabs support multi-tab selection (using CTRL-click), so properties can be entered for multiple stages simultaneously.

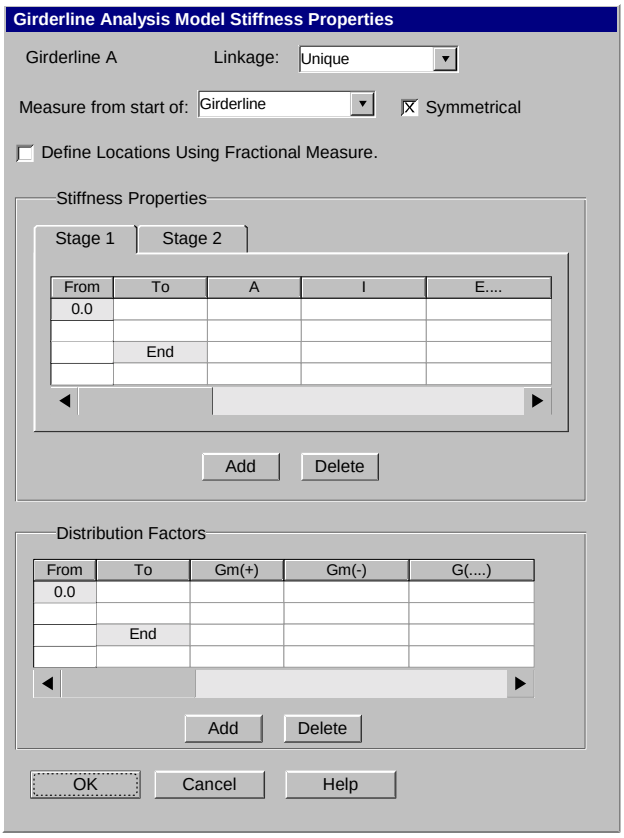


Figure 48 – Section Property Schedule for Girderlines

3.2.1.4.6.2 General Notes About Schedule-Based Editing

Section property schedules are described in segments along the length of the girderline using a "From"-"To" format. The From-To values can either be specified from the beginning to end of the given girderline, or from the centerline of the piers at the beginning and end of a given span.

3.2.1.4.6.2.1 Exact or Fractional Locations

Lengths can be defined using exact distances (in the current units), or by describing decimal fractional values.

3.2.1.4.6.2.2 Resolution When a Span Length Is Changed

Redefinition of span lengths can cause problems to section property schedules. This happens when lengths are shortened and segments no longer fit in the increment for which they were defined. In this case, the segment is highlighted in red and ignored (or middle if a symmetrical girder).

3.2.1.4.6.2.3 Resolution of Linked and Pasted Property Schedules.

Problems can occur if an explicit splice location from a longer girder is pasted or linked to a shorter girder such that the splice is located beyond the end of the shorter girder. In this case, the segment is simply highlighted in red and ignored (or middle if a symmetrical girder).

3.2.1.4.6.3 Girderline Stiffness Properties

Girderline stiffness properties are as follows:

Table 14 - Girderline Stiffness Properties

Property	Description
A	Area
A_{sw}	Area for self weight
I_{eff}	Moment of inertia based on effective flange width
I_{full}	Moment of inertia based on full flange width (used for deflections, see LRFD 4.6.2.6.1)
γ	Weight Density
E	Modulus of Elasticity

3.2.1.4.6.4 Girderline Distribution Factors

Distribution factors are as follows:

Table 15 - Girderline Distribution Factors

Property	Description
Gm(str+)	Strength, positive moment
Gm(str-)	Strength, negative moment
Gv(str)	Strength, shear
Gm(ftg)	Fatigue, positive moment
Gm(ftg)	Fatigue, negative moment
Gv(ftg)	Fatigue, shear

3.2.1.4.7 SPAN HINGES

This dialog is used to add, delete, and edit span hinge properties. A hinge cannot be added if it causes the structure to become unstable.

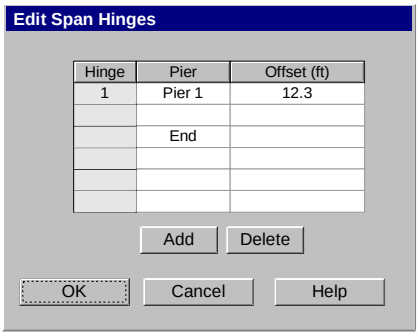


Figure 49 - Edit Span Hinges

3.2.1.4.8 PIERS

BAM's have two independent analysis models for each pier: A model used for the longitudinal analysis, and a model used in the transverse analysis. These models are described independently: Longitudinal models are accessed from the UI in the Framing Plan or the Elevation View; Transverse models are accessed via the Framing Plan View (via context menu only), Framing Plan dialog, or the Pier view.

3.2.1.4.8.1 Longitudinal Pier Models

Longitudinal pier models describe the pier for the longitudinal analysis.

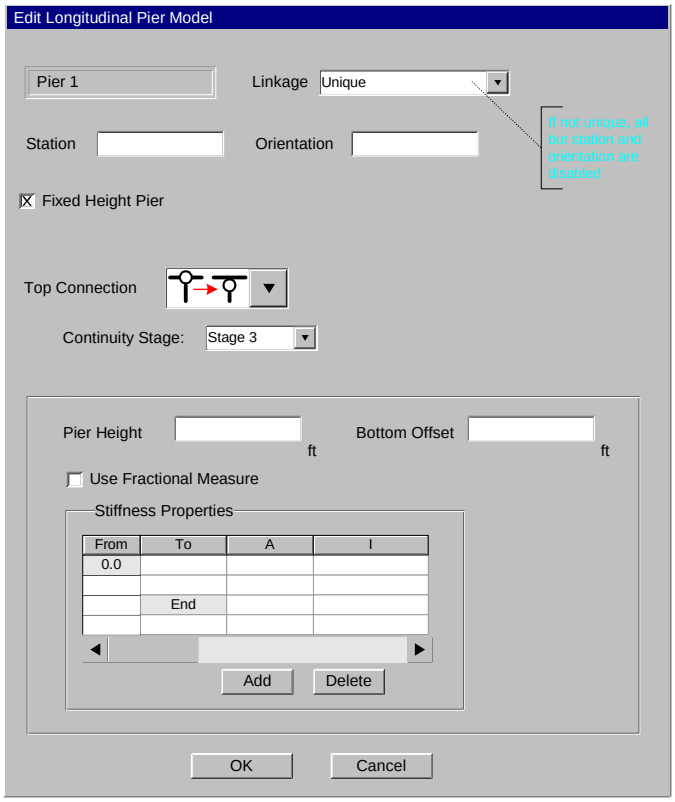


Figure 50 - Edit Longitudinal Pier Model

3.2.1.4.8.1.1 Pier Stiffness Properties

Pier stiffness properties are for the entire pier. This means that each girderline model receives 1/nth of the stiffness of the pier, where n is the number of girderlines. Pier stiffness properties are as follows:

Table 16 - Longitudinal Pier Stiffness Properties

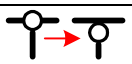
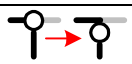
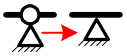
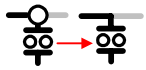
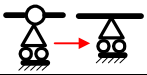
Property	Description
A	Area
I	Moment of inertia
γ	Weight Density
E	Modulus of Elasticity

3.2.1.4.8.2 Longitudinal Connections

Longitudinal connections describe the boundary conditions between the superstructure and the pier for the longitudinal analysis. The connection types can be selected in the Edit Span and Edit Longitudinal Pier Model dialogs. Each available connection type is represented as an icon as shown in the following table. Icons at abutments are shown with the appropriate side grayed out. Note that right abutments are shown only – left abutments are similar.

Table 17 - Connection Types

Fixed Height Support	**Simple Support (Zero Height)**	**Description**

At Piers	At Abutments	At Piers	At Abutments	
				Fully Pinned
				Pinned On Roller
				Continuous superstructure – pinned columns
				Continuous On Roller
				Pinned left, fixed right
				Fixed left, pinned right
				Fixed
				Simple – Continuous
				Simple - Fixed
				Simple – Continuous on Roller

3.2.1.4.8.3 Transverse Pier Models

Transverse pier models describe the 2D pier model for the transverse live load analysis. The Transverse Pier Analysis Model dialog is used to describe piers.
The dialog is presented using two tabs: The first tab is used to describe the width and stiffness properties of the pier cap; and the second tab is used to describe the column layout and stiffness.

3.2.1.4.8.3.1 Tab 1 – Cap Description

The first tab allows the user to choose whether to describe the full pier. If a full pier model is chosen, the cap beam and columns must be described. The cap length must be greater than or equal to the total width of girders crossing the pier.

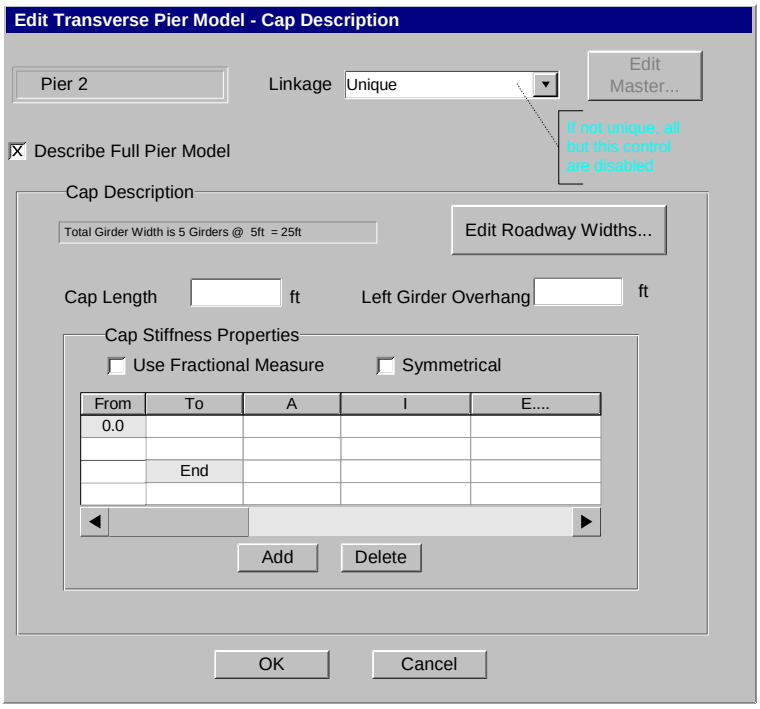


Figure 51 - Edit Transverse Pier Model

3.2.1.4.8.3.2 Cap Stiffness Properties

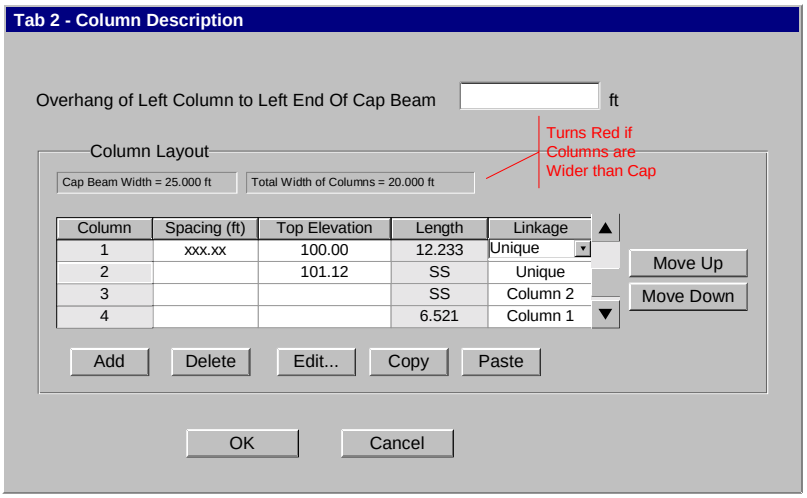
Cap stiffness properties are as follows:

Table 18 - Cap Stiffness Properties

Property	Description
A	Area
I	Moment of inertia
γ	Weight Density
E	Modulus of Elasticity

3.2.1.4.8.4 Tab 2 – Column Layout

This tab is used to describe the columns in the pier. This tab is disabled if a full pier model is not described. Columns may be zero-length (Simple Support), or fixed-height – in which case section properties must be described



3.2.1.4.8.5 Columns

Individual columns in a pier model are edited as follows. Simple support columns are the default and do not need column length or property descriptions.

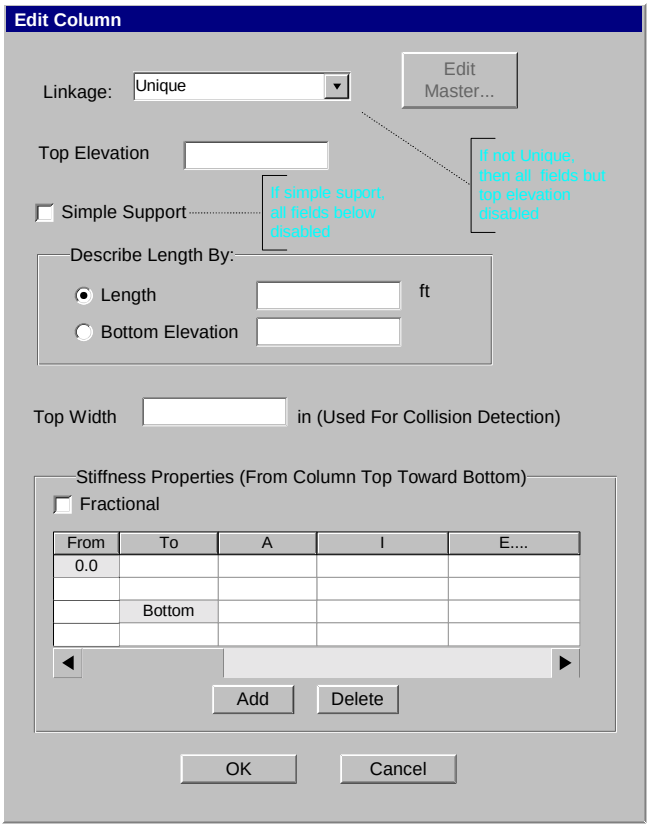


Figure 52 - Edit Columns

3.2.1.4.8.5.1 Column Stiffness Properties

Column stiffness properties are as follows:

Table 19 - Column Stiffness Properties

Property	Description

A	Area
I	Moment of inertia (in plane of bent)
γ	Weight Density
E	Elastic Modulus

3.2.1.4.9 ROADWAY WIDTHS

Roadway widths must be defined at each pier in order to perform the transverse live load analysis. Overhang is to edge of roadway. Overhangs are measured from the ends of the pier cap. These values are ignored if a full pier model is not described.

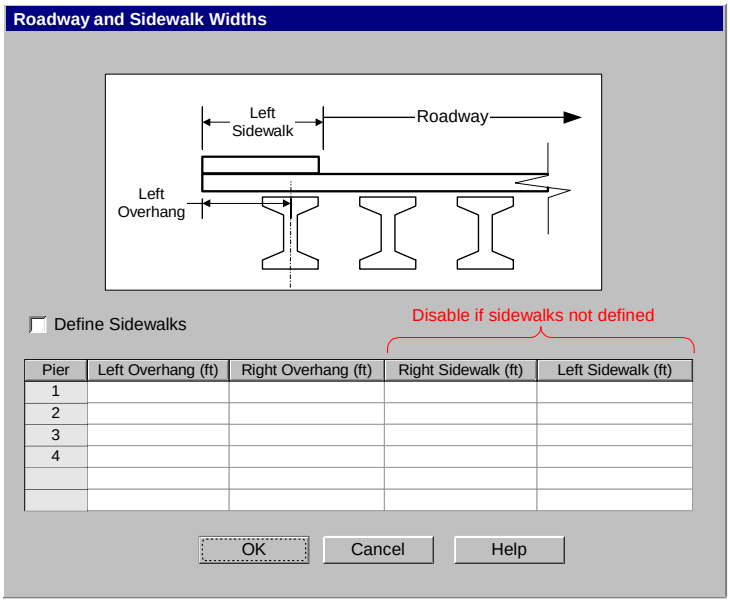


Figure 53 - Roadway Widths

3.2.1.5 Stage Management

Stages act as containers for loading and girderline stiffness information in a longitudinal analysis.

Stages are only applicable to the longitudinal analysis. The user is free to create any number of stages required to adequately analyze the structure. The following describes the stage management features of the Analysis Model Editor. Except for Stage Selection, these features are only available in Analysis Model Mode.

Stages are edited via the following dialog:

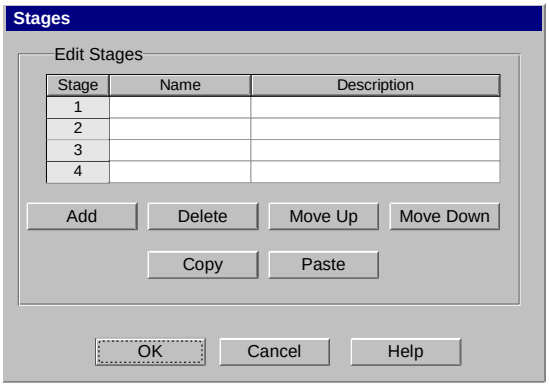


Figure 54 - Edit Stages

3.2.1.5.1 ADD STAGE

Newly created stages are exactly the same as the last stage except the loads aren't copied (this is to prevent double loading of the structure). The new stage name will be "Stage *n*", where *n* is the next available stage number.

3.2.1.5.2 DELETE STAGE

One or more stages can be deleted by selecting them on the grid and pressing Delete (keyboard or button). All loads and other information associated with the stage are also deleted. At least one stage must remain (you can't delete the last stage).

3.2.1.5.3 MOVE STAGE

Sequencing of stages can be changed using the Move Up and Move Down buttons. Stage rows may also be moved via drag and drop within the grid control.

3.2.1.5.4 COPY STAGE

The properties (loads and stiffness properties) of a stage can be copied to any other stage using the Copy and Paste buttons.

3.2.1.5.5 MISCELLANEOUS

When more than one stage is selected, only the Delete and "Move" options are available.

3.2.1.6 Load Editing on Longitudinal Models

The subsections that follow describe the various facilities available to the user for managing user-defined externally applied loads applied to longitudinal models.

Note: this section only describes the application of superstructure loads. Refer to the Transverse Analysis Model Editor section for a description of the user interface for editing loads applied to the TBAM.

3.2.1.6.1 ADD/EDIT LOAD

3.2.1.6.1.1 Manual Method

In the manual method, loads are applied to the currently selected span. To add a load select Loads | Add *type* Load or right click on the selected span and select Add *type* Load from the context menu, where *type* is Point, Linear, or Moment. For either case, the appropriate load input dialog is displayed to the user.

To edit a previously defined load, select the load and choose Edit | Properties. This will present the user with the appropriate Load Input Dialog.

3.2.1.6.1.2 Interactive Method

Loads can be interactively added to a span by dragging an icon from the toolbox palette and dropping the icon onto a span. When the icon is dropped on a span, the appropriate load input dialog is displayed.

The load icon must be dropped in the proximity of the span. When the cursor is some distance away from the span, the icon has the ∅ superimposed on it, indicating that the cursor is not over a valid drop zone.

The user may cancel this drag and drop operation by either pressing the Esc key or by dropping the icon somewhere other than a valid drop zone.

Loads can be interactively edited by double clicking on the load, selecting the load and pressing the Enter key, or right clicking on the load and selecting Properties from the context menu. This will present the user with the appropriate Load Input Dialog.

3.2.1.6.1.3 Load Input Dialogs

The Load Input Dialogs are used for creating or editing individual loads. The location of a load can be measured with an absolute distance or a fractional distance. The location of a load is measured from the left end of the span to which it is applied.

Loads with a positive magnitude are applied in the direction of gravity or, in the case of moments, counter-clockwise (moment vector out of the screen - positive Z direction).

If the "All Spans" option is selected, the load is applied to all spans. For this option, the location of the load must be defined using fractional measure. When the All Spans option is checked, the Fractional box will automatically become checked (if it isn't already) and disabled. The Location fields will convert absolute measure distances to equivalent fractional distances. If multiple spans are selected, the load must be defined using fractional measure as well.

The user must associate the load with a Load Group and a stage of application. Load Groups are predefined by the Analysis Criteria library entry.

3.2.1.6.1.3.1 Component Loaded

Each of the user-defined loads must be applied to one or all spans in the BAM. The following options are provided:

Option	Action Taken
Span 1…n	Applies load to selected span
All Spans	Applies load to all spans in the model

3.2.1.6.1.3.2 Point Load

The load input dialog for Point Loads is shown in Figure 55. A point load is defined by its location and magnitude.

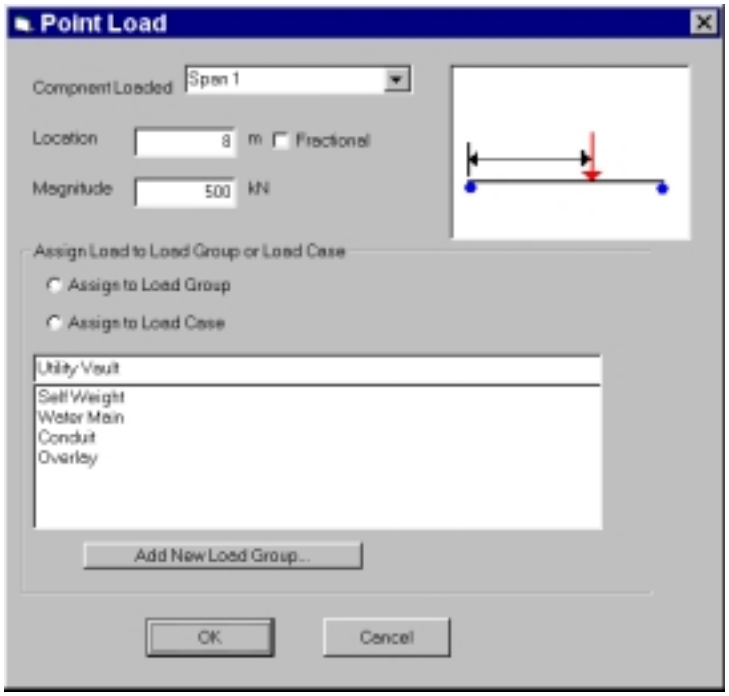


Figure 55 – Edit Point Load

3.2.1.6.1.3.3 Linear Load

The load input dialog for Linear Loads is shown in Figure 56. A linear load is defined by its starting and ending location and the intensity of the load at its starting and ending position. The user can select the Uniform Load option button to enter a linear load with a constant intensity.

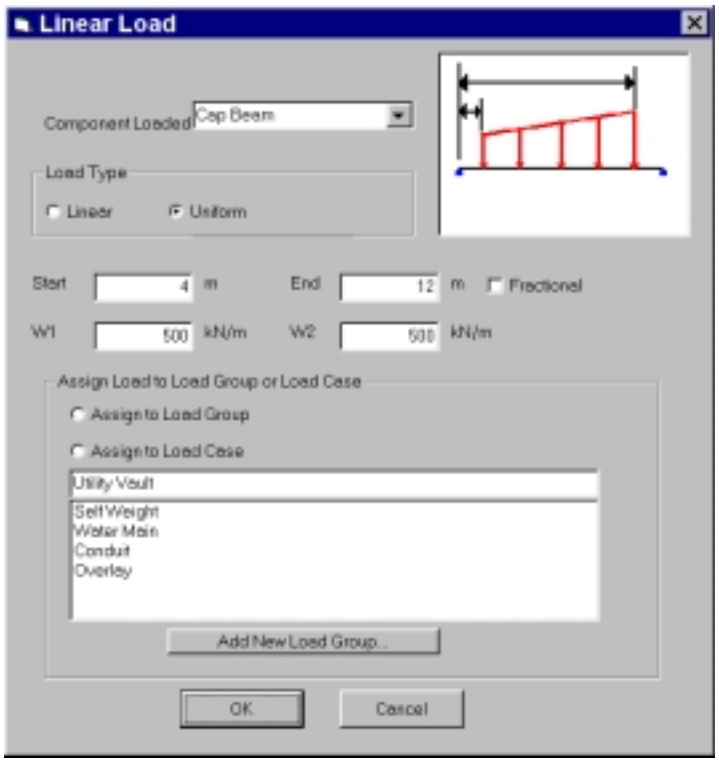


Figure 56 – Edit Linear Load

3.2.1.6.1.3.4 Moment Load

The load input dialog for Moment Loads is shown in Figure 57. A moment load is defined by its location and magnitude.

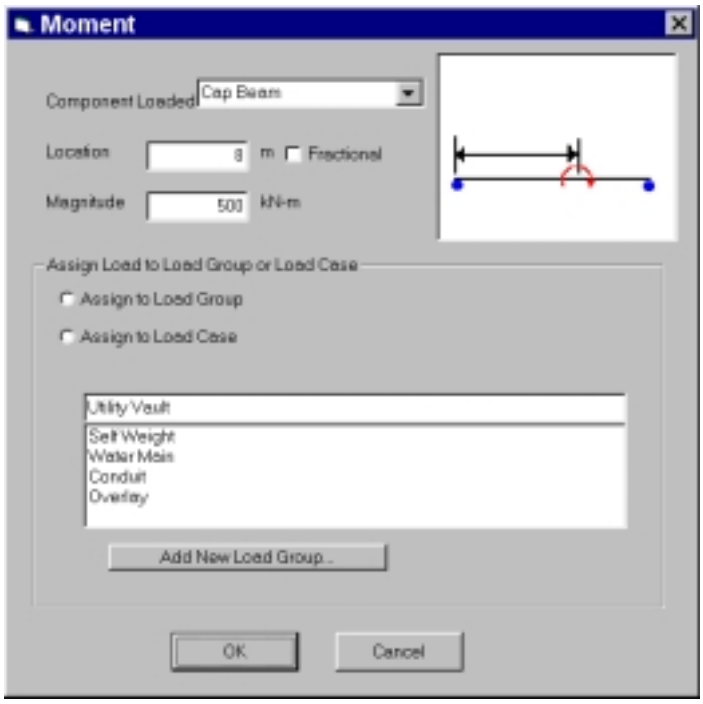


Figure 57 – Edit Moment Load

3.2.1.6.2 CUT/DELETE LOAD

Loads are removed from the analysis model by cutting or deleting them. Cutting a load places a copy of it on the clipboard for subsequent paste operations. Deleting a load removes it from the analysis model.

To cut a load, select the load and choose Edit | Cut, press the right mouse button and choose Cut from the context menu, or press Ctrl+X.

To delete a load, select the load and choose Edit | Delete, press the right mouse button and choose Delete from the context menu, press Ctrl+D, or press the Delete key.

3.2.1.6.3 COPY LOAD

The currently selected load can be copied to either the clipboard for a subsequent paste operation or copied to another span by dragging onto that span.

3.2.1.6.3.1 Copy to Clipboard

To place a copy of a load onto the clipboard, select the load and choose Edit | Copy, right click on the load and select Copy from the context menu, or press Ctrl+C.

3.2.1.6.3.2 Copy to Span

A load can be copied to another span by dragging it onto that span. To copy a load, press and hold down the Ctrl key, grab the load with the left mouse button, drag the load to a different span, and release the load onto the span by releasing the left mouse button. The mouse cursor will become charged with the “+” symbol.

If the load is dropped on the source span, this will result in two identical loads at the same location. See Pasting Semantics below for details on how QConBridge will resolve load-positioning issues.

3.2.1.6.4 PASTE LOAD

Loads copied or cut to the clipboard can be pasted to any span in the structure. To paste a load, select the target span and choose Edit | Paste, right click on the span and select Paste from the context menu, or press Ctrl+V.

3.2.1.6.4.1 Pasting Semantics

The following describe the behavior of the program when loads are pasted.

3.2.1.6.4.1.1 Load Defined with Fractional Measure

A load defined with fractional measure easily pastes into any span in the structure. The fractional measure is maintained.

3.2.1.6.4.1.2 Load Defined with Absolute Measure

One of two things can happen when a load defined with absolute measure is pasted into a span. The load with either fit into the span or it wont. If the load fits, it is simply added to the span.

If the user is dragging the load with the mouse, and it wont fit onto the span as defined, a tool tip will pop up that says "Load is beyond end of span… Making Fractional". When the load is dropped, it is made fractional.

If the user is pasting the load from the clipboard, and the load wont fit, it is made fractional.

3.2.1.6.5 MOVE LOAD

A load can be moved to a different span using the mouse, as described above in 3.2.1.6.3.2, except the Ctrl key is not used.

A load can also be moved or copied using Right mouse drag. Click with right mouse and drag the load to the desired span. When dropped, the user is presented with a context menu. The user has to choose Copy or Move.

3.2.1.6.6 SETTLEMENT LOADS

Settlement loads are applied to individual supports in the final stage of the analysis model. To input settlement loads, select a support in any stage and choose Loads | Support Settlement…, or right click on the support and choose Support Settlement... from the context menu. The Support Settlement Load dialog will be displayed. Settlement loads are automatically assigned to the “Settlement” load case.

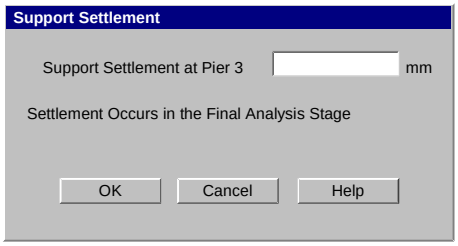


Figure 58 – Edit Support Settlement

When a support is copied or moved, the support settlement load remains associated with the support.

3.2.1.6.7 UNIFORM TEMPERATURE LOADS

Temperature loads are applied to entire structure in the final stage of the analysis model. To input temperature loads, select Loads | Temperature. The Temperature Load dialog will be displayed. Temperature loads are automatically placed in the "Temperature" load case.

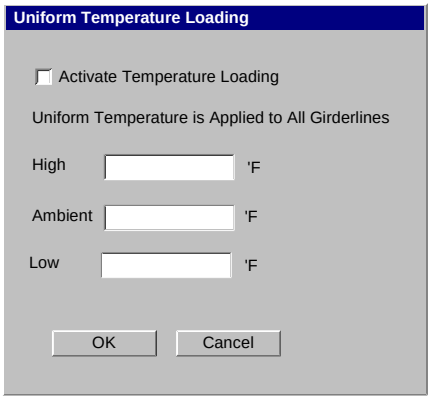


Figure 59 – Edit Temperature Loads

3.2.1.6.8 SELF-WEIGHT LOADS

Self-weight dead load is computed as the dead load area times the unit weight of the material. Self-weight dead load is applied to the entire structure during all stages. Self-weight dead load is either generated or its not. To Enable/Disable generation of self-weight dead load choose the Analysis Settings dialog as shown in Figure 40. Self-weight loads cannot be disabled in product model projects.

3.2.1.6.9 PEDESTRIAN LOADS

Pedestrian loads are applied to the left and right sides of the superstructure. They are distributed using the same rules as sidewalk dead load as defined in the spec entry. Loads are specified for the left and right sides of the structure.

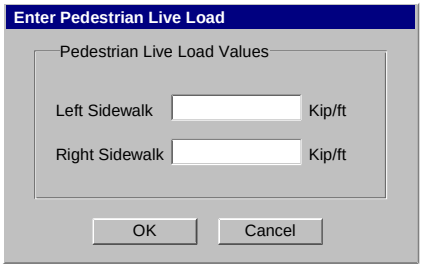


Figure 60 - Edit Pedestrian Loads

3.2.2 Transverse Analysis Model Editing

This section describes the various elements of the Transverse Analysis Model Editor. The Transverse Analysis Model Editor is used to display and edit the Transverse Bridge Analysis Model (TBAM), which is a model of a single pier taken from a bridge structure.

Much of the functionality of this editor is used to edit pier models in the Bridge Analysis Model Editor. However, the Transverse Analysis Model Editor also is a stand-alone application that can be used to edit custom pier configurations. The main differences between the Transverse Model Editor and using the Bridge Analysis Model editor are:

Only one pier at a time is edited.

A single stage is available for loading

The user must define bearing locations.

Live loads and dead loads from a longitudinal analysis must be entered manually.

3.2.2.1 Transverse Analysis Model Editor Actors

The Transverse Analysis Model Editor deals with the following UI Actors:

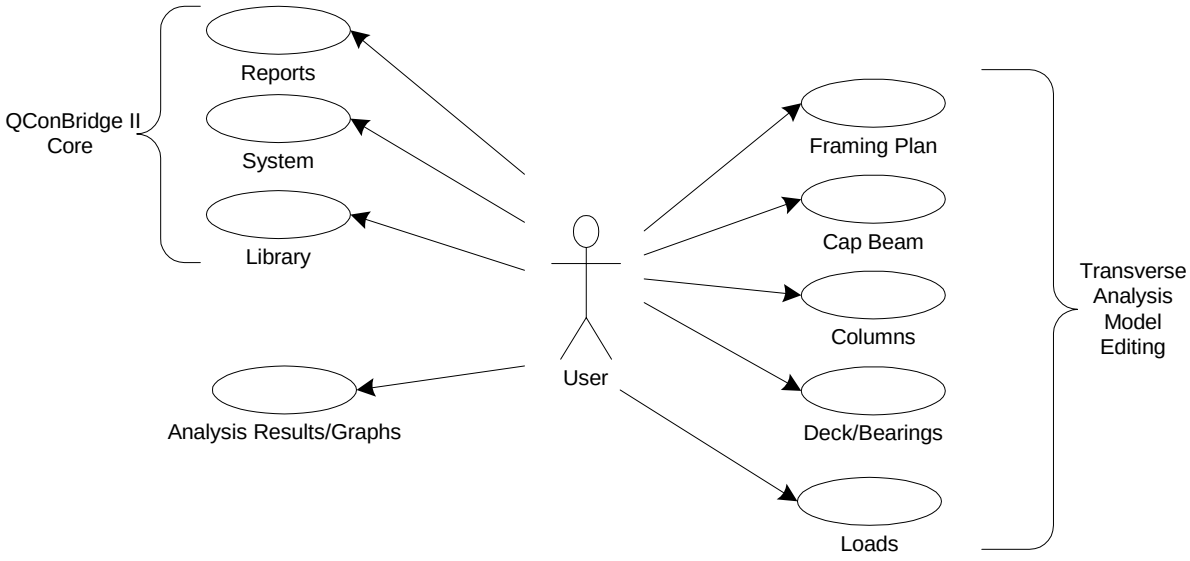


Figure 61 - Transverse Analysis Model Editor UI Actors

3.2.2.2 Analysis Model Editing Views

The Transverse Analysis Model Editor has only one editing view: the Pier View. The Pier View displays an elevation view of the pier model being edited. The editing operations available are presented in this section.

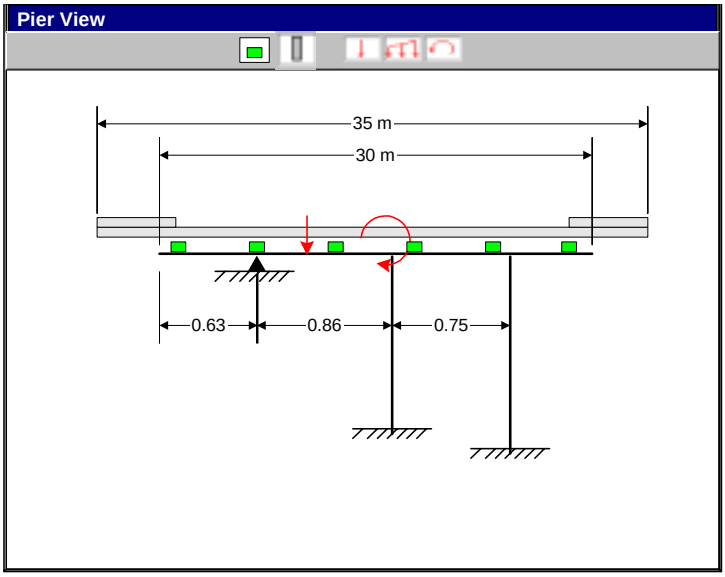


Figure 62 - Pier View

3.2.2.2.1.1 Keyboard Selection

Selections can be made using the keyboard by pressing the arrow keys. The left and right arrow keys are used to select the cap beam and columns. The selection sequence using the right arrow key goes as follows: cap beam → column 1 → column 2 → … → column n → cap beam. The order is reversed if the left arrow key is used.

The up arrow will jump the selection to user-defined loads. After the loads selection mode is enabled, the left and right arrow keys are used to change the selection. The down arrow key is used to jump into member selection mode.

3.2.2.2.1.2 Mouse Operations

The Pier View supports mouse interaction with many of the actors as specified in the following table.
Bearings are shown using ■ icons. Bearing locations can only be edited in stand-alone TBAM models.

Table 20 – Mouse Actions for Pier View

Actor	Action	Result
Cap Beam	Single Click	Select Cap Beam
	CTL/SHIFT-Click	Select cap – only one to select
	Double Click	Activate Pier Framing Plan dialog
	Right Click	Context menu
	Drag	⊗
	Ctl-Drag	⊗
Column	Single Click	Select – Activates a dimension line showing column length – this dimension can be clicked on and edited in-place.
	CTL/SHIFT-Click	Select multiple – dimension line disappears - Activates “Even Spacing Tool” on context menu
	Double Click	Activate Column Editing dialog
	Right Click	Context menu
	Drag	Moves column to the location of the column nearest drop location. The target column and columns in between the source and target column are shifted toward the source to fill the void. Column spacing is retained.
	Ctl-Drag	Copies source column into location between columns at drop location. Spacing on either side of new column is same as spacing at drop location. If column is dropped outside of column, it becomes an exterior column with the same spacing as the previous exterior column. Widens the cap beam accommodate new column. Cap beam extends to the left wrt girders and the roadway. Overhang dimensions with respect to the exterior columns are maintained.
Loads	Single Click	Select – Activates a dimension line location from end of cap or top of column – this dimension can be clicked on and edited in-place.
	CTL/SHIFT-Click	Multiple Select – dimension line disappears
	Double Click	Activate Load Editing dialog
	Right Click	Context menu
	Drag	Move load to new location. Positioning is based on cursor location at drop. If multiple loads are dragged, the "No" icon is displayed and a tooltip telling the user that only single loads can be dragged is displayed
	Ctl-Drag	Moves copy of load. See Drag for details.

Actor	Action	Result
Bearings	Single Click	Select – Activates a dimension line showing bearing location wrt left end of cap – this dimension can be clicked on and edited in-place.
	CTL/SHIFT-Click	Multiple Select – dimension line disappears
	Double Click	Activate Bearing Editing dialog with selected bearing highlighted
	Right Click	Context menu
	Drag	Moves bearing based on cursor location at drop.
	Ctl-Drag	Moves a copy of the bearing using the same semantics as Drag.

3.2.2.2.1.3 Copy and Paste Operations

Some of the actors in the Pier View also support Delete, Cut, Copy and Paste operations as detailed in the following table. The paste target is the item selected when a paste occurs. If the paste target is not valid, the Paste option on the Edit menu, toolbar, context menu, etc is disabled.

Table 21 - Copy and Paste Operations for Pier View

Actor	Action	Paste target	Result
Loads	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer
	Paste	Cap Beam	Paste load using location data for the load
		Bearings	Paste load to bearing
		Column	Paste load using location data for the load
	Delete		Delete
Pier Cap	Copy		Copy to paste buffer
	Cut		∅
	Paste	Pier Cap	Paste properties to paste target (in another running application).
	Paste Special	Pier Cap	Option to mirror properties
	Delete		∅
Column	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer (can't delete last one)
	Paste	Column	Paste properties to paste target
		Pier Cap	Paste a new column on the right hand side of the pier using the right-most column spacing. If the new column is beyond the end of the cap beam, the cap beam is extended, maintaining the current column/cap overhang.
	Paste Special	Same as Paste	Option to mirror properties
	Delete		Delete (can't delete last one)
Bearings	Copy		Copy to paste buffer
	Cut		Remove and copy to paste buffer (can't remove last one)

Actor	Action	Paste target	Result
	Paste	Cap Beam	Paste a new bearing on the right hand side of the pier using the right-most bearing spacing. If the new bearing is beyond the end of the cap beam, the cap beam is extended, maintaining the current bearing/cap overhang.
	Delete		Delete (can't delete last one)

3.2.2.2.1.4 In-Place Editing

The Pier View shows dimensions for cap length and column locations. These dimensions may be selected with the mouse, which changes the cursor to a text cursor and allows in-place editing.

3.2.2.2.1.5 Context Editing

These commands will become available on the context menu and main Edit menu if an applicable item is selected.

3.2.2.2.1.5.1 Mirror Properties

This command will mirror the stiffness properties if a cap beam or a column is selected. Can also mirror the entire pier if all are selected.

3.2.2.2.1.6 ToolBox

Across the top of the Pier View window is a palette (toolbox) for icons and widgets applicable to the pier view only. The purpose of the toolbox is to provide an interactive means for editing the pier analysis models. Editing features include addition of columns, loads, and bearings to the analysis model. All items are placed near the drop location.

3.2.2.2.1.7 Even Spacing Tool

This tool becomes available on the context menu when multiple, adjacent columns are selected. A dialog appears that asks for a single value: the spacing between selected columns.

3.2.2.2.2 MOUSE BEHAVIOR WITHIN VIEWS

3.2.2.2.2.1 Dynamic Cursor

In QConBridge II, the mouse cursor changes dynamically depending on the program state and what the mouse is hovering over.

Table 22 - Dynamic Mouse Icons

Mouse Icon	Mouse is Hovering over:
	Normal – over white space
	Loaded cursor – cursor is over a selected item
	In drag mode – cannot drop
	Cap Beam
	Columns

Mouse Icon	Mouse is Hovering over:
	Bearings Load

3.2.2.2.2.2 Gravity Well Around Bearings

When dragging point loads on structure, the bearings shall act as a “gravity well” which snaps the load to the exact bearing location. Loads that are attached to bearings will move with the bearings and show a red dot on the bearing when the load is selected.

3.2.2.2.3 VIEW SETTINGS

All views will have the capabilities to turn dimensions off and on, and to overlay member area or moment of inertia information in a graph-like style

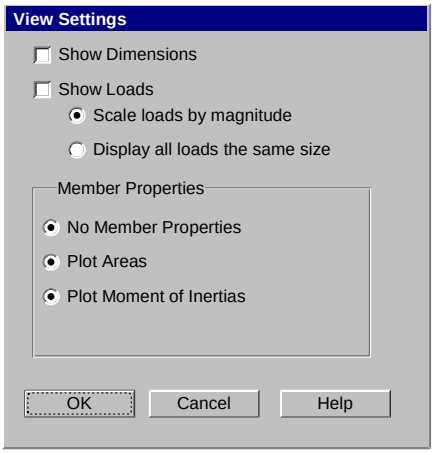


Figure 63 - View Settings

3.2.2.3 Bridge Contractor

The Bridge Contractor is a wizard used to edit bridge and create QConBridge II projects. The Bridge Contractor is reentrant for all TBAM project types.

3.2.2.3.1 STEP 1

Step 1 is seen for all types of projects. It allows you to choose the project type and system-level properties for a project. Note that the Number of Spans selection is disabled for a transverse model.

The project type on this dialog is available only for new projects. The type of an existing project cannot be changed.

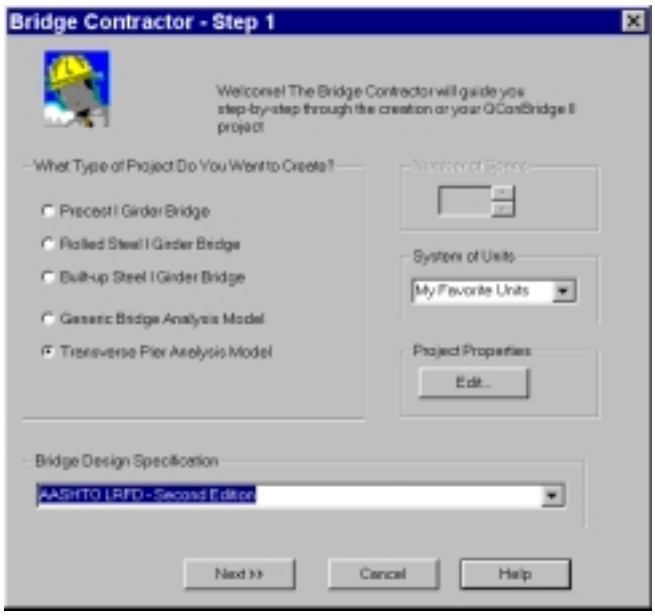


Figure 64 - Bridge Contractor Step 1

3.2.2.3.2 STEP 2 – ANALYSIS SETTINGS

See Section 3.2.2.4.1 below

3.2.2.3.3 STEP 3 – DEFINE PIER CAP AND BEARING LAYOUT

Step 2 brings up a dialog similar to Figure 66. For a new file, values are initialized to a 20 ft pier cap with two simple-support columns.

3.2.2.3.4 STEP 4 – DEFINE DECK

In Step 3, we define the roadway width and sidewalk widths. This step is identical to Figure 67.

3.2.2.3.5 STEP 5 DEFINE COLUMN LOCATIONS

In this step, we define the locations of the columns and their stiffness properties. This step is similar to Figure 68.

3.2.2.4 Detailed Editing for Actors

This section describes the primary dialogs for editing each of the major UI actors for TBAM editing. Access to these dialogs is either through context menus from the main views, or by menu selections.

3.2.2.4.1 SETTINGS

Settings affect the way the model is analyzed.

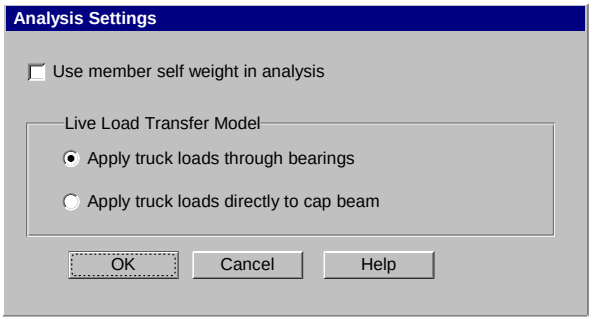


Figure 65 - Analysis Model Settings

3.2.2.4.2 PIER FRAMING PLAN

The Pier Framing Plan is consists of three tabs and is the primary editing interface for piers. The first tab provides for description of the cap beam and roadway, the second tab is used to define the roadway with respect to the pier, and the third tab is used to describe the column layout.

3.2.2.4.2.1 Cap and Bearing Layout

This dialog is used to define the cap length, stiffness and bearing layout. At least on bearing must be present on the structure.

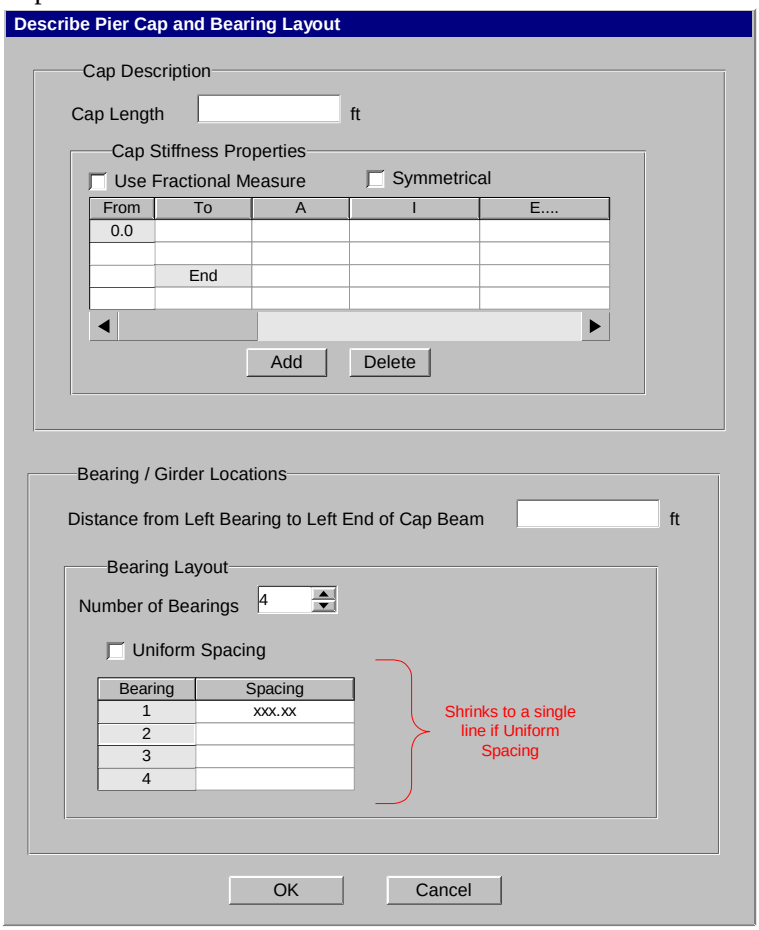


Figure 66 – Pier Cap and Bearing Layout

3.2.2.4.2.1.1 Cap Stiffness Properties

Cap stiffness properties are as follows:

Table 23 - Cap Stiffness Properties

Property	Description
A	Area
I	Moment of inertia
γ	Weight Density
E	Elastic Modulus

3.2.2.4.2.2 Tab 2 Roadway and Deck Description

Roadway widths must be defined at the pier in order to perform the transverse live load analysis. Deck overhangs are measured to the centerlines of exterior girders.

Negative deck overhangs are not permitted.

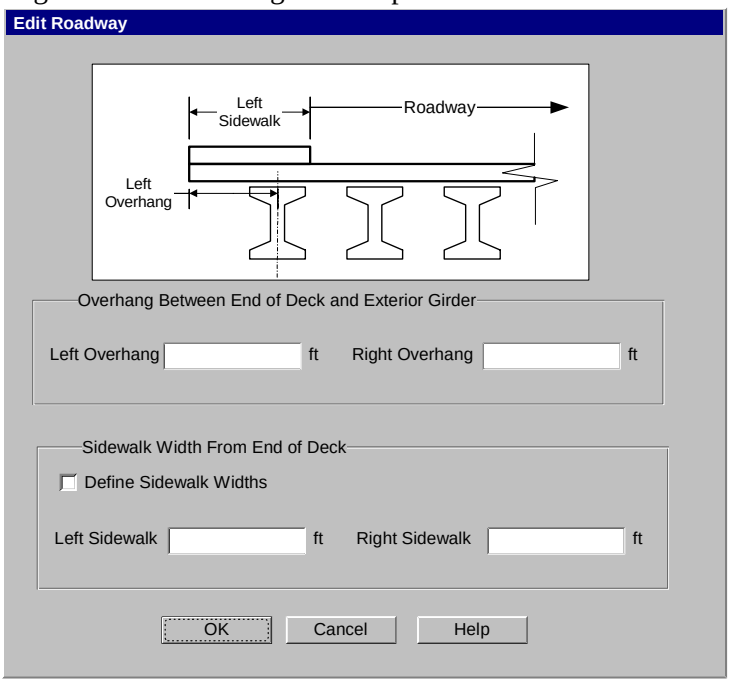


Figure 67 - Roadway Widths

3.2.2.4.2.3 Tab 3 – Column Layout

This tab is used to layout the column locations. Columns may be zero-length (Simple Support), or fixed-height – in which case section properties must be described

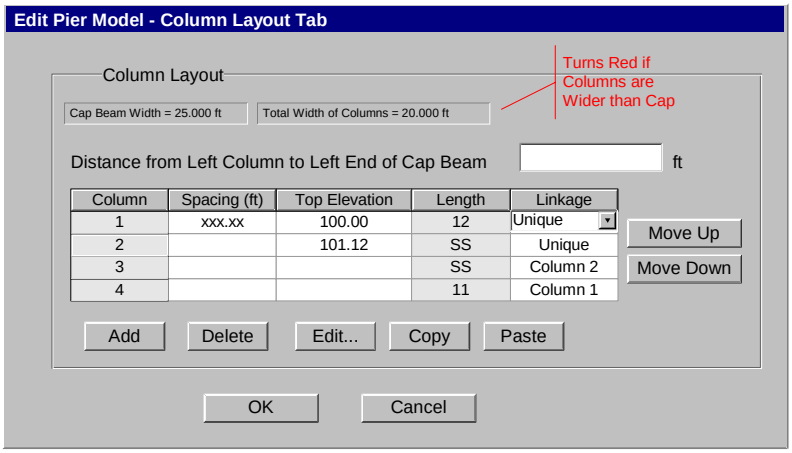


Figure 68 - Column Layout

3.2.2.4.3 COLUMNS

Individual columns in a pier model may be edited as follows. Simple-support columns do not have column length or property descriptions.

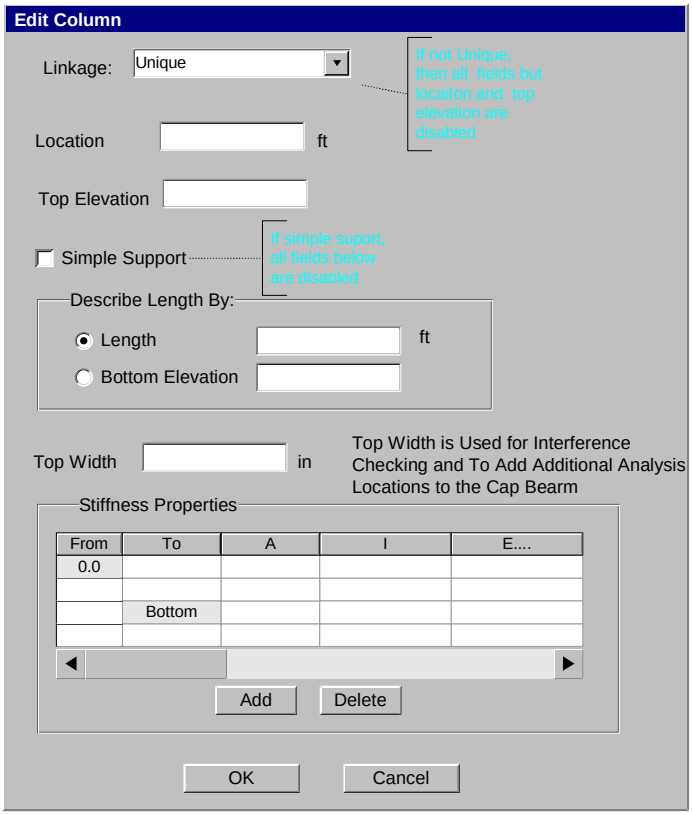


Figure 69 - Edit Columns

3.2.2.4.3.1.1 Column Stiffness Properties

Column stiffness properties are as follows:

Table 24 - Column Stiffness Properties

Property	Description
A	Area
I	Moment of inertia (in plane of bent)
γ	Weight Density
w	Additional wt/length

3.2.2.5 Load Editing on Transverse Models

The subsections that follow describe the various facilities available for managing user-defined loads applied to pier models.

3.2.2.5.1 STAGING

TBAM's have no concept of staging. When a TBAM is in a product model, loads can only be applied to them in the final stage of the analysis.

3.2.2.5.2 COORDINATE SYSTEMS – LOAD LOCATION

Loads may be applied to the cap beam, columns, and bearings. Loads that are applied to bearings are applied at the bearing location and may not be offset. Loads that are applied to the cap beam are located from the left end of the cap beam, with the positive direction going left-to-right. Loads

that are applied to columns are located from the top of the column (connection between cap beam and column), with the positive direction going from top-downward.

3.2.2.5.3 SUPERSTRUCTURE LOADS

This section describes how loads that represent reactions from the superstructure are imparted onto the substructure model.

3.2.2.5.3.1 Dead Loads

Dead loads are applied as concentrated vertical loads at the bearing locations.

3.2.2.5.3.2 Live Load Application

Live loads are applied to the pier structure using a lane reaction. This reaction is calculated from the results of a longitudinal analysis. If the TBAM is part of a product model or BAM project, this value is automatically supplied by the system and cannot be entered manually. Hence the dialog shown in is only available for editing in TBAM-based projects.

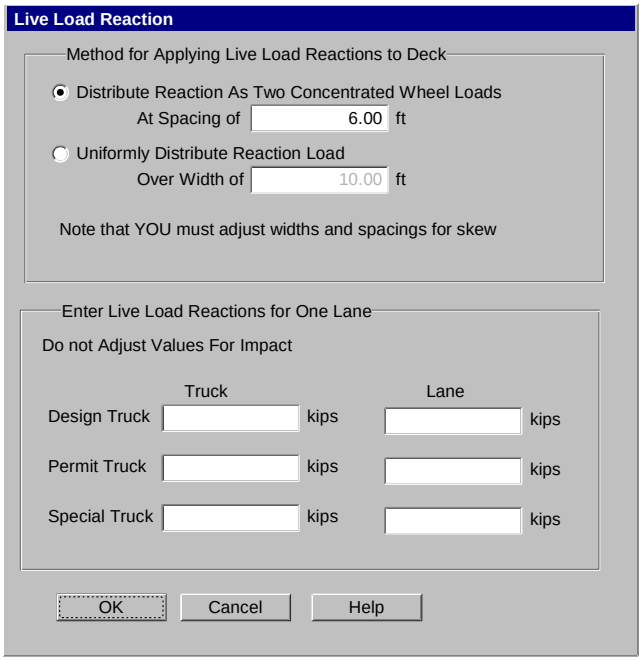


Figure 70 - Live Load for TBAM

3.2.2.5.4 ADD/EDIT LOAD

3.2.2.5.4.1 Manual Method

In the manual method, loads are applied to the currently selected actor. To add a load select Loads | Add *type* Load or right click on the selected actor and select Add *type* Load from the context menu, where *type* is Point, Linear, or Moment. For either case the appropriate load input dialog is displayed to the user.

To edit a previously defined load, select the load and choose Edit | Properties. This will present the appropriate Load Input Dialog.

3.2.2.5.4.2 Interactive Method

Loads can be interactively added to a span by dragging an icon from the toolbox palette and dropping the icon onto the cap beam, a bearing, or a column. When the icon is dropped on an actor, the appropriate load input dialog is displayed.

The load icon must be dropped in the proximity of the actor. When the cursor is some distance away from any actors, the icon has the "no circle" superimposed on it, indicating that the cursor is not over a valid drop zone.

The user may cancel this drag and drop operation by either pressing the Esc key or by dropping the icon somewhere other than a valid drop zone.

Loads can be interactively edited by double clicking on the load, selecting the load and pressing the Enter key, or right clicking on the load and selecting Properties from the context menu. This will present the appropriate Load Input Dialog.

3.2.2.5.4.3 Load Input Dialogs

The Load Input Dialogs are used for creating or editing individual loads. The location of a load can be measured with an absolute distance or a fractional distance. The location of a load is measured from the left end of the horizontal member, or top of the vertical member, to which it is applied.

Vertical Loads with a positive magnitude are applied in the direction of gravity or, in the case of moments, counter-clockwise (moment vector out of the screen - positive Z direction). Transverse loads are applied in the positive X (left-to-right) direction.

All loads must be associated with a Load Group. Load Groups are either predefined by the Analysis Criteria library entry, or can be created on the fly for the current project.

3.2.2.5.4.3.1 Component Loaded

Each of the user-defined loads must be applied to a component in the TBAM. The following options are provided:

Option	Action Taken
Cap Beam	Applies load to cap beam
Column 1…n	Applies load to selected column
All Columns	Applies load to all columns in the model
Bearing 1…n	Applies load to selected bearing
All Bearings	Applies load to all bearings in model

Note that moments cannot be applied to bearings, so bearings will not be presented as an option in the moment editing dialog.

3.2.2.5.4.3.2 Point Load

The load input dialog for Point Loads is shown in Figure 71. A point load is defined by its location, direction and magnitude.

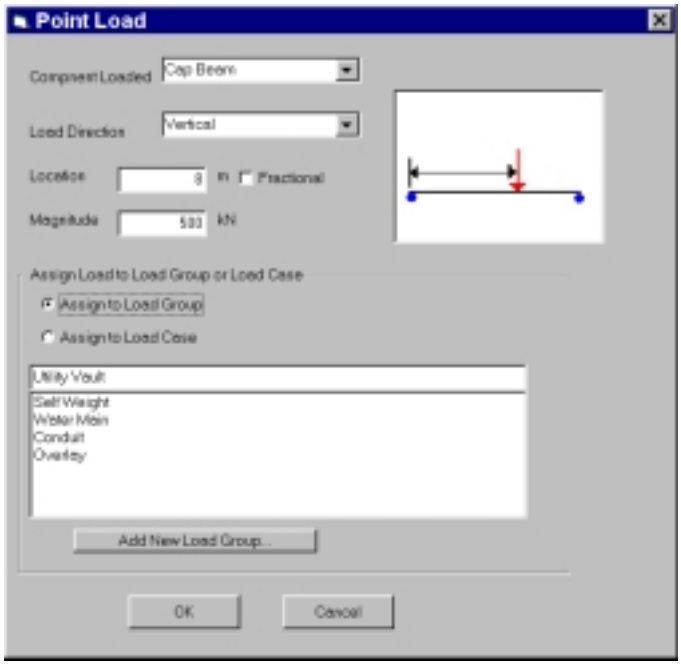


Figure 71 – Edit Point Load

3.2.2.5.4.3.3 Linear Load

The load input dialog for Linear Loads is shown in Figure 72. A linear load is defined by its starting and ending location and the intensity of the load at its starting and ending position. The user can select the Uniform Load option button to enter a linear load with a constant intensity.

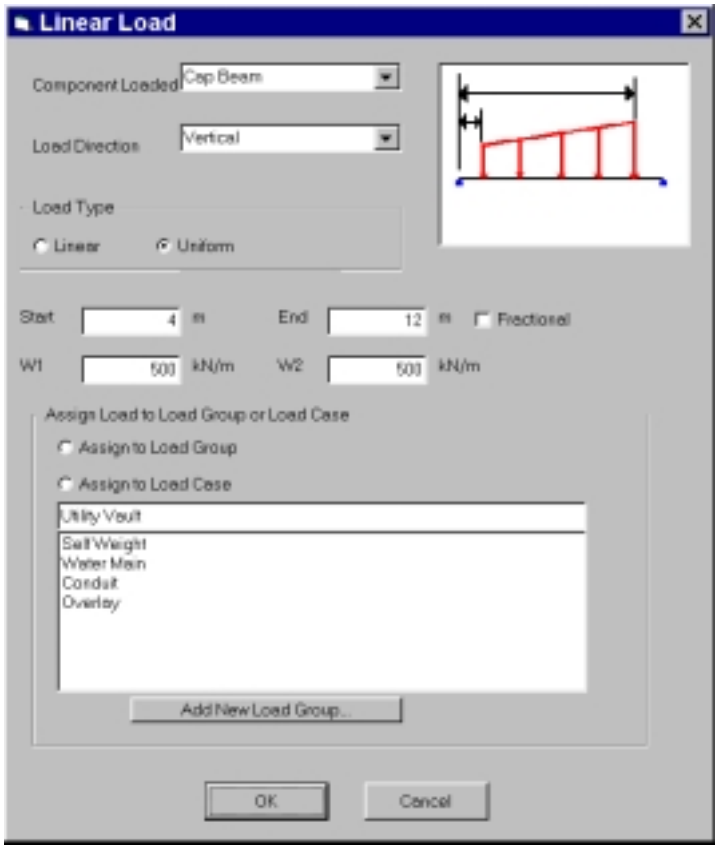


Figure 72 – Edit Linear Load

3.2.2.5.4.3.4 Moment Load

The load input dialog for Moment Loads is shown in Figure 73. A moment load is defined by its location and magnitude. Note that moments cannot be applied to bearings.

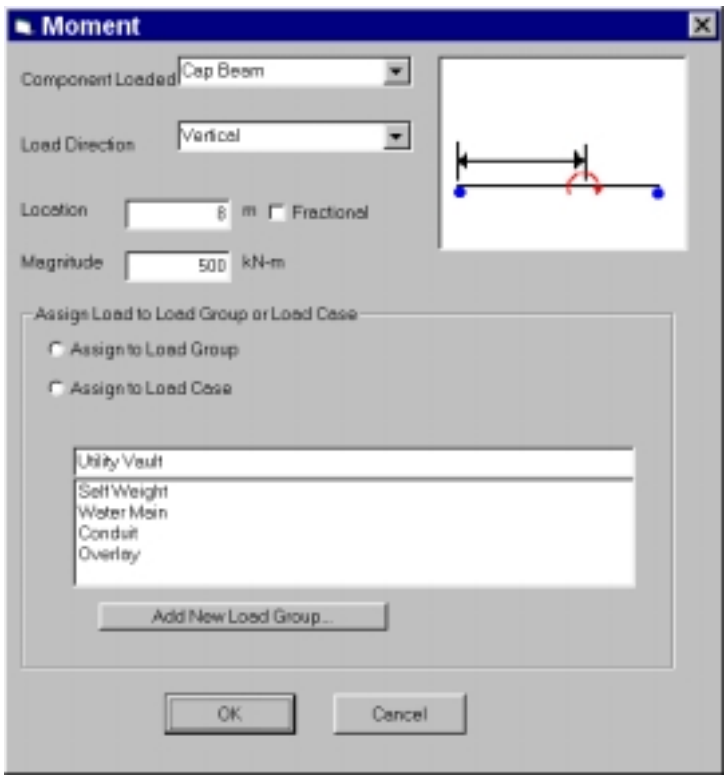


Figure 73 – Edit Moment Load

3.2.2.5.5 CUT/DELETE LOAD

Loads are removed from the analysis model by cutting or deleting them. Cutting a load places a copy of it on the clipboard for subsequent paste operations. Deleting a load removes it from the analysis model.

To cut a load, select the load and choose Edit | Cut, press the right mouse button and choose Cut from the context menu, or press Ctrl+X.

To delete a load, select the load and choose Edit | Delete, press the right mouse button and choose Delete from the context menu, press Ctrl+D, or press the Delete key.

3.2.2.5.6 COPY LOAD

The currently selected load can be copied to either the clipboard for a subsequent paste operation or copied to another actor by dragging onto it.

3.2.2.5.6.1 Copy to Clipboard

To place a copy of a load onto the clipboard, select the load and choose Edit | Copy, right click on the load and select Copy from the context menu, or press Ctrl+C.

3.2.2.5.6.2 Copy to Actor

A load can be copied to another actor by dragging it onto that actor. To copy a load, press and hold down the Ctrl key, grab the load with the left mouse button, drag the load to a different actor, and release the load onto the actor by releasing the left mouse button.

3.2.2.5.7 PASTE LOAD

Loads copied or cut to the clipboard can be pasted to any actor (cap beam or column) in the structure. To paste a load, select the target actor and choose Edit | Paste, right click on the actor and select Paste from the context menu, or press Ctrl+V. The load will be pasted onto the new actor at the same location (fractional or absolute) as the originating actor.

If a load is pasted onto another load, it is placed at the same location as the target load.

3.2.2.5.7.1 Pasting Semantics

Refer to the BAM loads reference Section 3.2.1.6 above for pasting semantics.

3.2.2.5.8 MOVE LOAD

A load can be moved to a different actor using the same method outlined in Section 3.2.2.5.6.2 above, except the Ctrl key is not used.

A load can also be moved or copied using Right mouse drag. Click with right mouse and drag the load to the desired span. When dropped, the user is presented with a context menu asking the user to choose Copy or Move.

3.2.2.5.9 SELF-WEIGHT LOADS

Self-weight dead load is computed as the dead load area times the unit weight of the material. Self-weight dead load is applied to the entire structure during all stages. Self-weight dead load is either generated or its not. To Enable/Disable the generation of self-weight dead load, choose the Analysis Settings dialog as shown in Figure 65.

3.2.2.5.10 LOAD DISPLAY

Loads can be displayed with a schematic or a scaled representation.

4 ANALYSIS RESULTS VISUALIZATION

This section describes the various elements of the Analysis Results View. The Analysis Results View is used to display analysis results along structural components or groups of structural components. Examples of singular structural components are girderlines, girders, pier cap beams, and columns. Examples of groups of components are piers and girderline analysis models. Viewing becomes especially complex when considering the large number of load cases, associated values, and loading configurations that can be involved in bridge structural analyses.

4.1 Analysis Results View Actors

The Analysis Results Viewer deals with the following UI Actors:

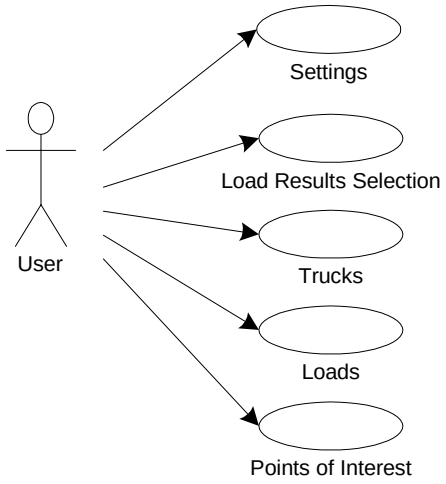


Figure 74 – Analysis Results View UI Actors

4.2 Analysis Results View

This section discusses the main views for analysis results viewing and the primary interactions in these views.

The Analysis Results View contains up to three panes for viewing analysis results. Each results pane contains a tree control on its left side for selecting the results to be displayed. Results panes are laid out in a hierarchy either from top-to-bottom or left-to-right, depending on the view layout (See View Settings). The primary view pane (either the top-most or left-most) is used to select the primary results. The other results panes display data associated to the primary view pane. Section 4.5 below contains a detailed discussion on the selection of analysis results for graphing. Figure 75 shows a schematic of the Analysis Results View.

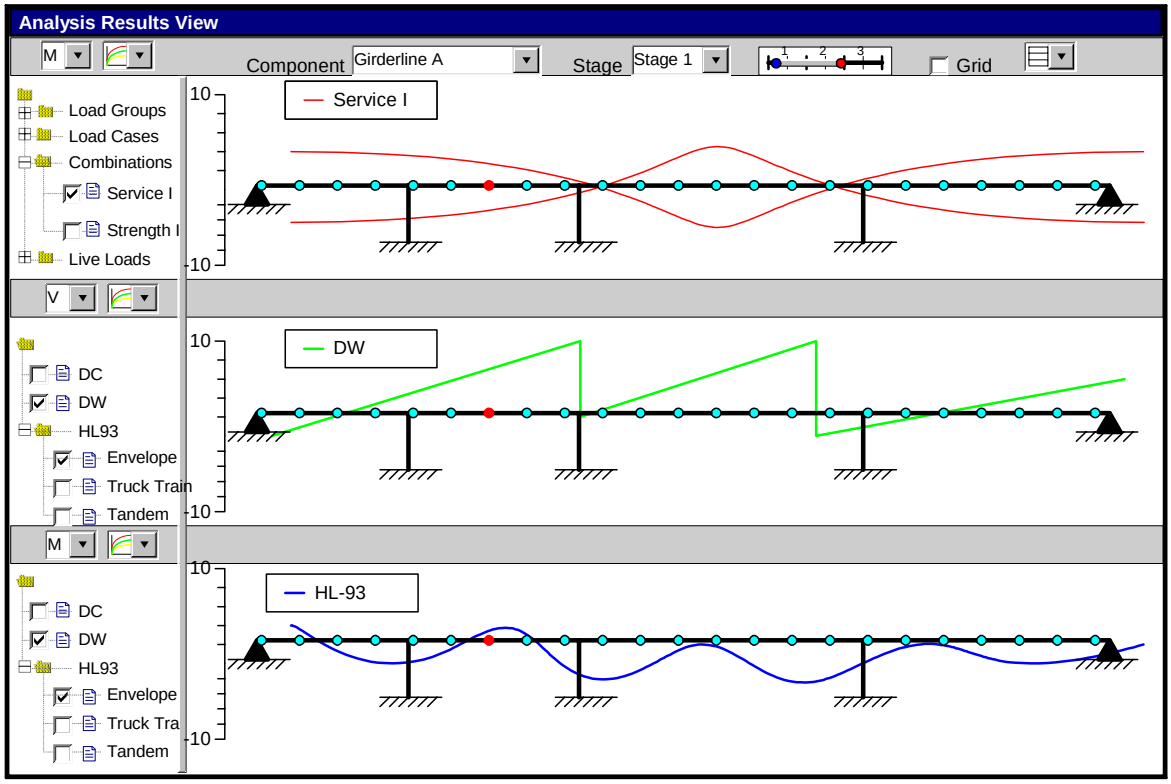


Figure 75 - Analysis Results View

4.2.1 Mouse Operations

The Analysis Results View supports mouse interaction as specified in the following table.

Table 25 – Mouse Actions for Analysis Results View

Actor	Action	Result
Points of Interest	Single Click	Select – synchronized in all panes
	CTL/SHIFT-Click	Select – no multiple
	Double Click	Select
	Right Click	Select
	Drag	⊗
	Ctl-Drag	⊗

4.2.2 Mouse Behavior Within View

4.2.2.1 Dynamic Display of Data

If the mouse is paused for over one second above a graph data point, or a POI, a tool tip box shall display appropriate data for that location.

4.2.3 Analysis Results Panes

Each analysis results pane contains a tree control for selection of the data to be viewed, a toolbar for selection of the action and graph type, and a palette onto which analysis results are painted.

4.2.3.1 Primary – Secondary Pane Relationship

The primary pane and main toolbox set the context for graphs and reports to be displayed in the secondary panes. For example, if moment results for self-weight are shown in the primary pane, only results associated with the self-weight load group can be displayed in the secondary panes. The secondary pane could show shears, reactions, and displacement.

4.2.3.2 Graph Type Selection

Graph types for a results pane are selected using the two combo boxes above the tree control. 

. The left combo box selects the action to be plotted. The choices are moment (**M**), shear (**V**), axial force (**A**), deflection (**D**), and reaction (**R**). The right combo box selects the graph type. Available graph types are Individual Results (), Stacked Results (), Overlaid Results (), and Tabular ().

Individual Results graphs allow multiple selections of load results within graphs. In fact, a user could select a limit state and all of its component loads to get an effect similar to selecting an Overlaid graph for that limit state.

Stacked Results and Overlaid graphs are available only for load results that are composites. Composite load results are load results that are composed from other load results. Composite load results available in QConBridge II are Limit States and Load Cases. These graphs automatically display all of the component load results for a composite in a single view. Stacked graphs show how the loads sum to create the composite case. Overload graphs show all load results using zero as a datum.

Tabular Results display one or more tables describing the load results in question. These, small tabular reports may be viewed, printed, and copied into the clipboard paste buffer. Like Individual Results Graphs, Tabular Results allow multiple load results to be selected.

4.2.3.3 Load Results Selection

Each analysis results pane contains a tree control used to select the load results to be displayed. The tree control in the primary (top or left) pane displays all possible load results that can be displayed for the current graph type. Tree controls in secondary panes display only load results that are associated with the selected result(s) in the primary pane. A complete discussion of the load selection hierarchy is given in Section 4.4.

4.2.3.4 Points of Interest Selection

Points of interest may be selected by clicking on them with the mouse in any pane or cycling through them with the left and right arrow keys. This causes the same point of interest to be selected in all panes in a synchronized fashion. If the graph on the pane is location-dependent, the graph will display results for that POI.

A POI must be selected at all times. By default, a POI is selected somewhere toward the middle of the structure in question.

4.2.4 ToolBox

Across the top of the Analysis Results window is a palette of widgets. This palette is called the Toolbox. The purpose of the toolbox is to provide an interactive means for specifying the results to be viewed. Items that can be specified are components, stages, and grid on/off.

4.2.4.1 Stage Selection

This combo box is used to select the stage for which analysis results are to be viewed. The combo box is not shown in TBAM projects, and is disabled when pier components are selected because TBAM's are not stage-dependent.

4.2.4.2 Span Zoom Tool

The Span Zoom Tool is present when girderlines are selected. It allows users to select which span(s) to be viewed.

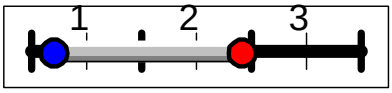


Figure 76 - Span Zoom Tool

4.2.4.3 Grid On/Off

This control controls the display of an optional grid that can be overlaid onto results data.

4.2.4.4 View Layout

The View Layout Selector  provides a shortcut method for defining the view layout. The different layout options are shown in Figure 77.

4.2.5 View Settings

Views will have the capabilities to display forces in global or element coordinates (only applicable to bent graphs), turn component dimensions off and on, display applied loads for load groups, and to specify the layout of the view screen. The screen may be split into panels as shown in the Figure. This dialog is available from the context menu.

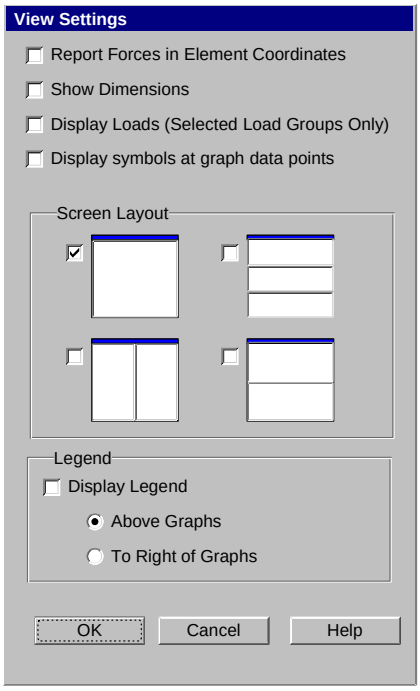


Figure 77 - View Settings

4.2.5.1 Legends

Legends that identify selected load results shall be shown in all results panes. Legends may be displayed either above graphs, or to their right. Figure 78 shows an example of a legend that would be placed to the right of a graph.

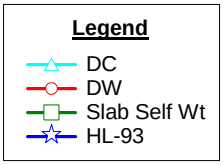


Figure 78 - Graph Legend

4.2.5.2 Symbols

Graphs must be able to show graph lines in color and to optionally display symbols at data points. Data symbols and line colors are not customizable by the user.
Colors in the legend will be identical to colors used to display load cases and groups in other views in the application.

4.3 Presentation of Graph Data

The Analysis Results View can plot data for both bent frame (transverse piers, or girderlines with piers), or line elements (girderline only, columns, etc.). The ways graphs are presented for these types is discussed in this section.

4.3.1 Bent Frames

Bent frames have beam elements that can intersect at any angle. This means that results plots on bent frames cannot utilize a common orthogonal scaling axis making it necessary to create separate graphs for each straight beam segment. Figure 79 shows an example of a shear diagram for a bent frame-type structure. Forces in bent frame structures are plotted in global coordinates.

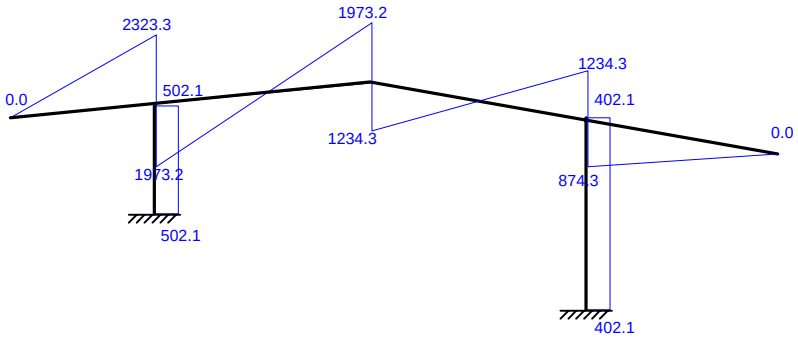


Figure 79 - Bent Frame Graphs

Optional labels showing data values at significant points along can be placed on the graph. Labels may be placed at the following locations along beam elements:

Ends
Points of maximum and minimum values

In complex graphs, collisions can cause data labels to become unreadable. The drawing algorithm for data label placement shall be written to minimize data label collisions.

4.3.1.1 Overlay of Load Data on Bent Frames

Bent frames may also optionally be able to display user-defined loads on the structure. Loads display may be turned on or off via the View Settings Dialog. Loads will be displayed using the same color scheme for load groups as other views in the application.

4.3.2 Line Elements

Line elements represent linear segments of a structure. Line elements are simpler to provide x-y plots for because scaling axis' can be easily placed onto plots to provide scaling for plotted values. The program shall also be able to place an optional grid over the plots.
Cap beams can be represented as line elements even if they are not straight.

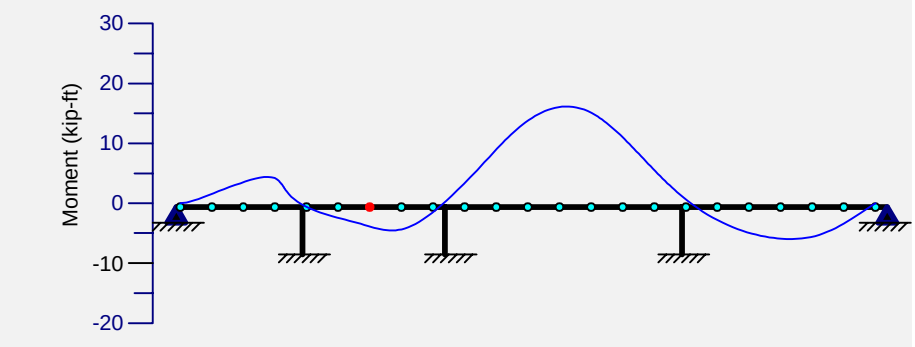


Figure 80 - Results Graphing for Line Elements

4.3.3 Tabular Reports

The Analysis Results View shall be able to display the values associated with any graph in tabular, printable form. Only the data for the currently displayed graph is printed. The text in Tabular data reports shall also be selectable using the mouse cursor, or by choosing Select All in a context menu, or Ctrl-A from the keyboard. Selected text can be copied to the Windows Clipboard.

4.4 Load Results Hierarchy

The analysis results for a load combination are typically dependent on hundreds of other calculations involving load groups, load cases, live loads, distribution factors, and the load combination process. Figure 81 shows the dependency of load results for QConBridge II in a graphical format. Most programs make if difficult, if not impossible to easily perform detailed calculation checks on this type of data. In QConBridge II, we organize these dependencies using tree controls to help users understand them and easily select and visualize associated and dependent results for the loading in question.

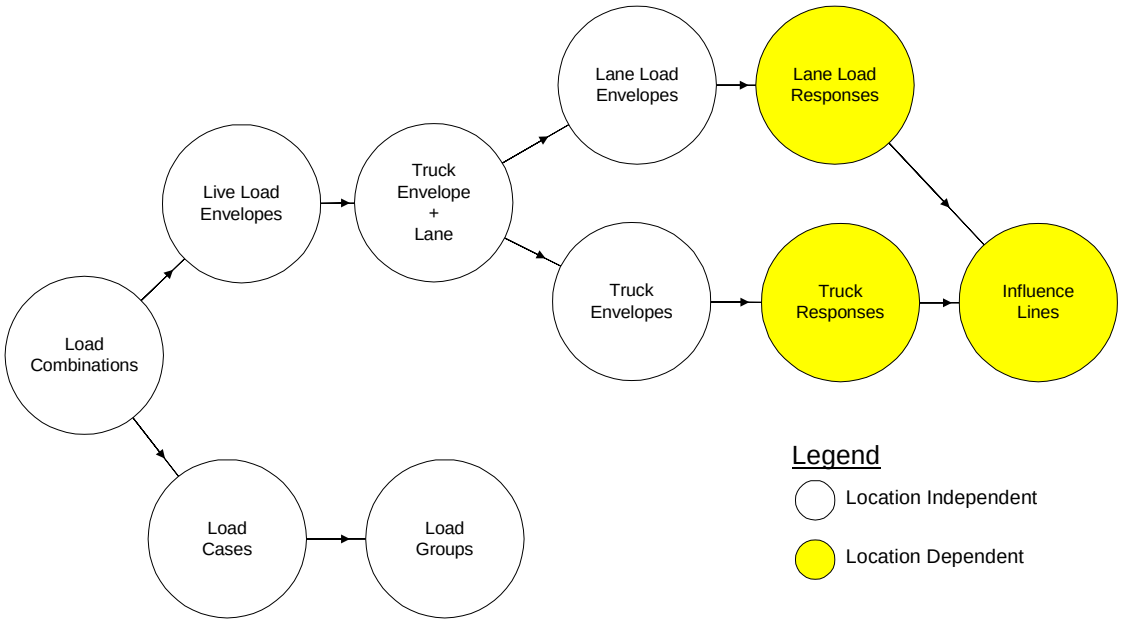


Figure 81 - Load Results Hierarchy

The table below provides a description for each of the Load Results types.

Table 26 Descriptions of Load Results

Load Result Type	Description
Load Combination	Factored combination of Live Load Envelopes and Load Cases. Load Combinations are dependent on all other results types. Examples are LRFD Limit States
Load Case	Sums of Load Groups. Examples are DC, DW
Load Group	Lowest-level static load results. Have no dependents. Examples are girder self weight and slab self weight.
Live Load Envelope	Envelope of Truck+Lane Envelopes
Truck + Lane Envelope	Combination of Lane Load Envelopes and Truck Envelopes
Lane Load Envelope	Envelope of lane load responses for all POI's
Lane Load Response	Lane response for a selected POI. Lane location is overlaid onto graph. Must select for max or min.
Truck Envelope	Envelope of truck response for all POI's
Truck Response	Truck response maximized for a selected POI. Graph shows truck positioned on structure and label denoting truck description and location. Must select for max or min.
Influence Line	Unit load influence line for selected POI.

As mentioned earlier, the view panes in the Analysis Results View are laid out in hierarchical fashion. The tree control in the top-most (or left-most view) is used to specify the primary load selection. The other view panes can only display load results associated with the primary selection. This helps to manage the number of possible permutations of load results that can be displayed simultaneously. As a rule, moment, shear, displacement, and axial force diagrams can be displayed for all load results.

For example, if a moment diagram for Girder Self Weight load group is selected in the primary window; then the other views can only display shear, displacement, and axial force diagrams for Girder Self Weight. This is because Girder Self Weight is not dependent on any other load results. The most complicated example comes when a Load Combination is selected. In this case, all load cases and live loads that were involved in the calculation of the limit state can be selected in the secondary results panes.

4.5 The Load Selection Tree Control – In Depth

Each results pane in the Analysis Results View contains a Load Selection Tree Control (LSTC) on its left side. Each LSTC displays all of the load results that can be graphed based on the loads available in the current project file. LSTC's display load results in a hierarchical fashion as discussed in Section 4.4 above. The load hierarchy in the primary pane is displayed in a flattened out manner to simplify load selection for the user.

4.5.1 General Behavior

Figure 82 shows a typical LSTC. Categories of load results are shown as folders ( ). Load results available for display are shown as documents ( ). More descriptive icons can, and probably should, be used in the finished application.

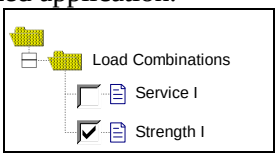


Figure 82 - Load Selection Tree Control

Load results are selected by clicking the left mouse button on the adjacent check box. Multiple load results can be selected by CTL/SHIFT-Click. If no load results are selected, the accompanying graph or report pane will read "No Loads Selected".

4.5.2 Primary Pane Control

The primary results pane is always the top-most or left-most pane in the Analysis Results View. The LSTC in this pane allows selection of all available load results for the current project. Load results in the primary pane control are organized into folders as shown in Figure 83.

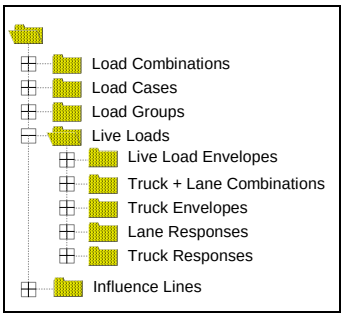


Figure 83 - Primary Pane Load Selection Tree Control

4.5.3 Secondary Pane Controls

Secondary pane LSTC's only display load results that are dependents of the load results selected in the primary LSTC. Depending on the load result selected in the primary pane, the secondary pane is capable of displaying all of the folders in the primary LSTC except for Load

Combinations, which are not dependents for any other type of load result. In addition, secondary LSTC's will show all load results selected in the primary LSTC as top-level documents. Note that if envelopes are selected in the primary LSTC, then two types of load responses can be displayed: other enveloped actions, or action values associated to the selected envelope.
For example, Figure 84 shows a secondary LSTC when the primary LSTC has the Service I combination selected. Note that not all possible load results are shown (user-defined load cases and groups could exist), however this should give a good idea of how load results are laid out.

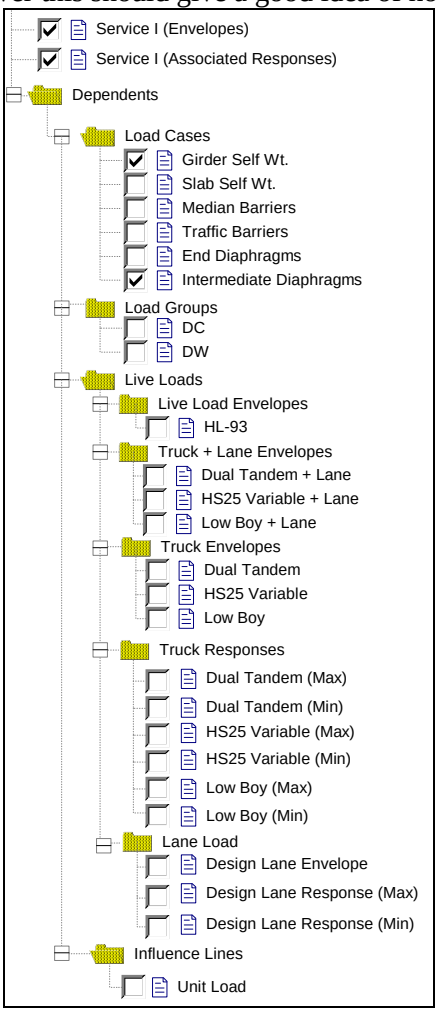


Figure 84 - Secondary Pane Load Selection Tree Control (Service I load shown selected)

If no load results are selected in the primary LSTC, secondary LSTC's are shown as empty trees.

4.6 AutoCalc Mode

The Analysis Results View shall be compatible with QConBridge II's AutoCalc Mode. It must be capable of updating itself automatically when AutoCalc is turned on, and able to display itself in "License Plate Mode" when AutoCalc is turned off.
If analysis results cannot be obtained, the Analysis Results View is disabled and a message is shown in one of the panes that explains why analysis results could not be computed. This is similar to PGSuper's functionality.

4.7 Printing and Plotting

Windows has one default printer for all applications. Our experience with QConBridge I has been that this is a problem. WSDOT engineers generally want force results plots on 11x17 paper, in landscape orientation, but there default printer is generally set to 8 1/2 x 11 portrait for printing documents. Windows does not allow users to setup a default printer configuration on a per application basis. To compound matters, within the context of QConBridge I, WSDOT engineers want to print reports on 8 1/2 x 11 portrait paper and print forces results on 11x17 landscape paper. It is very inconvenient to have to remember change the printer settings for reports and force result plots. To address this problem, QConBridge II will support the notion of Printing and Plotting.

Printing is the standard printing that comes with Windows. For QConBridge II, it can be used for either reports or force result plots, but is generally intended to be used for reports.

Plotting is the same as printing, except that it is generally intended for plotting force results diagrams. In QConBridge II plotting can be configured independently printing. This allows the user to quickly print force diagrams in a format that is different from the format used for reports.

4.7.1 Symbols

On black printers, data points symbols will be provide on graphs so that data can be discriminated.

4.7.2 Scaling

When printed, graphs shall be optionally printed to a standard US or SI engineering scales. The program shall calculate the “best” scale for the given graph size. The user will also have the option of turning off "best fit" scaling. In this case, the force results graphs will be drawn as large as possible.

4.7.3 Tabular Reports

Printing of tabular reports shall be supported.

4.8 Use Cases

4.8.1 Load Group Selected – User Wants Stacked Graph

Stacked graphs show dependency of loads for a single selected load case. Load Groups are independent of other loads – stacked graph shows only a single curve.

4.8.2 Stacked Graph currently in view – User attempts to select multiple loads

Stacked graphs may only be displayed for a single load selection. If the user attempts to CTL-Click another load while in stacked graph mode, the program will silently switch to Individual graph mode and display the new load results along with those previously selected.

4.8.3 Specific Primary Load is Selected – User Selects New One

In the case where the user selects a new primary load result, the secondary graphs will attempt to retain their current load selections, but only retain them if they are dependents to the newly selected primary results. If the new primary results are completely unrelated to the old secondary results, the secondary view will unselect all results and display “No Results Selected”. - Either this, or display the first available option for the secondary graphs.

4.8.4 New Specification Selected

If a new specification is selected, the graph will attempt to display its current selected data. If it cannot, it will return to its default, initial configuration.

5 REPORTS

This Section Describes the user interface for accessing and viewing reports in QConBridge II.

5.1 Report Selection

The available reports for QConBridge II are enumerated in the *QConBridge II System Design* document. Exact layout of all reports will be determined at the time of report creation.
The selection of available reports is dependent on the type of the current project. Some reports are component-dependent while others are applicable to the entire project. Some reports are also stage-dependent. Each type of report has one or more chapters that can be optionally displayed.

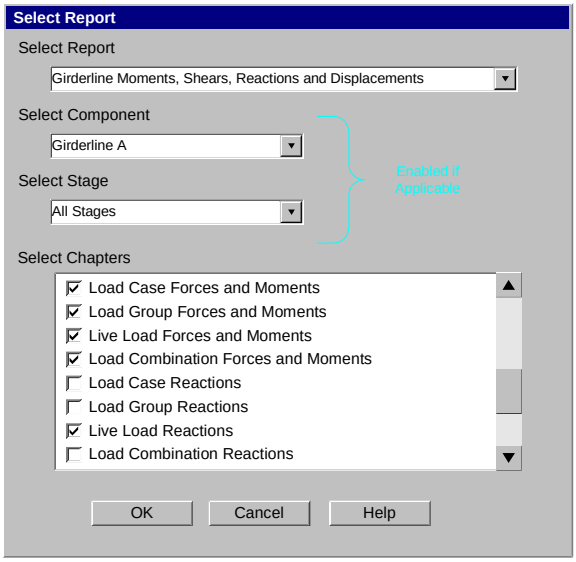


Figure 85 - Select Report Dialog

5.2 Viewing Reports

Report Viewing in QConBridge II will be identical to PGSuper. As with PGSuper, QConBridge II will have context menu support for selecting report text, printing, and chapter navigation.

5.3 AutoCalc

The QConBridge II report view windows will support AutoCalc. AutoCalc support will be identical to PGSuper's.

6 LIBRARY SYSTEM EDITOR

The Library Editor for QConBridge II will be nearly identical to the library editor used for the present version of PGSuper. The primary difference will be the new library entries for QConBridge II. Some small differences will also be present because we now have library entries that are dependent on the existence of other library entries (e.g., Specification entries depend on Loading Definition entries). The only effect that this will have on the editor interface will be a dialog that will explain that entries cannot be deleted if another entry depends on them. A icon will be used next to the entry to show that it is dependent on another.

6.1 Library Editor View

The Library Editor View is shown in Figure 86. This view has the same mouse, keyboard, and menu behavior as the Library Editor in PGSuper.

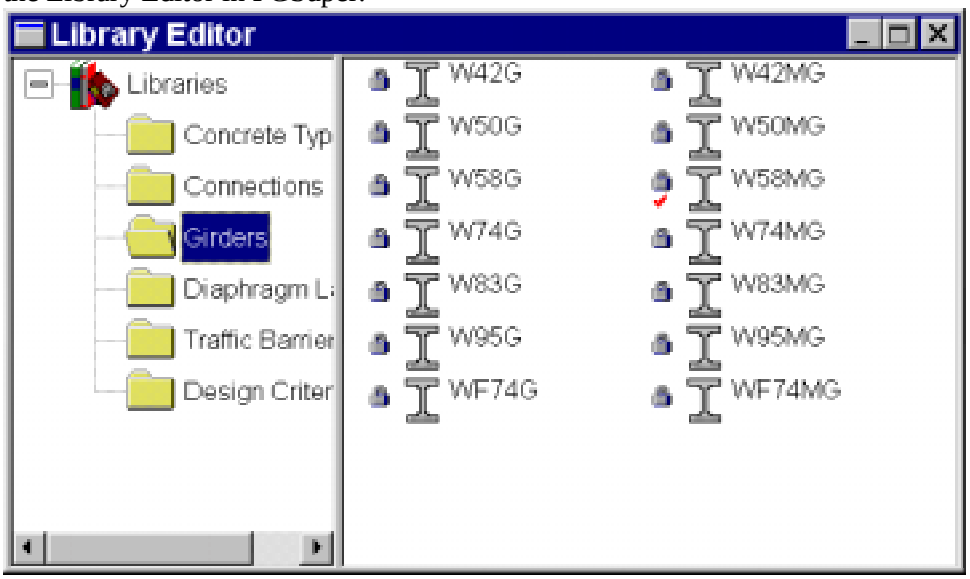


Figure 86 - Library Editor View

6.2 Library Entries for QConBridge II

The following dialogs are the editing interfaces for the various library entry types.

6.2.1 Girders Library Entry

The Girders dialog provides an interface for describing pre-defined girders for Precast Concrete and Rolled Steel girder product models. Girder selection is dependent on the type of product model project under consideration. Two types of product models use library girders: precast concrete girder bridges, and rolled steel girder bridges. It makes no sense to describe built-up I girders using library entries because they are unique for each project.

6.2.1.1 Precast I Girders

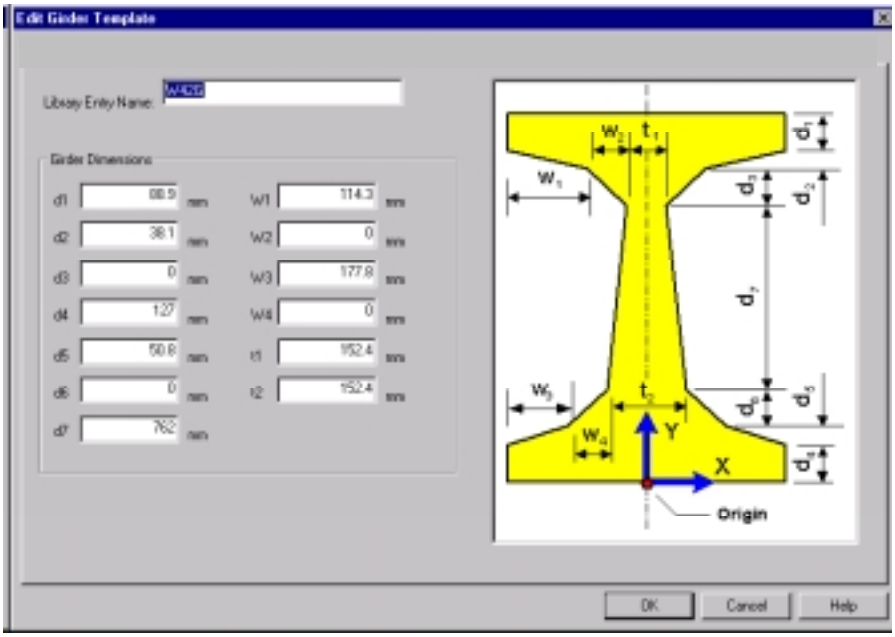


Figure 87 Girder Description Dialog – Precast I Girder Dimensions

6.2.1.2 Rolled Steel Girders

The system allows input of custom rolled steel girders. Standard AISC W and S shapes will be pre-loaded into the library.

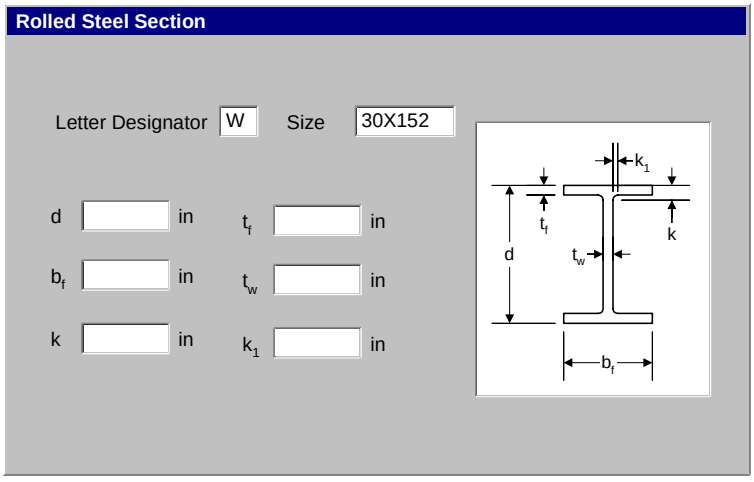


Figure 88 - Edit Rolled Steel Entry

I think we have to input all of the section properties as well. I don't think they are directly computable from the dimensions (probably something related to the Pittsburgh Mafia).

6.2.2 Concrete Library Entry

The Concrete Library Entry describes concrete material properties. Concrete is described by a name, 28 day compressive strength, density for strength calculations, density for weight calculations, and maximum aggregate size.

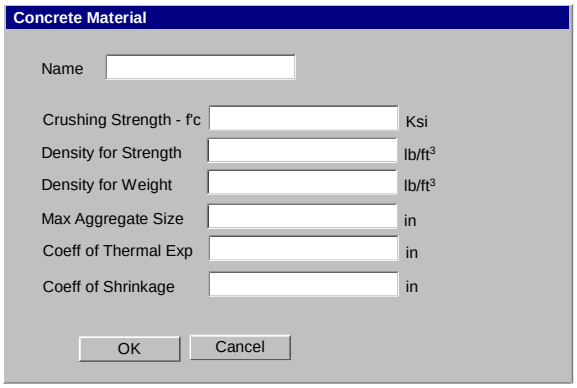


Figure 89 Concrete Properties Dialog

6.2.3 Steel Plate Library Entry

This entry provides entry for standard grades of steel plate and standard thicknesses for each grade. The Max. Available Width field is optional and may be used to later check constructability issues.

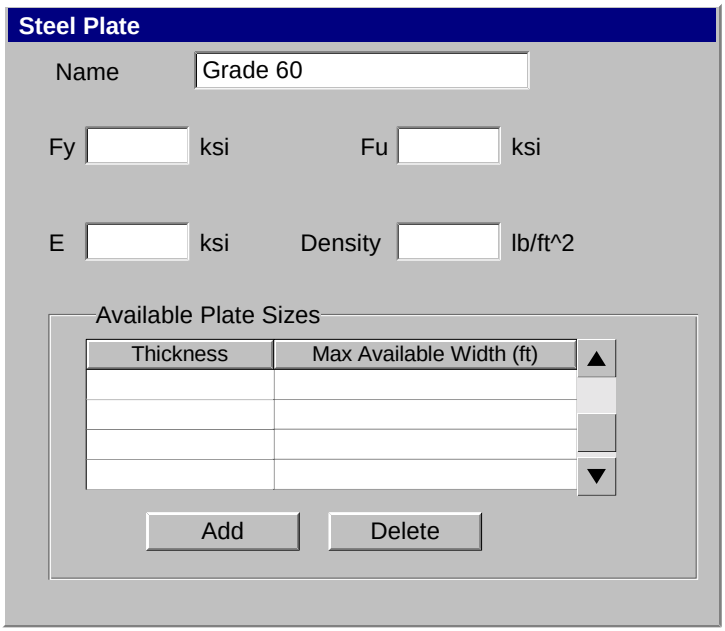


Figure 90 - Edit Steel Plate Entry

Need input for coefficient of thermal expansion.

6.2.4 Traffic Barrier Library Entry

The Traffic Barrier Library Entry dialog provides an interface for describing traffic barriers. The user has an option to include the stiffness of the diaphragms for the live load deflection analysis.

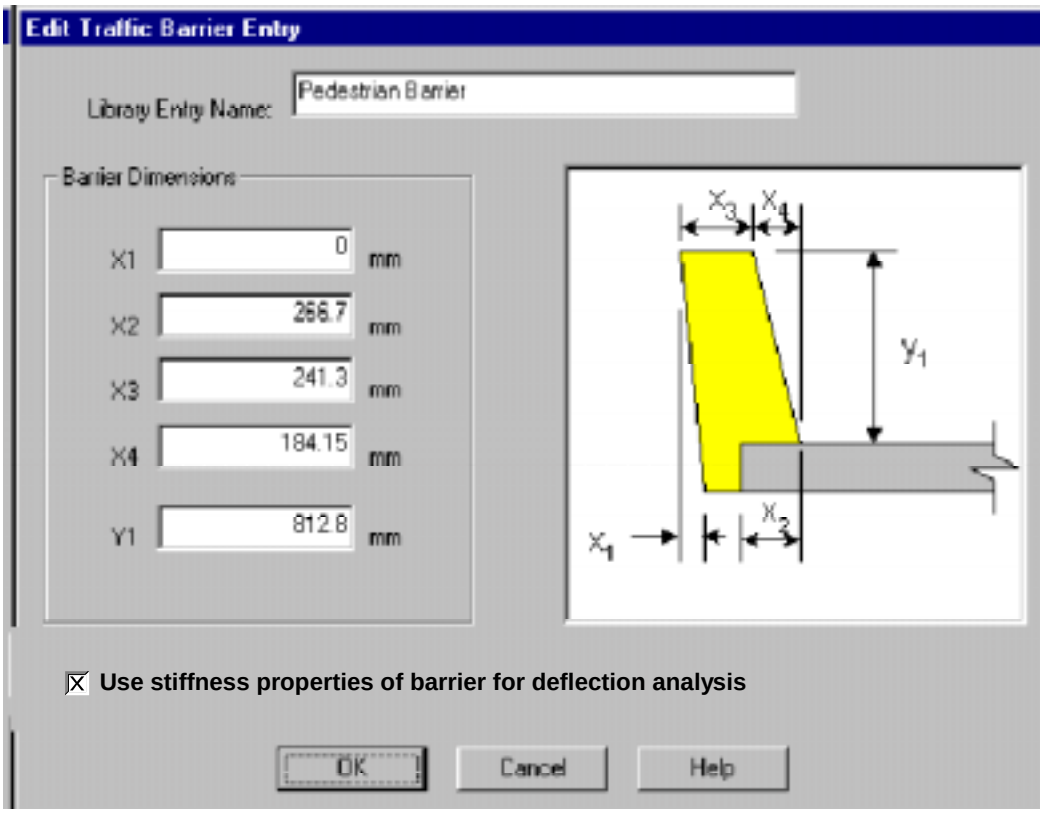


Figure 91 Traffic Barrier Dialog

6.2.5 Median Barrier Library Entry

The Median Barrier Library Entry provides an interface for describing median barriers.

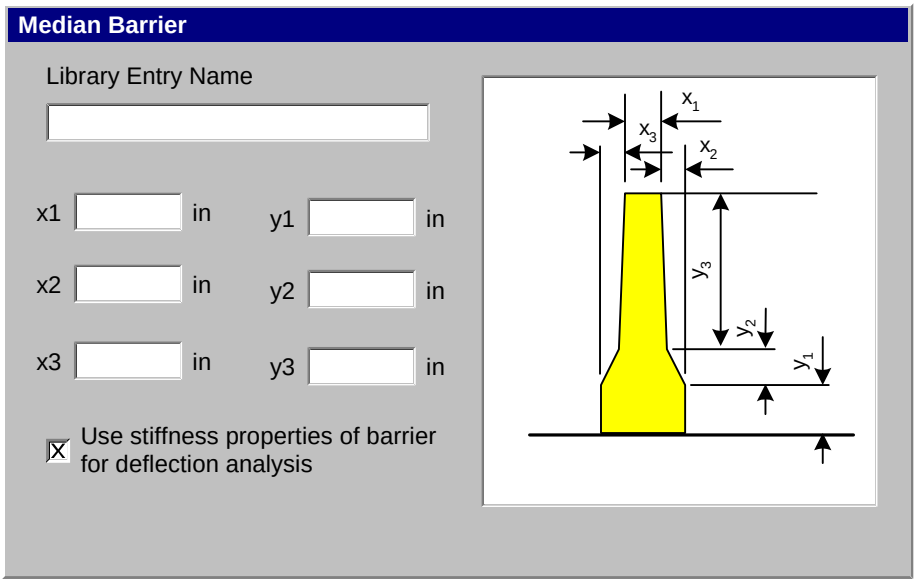


Figure 92 - Edit Median Barrier Entry

6.2.6 Connections Library Entry

This entry is a two-tabbed dialog. The first tab describes the geometry of the connection between the girder and its supports and the connection boundary conditions. Icons used to describe connection boundary conditions are described in detail in the Analysis Model Editor specification. The second tab describes the end diaphragm and how its weight is applied to the support. Connections can be either integral, or standard slab-on-girder bearing connections.
Minimum Edge Distances are used for checking that the cap has adequate width to accommodate the bearing.

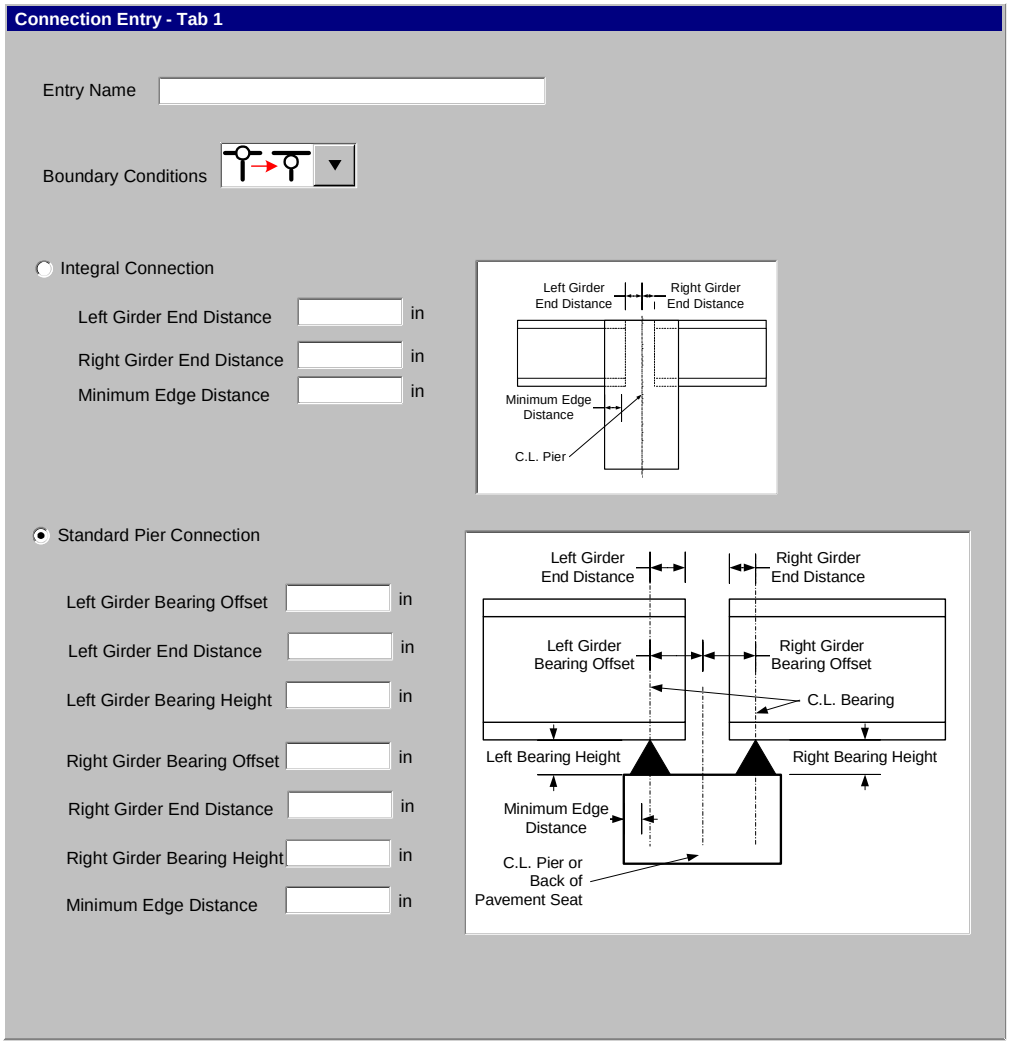


Figure 93 Girder Connection Dialog – Tab 1

Tab 2 is used to define the end diaphragms and how slab loads and loads transferred through the slab are applied at a connection. Note that the diaphragm height field is disabled for Integral Connections.

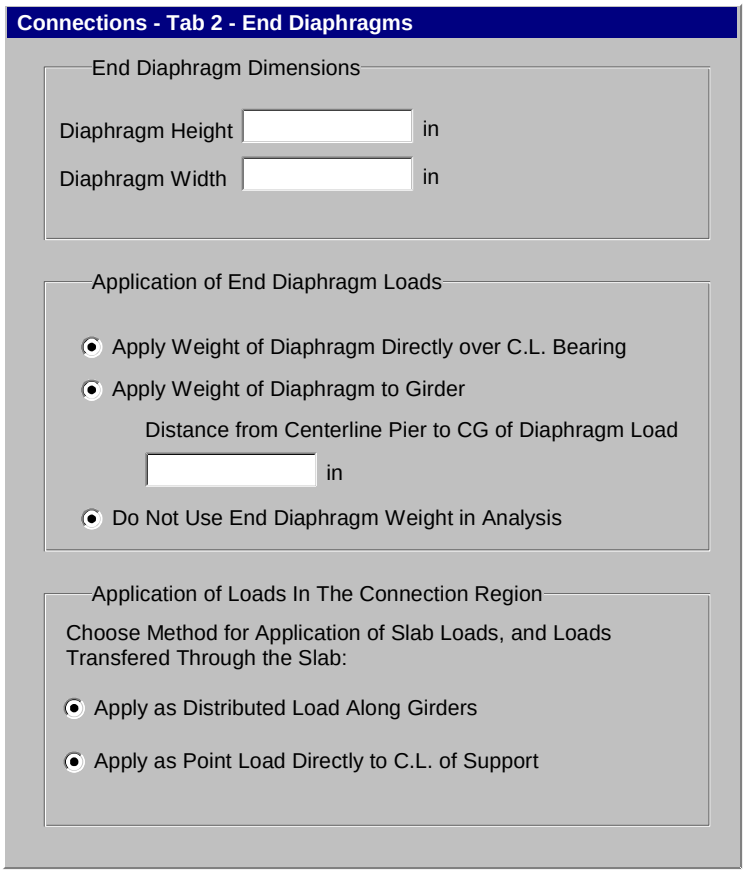


Figure 94 - Girder Connection Dialog - Tab 2

For end diaphragms, the end distance load location is used only for abutments. The load is always applied at the centerline of the support for pier connections.

6.2.7 Intermediate Diaphragm Layout Rules

The Diaphragm Layout Rules entry describes the logic for laying out intermediate diaphragms. Diaphragms can be laid out using fractional distances or exact distances.

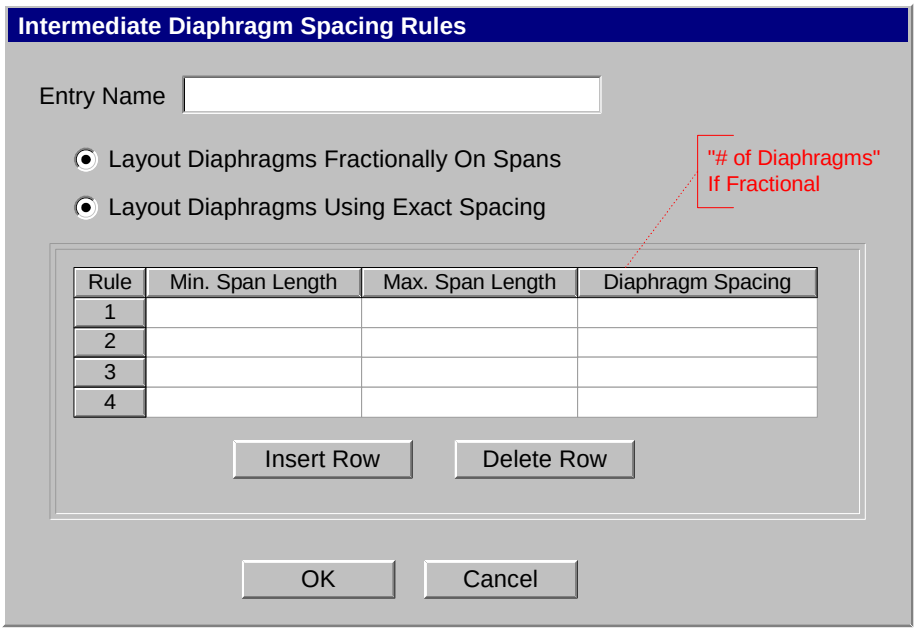


Figure 95 Diaphragm Layout Dialog

The starting length of rule N is always the ending length of rule N-1.

6.2.8 Design Criteria Library

The design criteria library is used to contain specification-related and design-related information that can be configured in QConBridge II. It is not meant to be a catalog of all specification parameters, but contain those parameters most likely to be modified by WSDOT engineers.

6.2.8.1 Description Page

This page provides input for a name and description for the design criteria entry. The user must also select a specification upon which to create this new design criteria.

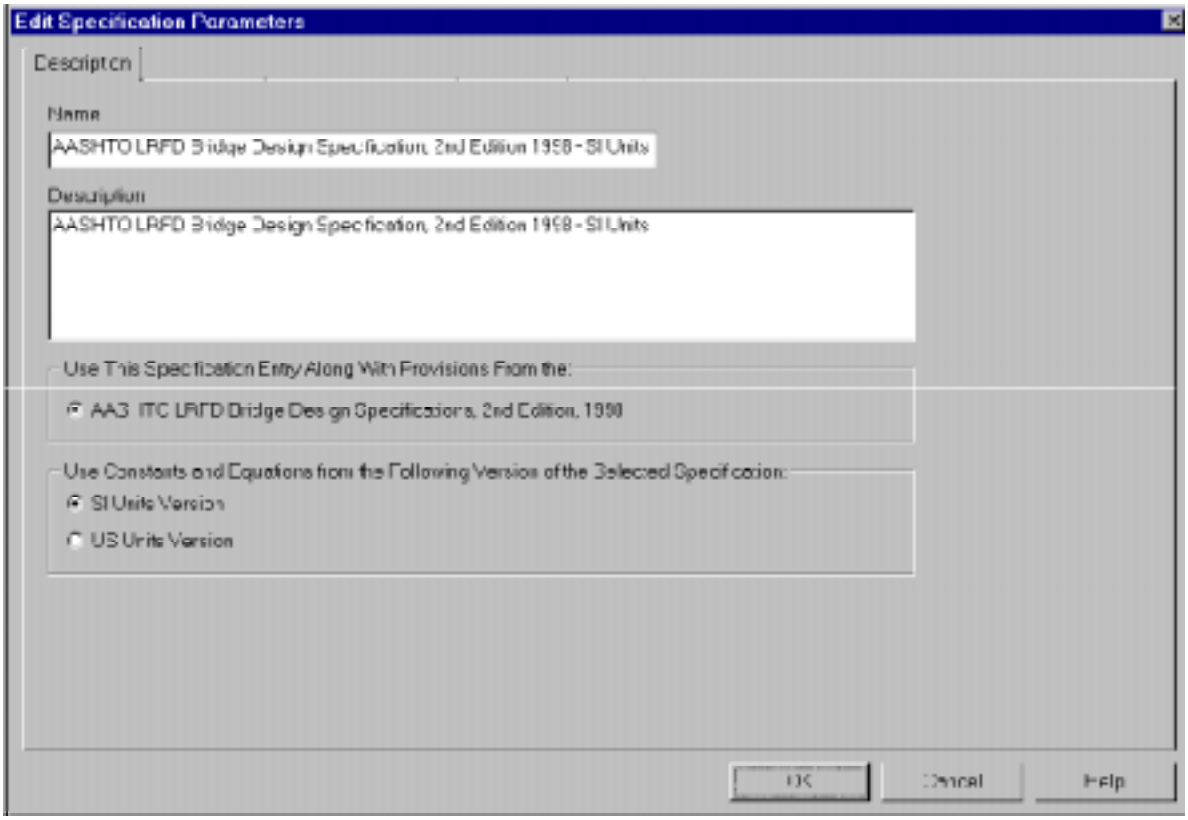


Figure 96 Design Criteria Dialog - Description Page

This tab allows you to name the specification entry and provide a description for it. You may also select the version of the AASHTO LRFD Specification that this specification entry will augment.
Specification Entries are not meant to describe all provisions. They are only meant to augment the specification for those provisions use in QConBridge that are likely to be agency-specific, or for provisions that are not defined in the Specifications.

Entry Name

Each specification entry must have a unique name. The name may contain any characters and can be up to 32 characters in length.

Description

Provide a description for the specification entry.

Use this Specification Entry along with Provisions from…

Check the radio button item to select the version of the LRFD Specifications that you would like to use along with the provisions described by the current specification entry. Note that the versions that may be selected are unit dependent. Hence, selection of an SI version of the Specifications means that SI-specific constants will be used when evaluating specification equations.

6.2.8.2 Loading Criteria Page

This page describes the loading criteria needed by QConBridge II.

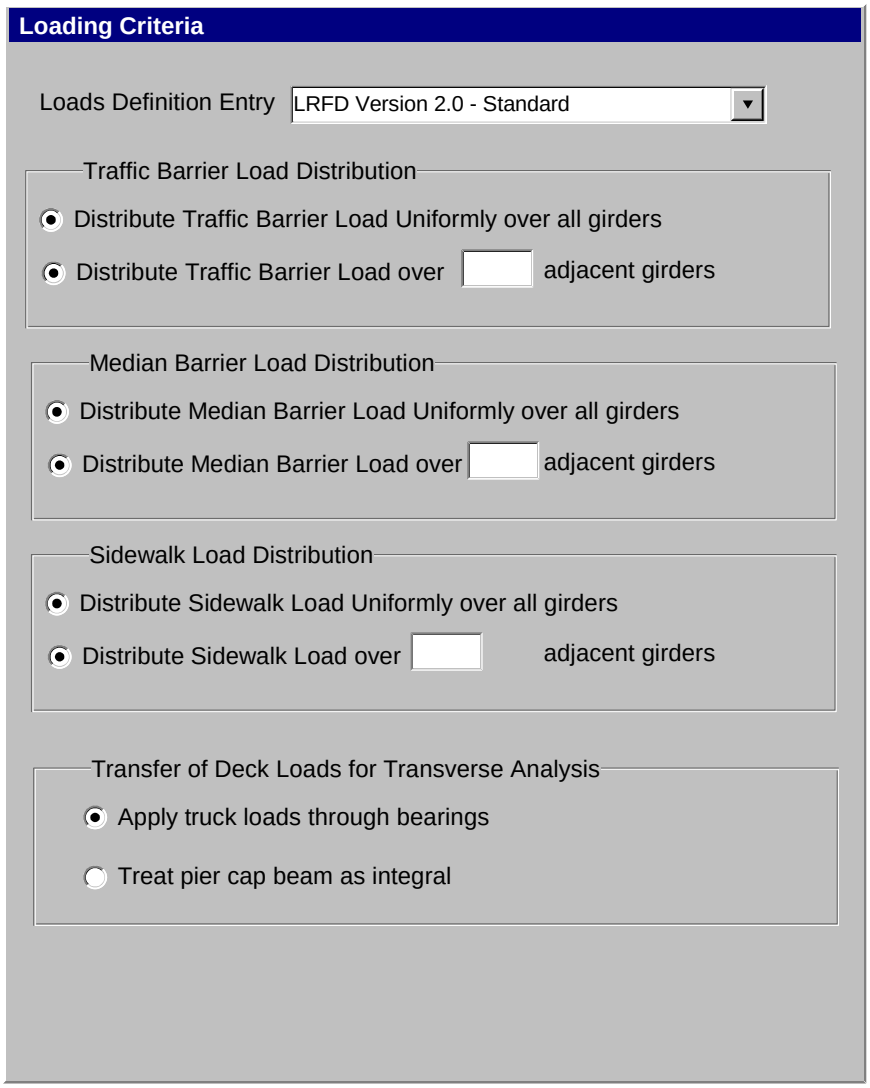


Figure 97 Design Criteria Dialog - Loading Criteria

6.2.8.3 Miscellaneous Page

This page describes miscellaneous design criteria.

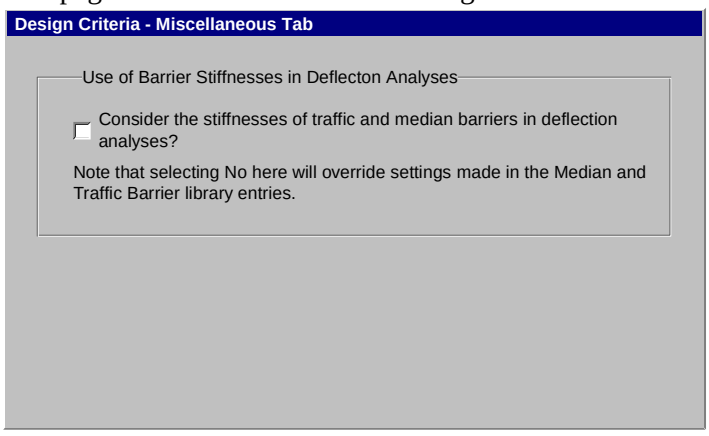


Figure 98 - Design Criteria - Miscellaneous Tab

6.2.9 Loads Definition Library Entry

This library is used to define categories for built-in loads, user-defined loads, and load combinations. This definition is sequential by nature, so a wizard is used as the user interface.

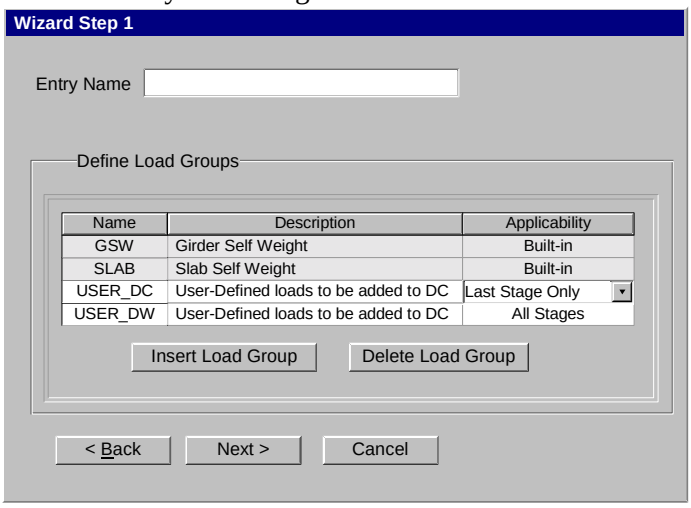
6.2.9.1 Wizard Step 1 – Define Load Groups

Load Groups come in two forms: built-in loads provided by the application, and containers for raw, user-input loads. Users can define their own Load Groups for user-input loads. Load groups can have three different types of applicability: Built-in, Last Stage Only, or All Stages. As implied, Built-in load groups are an integral part of the application. They cannot be deleted or renamed. , Last Stage Only applicability means that loads can only be added to the group for the final stage of analysis. All Stages applicability means that loads can be added to the load group for any analysis stage.

Each entry must have a unique name, which is used to identify it. Names have no length limit but should be kept short so they don't hog too much space in legends and load group selection dialogs.

Load groups defined in the library are copied into each project. Users can also define project-specific load groups to meet their needs while entering data for a project. Load groups added on the fly are always "All Stages".

Refer to the System Design Document for a list of Built-in Load Groups.



6.2.9.2 Wizard Step 2 – Define Load Cases

In this step, we create our load cases and assign load groups them. Each Load Group must be assigned a load group ID and a Description. A warning message is displayed if a load case is created that does not include any load groups. Load case editing is performed using Load Case Editing Dialog (Figure 99).

Load Cases are collections of load groups. As with Load Groups, there are two different types of load cases: user-defined and built-in.

The built-in load cases include all of the load cases available in Table 3.4.1-1 in the LRFD Specifications. Built-in load case names cannot be redefined. Many of the built-in load cases are not assigned any load groups by default. In this case, no results for the load case are reported.

In addition, the following live load cases are built in as well: "HL93:, "Fatigue", "Permit" and "Special". "HL93" and "Fatigue" represent live loads that are predefined by the system. "Permit" and "Special" are selected by the user within a project. If a load group is assigned to one of these cases, its results are simply added to the envelope of live load results.

You may add your own load cases in order to report results that you don't want to see added to the LRFD limit states, or as a step in defining your own custom load combinations.

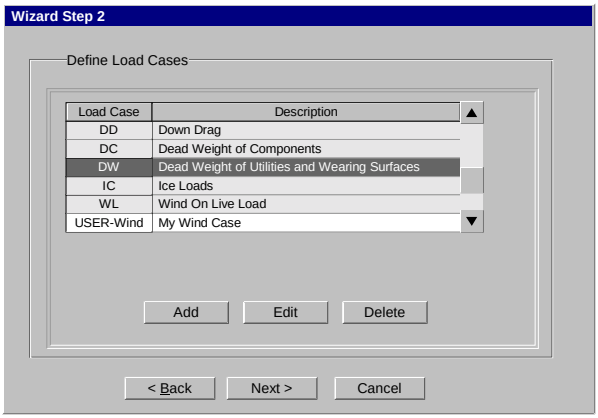


Figure 99 - Define Load Cases

6.2.9.3 Wizard Step 3 – Define Load Combinations

This dialog is used to define load combinations and the load factors to be used in the combinations. Min/Max load factors are entered using an nnnn.nnn/nnn.nnn format.
Load combinations are designated for one of two purposes: force calculation or deflection calculation.
The input grid has a column for all Load Cases defined by the system including live load cases.
All of the load combinations in Table 3.4.1-1 of LRFD are built-in and cannot be modified. These load combinations are activated by selecting the "Use Standard LRFD load combinations" check box. Users can then add additional load combinations to the list.

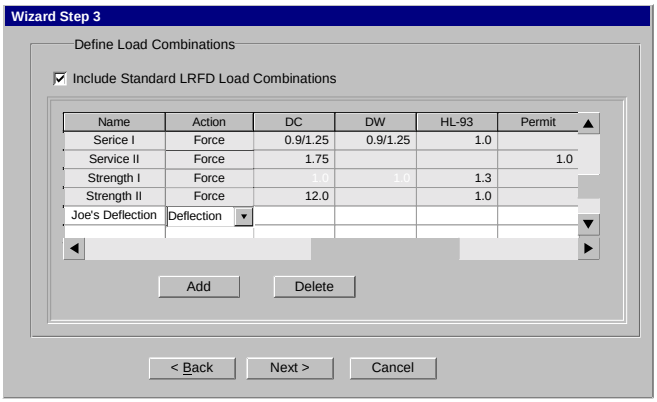


Figure 100 - Edit Load Combination

6.2.9.4 Load Case Editing Dialog

This dialog is used to select the load groups to be put into the current load case. Note that it is possible to use a load group in more than one load case. If this occurs, a warning is issued.

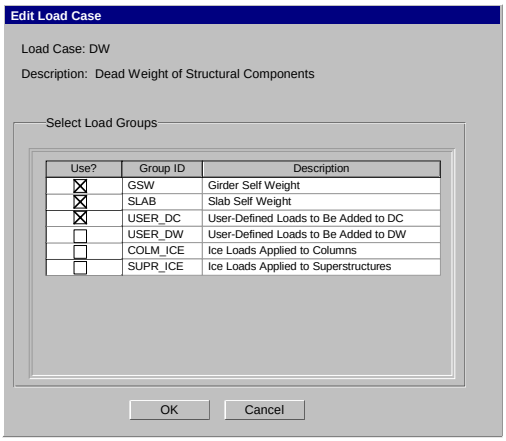


Figure 101 - Edit Load Case

6.2.10 Truck Definition Library Entry

Truck definition entries are used to describe a single truck and lane combination that can later be applied either as a “Permit” or “Special” truck for a specific project.
Variable axle trucks can be entered by specifying the axle spacing in “min – max” format. Only one variable spacing can be specified.

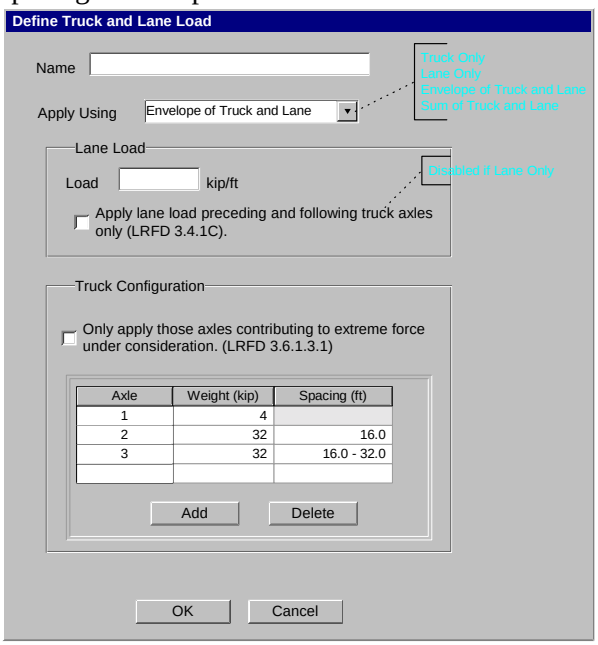


Figure 102 - Truck Definition Library Entry

6.2.11 Unit Categories Library Entry

Unit categories are used to allow users to customize the display of units in the QConBridge II GUI and in Reports. Physical Quantity types and their associated units are built in to the system. This dialog allows users to associate the units that they would like to see for given physical quantities.

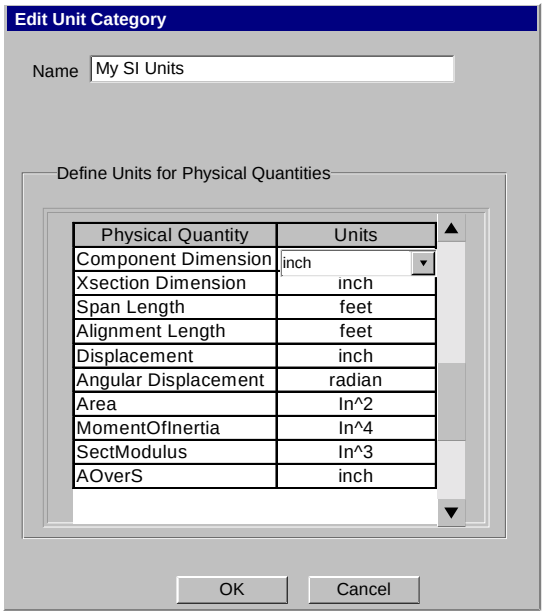


Figure 103 - Edit Unit Categories

7 HANDLING OF INPUT "ERRORS"

In general, computer programs are intolerant and arrogant. Most programs consider it a "user error" or an "input error" when given input data that they cannot handle. These types of situations are typically rectified by an "error dialog" that rudely gets in your face until the data is changed to the program's liking (or until you give up and use another program). In fact, these "errors" are typically not due to a mistake made by the user, but by the program's inability to perform the task at hand.

This is not to say that programs should just quietly accept data that it can't deal with and blindly calculate and report results based on them. Users must be informed if their input creates physically impossible situations, surpasses known limitations of the program, or just doesn't make sense. However, we believe that problems with input data should be presented in a more tactful and productive manner. Our solutions to handling input "errors" are given below.

7.1 Types of Input Errors

The best way to avoid erroneous input data is simply to make it impossible for users to enter bad data. However, this is practically impossible for all but the most trivial programs. If we are to deal with input errors, we must first know where they come from. For our purposes, we have categorized errors into three categories: 1) Simple Errors, 2) Suspicious Errors, and 3) Complex Errors.

Simple Errors are straightforward and can be detected and prevented directly at the time of input. An example of a simple error is if the user input a negative modulus of elasticity, or a negative span length. Another example of a simple error is a syntax error (e.g., entering "abc" as a floating point value). Errors of this type are typically restricted to a single dialog and can be rejected directly in the dialog that they are entered in.

Suspicious Errors represent data that is possible, but not within the normal range expected. An example of this would be an 85 degree skew angle, or an elastic modulus of concrete of 30ksi. These values may be mathematically possible, but are not practical in the real world. In these cases, we will want to inform the user that something may be wrong, but still allow him to continue his work.

Complex Errors stem from problems caused by dependencies on other data. An example of this could occur if a user wants to move his entire bridge by changing the stationing of each pier. During this process, he might place pier 1 down station from pier 3. This is clearly an error, but it is only temporary – he will soon get around to moving the other piers, and he should not be harassed by the program getting in his face about it.

7.2 Dealing With Simple Errors

The program should deal with simple errors immediately. It should not be possible to close a dialog if it has "abc" in a field that requires a floating-point number. The first line of defense is to eliminate the possibility of simple input errors. For example, if an input is to be numeric, letter key presses should be ignored. If a number should be positive only, the minus key should not be accepted.

The second line of defense is to provide a hint to the user of erroneous input. If the input control cannot restrict input to only valid data, then it should color the input data. For example, if an input can only be positive values, it is difficult to prevent the user from entering 0.00. If the value in the input control is zero, it will be displayed in red. This will tell the user that if he tries to close the dialog he will be presented with an error message and directed to correct the erroneous input. Then, odds are, the user will correct the problem before the program has to get into his face.

7.3 Dealing with Suspicious and Complex Errors – The Status Center

In the preceding paragraphs, we have decided that users should not be forced into dealing with suspicious and complex errors immediately. This means that we need a user interface element that serves to remind users that

problem(s) exist, but does not force them to fix them immediately if they don't want to. This is the purpose of the Status Center.
The Status Center has two main viewable elements in the user interface: A color bar in the Status Bar of the main application window, and a modeless dialog that displays active status items. It is also accessible from the main menu via View|Status Center

7.3.1 Status Center Color Bar

The Status Center Color Bar is located in the far right side of the status bar for the main window and is visible at all times while the application is running. The color bar, as shown in Figure 104, is filled with a color representing the state of the input data for the current project.

Figure 104 - Status Color Bar

There are four possible states that can be represented by the color bar as shown in the following table

Table 27 - Status Bar Color States

Color	Status Level	Description
	1	All input data is within normal operating parameters. Informational messages may be in the status center and are signified by the starburst mentioned below.
	2	Input data is unusual or suspect, but analyses can be run.
	3	Input data is out of range for one or more analyses (or plug-ins), but model can be viewed, exported, and processed by some analyses. Analyses with problems cannot be run.
	4	Input data represents model that is physically impossible (e.g., negative span length). One or more views may not be able to display their data. Must be fixed in order to update views (should never happen).

When new messages are logged, the status color bar shall show a starburst until a subsequent command is executed or until the Status Center is visited. For example:

Figure 105 - Status Color Bar Indicating New Message

7.3.1.1 Color Bar Mouse Interactions

The color bar supports limited mouse interactions as shown in the table below.

Action	**Result**
Pause mouse-over or Single Click	Display tooltip showing most recent, highest-level message in status queue, or "Status OK" if green.
Double Click	Bring up Status Center dialog

7.3.2 Status Center Dialog

The Status Center dialog is a modeless dialog that can be access by double clicking on the Status Color Bar or via View|Status Center on the main menu.

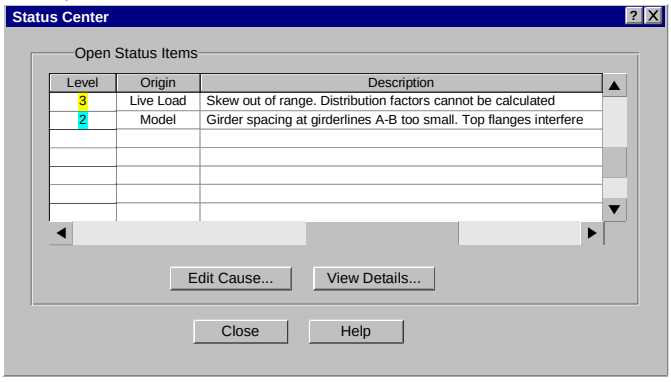


Figure 106 - Status Center Dialog

The Status Center dialog contains a list box of status items. Each status item represents an issue with the current input data. Issues are listed in order of relative importance (as determined by the system).
The fix for some status items will be straightforward enough so that the cause can be edited directly. In this case, the issue can be double-clicked, or selected and then pressing the “Edit Cause” button. This will bring up the editing dialog for the data item with the item highlighted.
The “View Details…” button will bring up a dialog showing the details of the current status item. This dialog contains a text field that can show additional information about how to reconcile the problem in question.

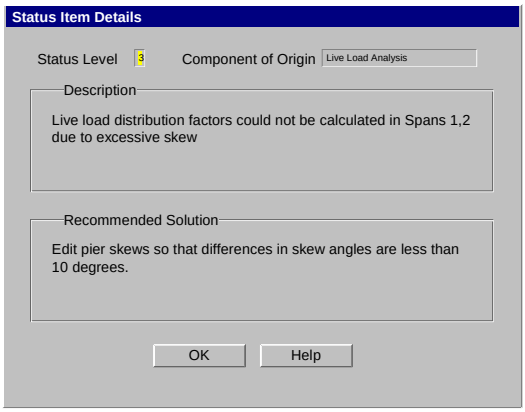


Figure 107 - Status Item Details

8 MENUS

This section details all of the menus in the program.

8.1 Application Menus

These menus are displayed when no project is opened. This occurs when the program is first started or after project is closed.

Table 28 Application Menu

Menu Items	Submenu Items
File	New Open Printer Setup Plotter Setup Program Settings… *MRU* Exit
Help	See Section 8.3 below

8.1.1 File

The File menu provides an interface for the operations that apply to a file.

8.1.1.1 New…

Displays the New Project dialog that lists the available project templates.

8.1.1.2 Open…

Opens an existing project workspace. At the completion of this operation all windows will be restored to their previous size and location.

8.1.1.3 Printer Setup

Used to select and configure a printer. Displays the standard print setup dialog.

8.1.1.4 Plotter Setup

Used to select and configure a printer for plotting. Displays the standard print setup dialog, but is used to manipulate QConBridge II settings and has nothing to do with the Windows default printer.

8.1.1.5 Program Settings

Brings up dialog to allow user to select settings associated with the QConBridge II application.

8.1.1.6 MRU

List of the 4 most recently used projects. Selection of one of the most recently used projects will cause that project to be opened. The behavior is similar to that of File | Open.

8.1.1.7 Exit

Exits the program.

8.2 Product Model Project Menus

These menus provide an interface for manipulating the elements of a QConBridge II product model-based project.

Table 29 QConBridge II Product Model Project Menu

Menu Items	Submenu Items
File	*Standard File commands listed in Section 8.5.1 below, plus:* _____ Export Bridge Analysis Model… Export Girderline Model… Export Pier Analysis Model…
Edit	*Refer to Section 8.5.2 below plus:* _____ Mirror Properties… Swap Piers
Bridge	Bridge Contractor… _____ Add Span/Pier… _____ Edit Alignment… Edit Framing Plan… Edit Span Hinges… Edit Slab and Appurtenances… _____ Move Bridge… Edit Span Lengths… _____ Edit Plate Schedule (built up steel girders only)
Models	View Framing Plan… View Span Hinges… _____ Analysis Settings…
Loads	Add Concentrated Load… Add Distributed Load… Add Support Settlement…

Menu Items	Submenu Items
	Temperature Load…
Library	*Refer to Section 8.5.2.9 below.*
Options	*Refer to Section 8.5.5 below*
View	Framing Plan Girderline Elevation Pier Elevation ———————- Analysis Results Reports... ——————— Status Center ——————— View Settings Toolbars... Status Bar
Window	*Refer to Section 8.5.6 below*
Help	*Refer to Section 8.5.7 below*

8.2.1 File

The File menu provides an interface for the operations that apply to project files. Most commands for the file menu are common to all project types. These common commands are described in Section 8.5.1 below. Only custom commands for Product Models are shown here.

8.2.1.1 Export Bridge Analysis Model

Displays the standard File|Save As dialog. Allows the BAM for the current product model to be saved as a BAM project file.

8.2.1.2 Export Girderline Model

Displays the standard File|Save As dialog. Allows a single girderline model from the current product model to be saved as a BAM project file.

8.2.1.3 Export Pier Model

Displays the standard File|Save As dialog. Allows a single pier model from the current product model to be saved as a TBAM project file.

8.2.2 Edit

Most editing operations are explained in 8.5.2 below.

8.2.2.1 Mirror Properties

Causes the properties of the selected actor to be mirrored about the actor’s center point.

8.2.2.2 Swap Piers

Swaps the two piers for a selected span/pier. Only enabled if a span/pier is selected.

8.2.3 Bridge

The menu supports operations affecting the bridge product model.

8.2.3.1 Bridge Contractor…

Brings up the Bridge Contractor wizard to edit the current project.

8.2.3.2 Add Span/Pier

Adds a span/pier to the left end of the model. See 3.1.4.5.3 above for details

8.2.3.3 Edit Alignment

Brings up the Alignment/Roadway editing dialog.

8.2.3.4 Edit Framing Plan

Brings up the Framing Plan editing dialog.

8.2.3.5 Edit Span Hinges

Brings up the Span Hinge Editing dialog. Only available for built-up steel girder bridges.

8.2.3.6 Edit Slab and Appurtenances

Brings up the Edit slab and Appurtenances dialog.

8.2.3.7 Move Bridge

Bridges up the Move Bridge Dialog to translate entire bridge along alignment.

8.2.3.8 Edit Span Lengths

Brings up Edit Span Lengths Dialog

8.2.3.9 Edit Plate Schedule (built up steel girders only)

Brings up the plate schedule editing dialog for the selected girder

8.2.4 Models

This menu is used to view and edit the current Bridge Analysis Model.

8.2.4.1 View Framing Plan

Brings up the BAM Framing Plan dialog in read-only mode.

8.2.4.2 View Span Hinges

Brings up the BAM Edit Span Hinges dialog in read-only mode.

8.2.4.3 Analysis Settings

Brings up the Analysis Settings Dialog.

8.2.5 Loads

This menu supports editing commands for the Current BAM and TBAM. Menu items are disabled except when loadable actors are selected in the BAM and TBAM views.

8.2.5.1 Add Concentrated Load

Adds a concentrated load to the currently selected BAM or TBAM actor. Brings up the Edit Point Load dialog.

8.2.5.2 Add Distributed Load

Adds a distributed load to the currently selected BAM or TBAM actor. Brings up the Edit Distributed Load dialog.

8.2.5.3 Add Support Settlement

Adds a concentrated load to the currently selected BAM or TBAM support actor. Brings up the Edit Support Settlement dialog.

8.2.5.4 Temperature Load

Edits the superstructure temperature load. Brings up the Edit Superstructure Temperature Load dialog.

8.2.6 View

The View menu provides an interface for the operations that display and hide elements of the user interface, access the various views the program offers, and that control how information is presented in the views.

8.2.6.1 Framing Plan

Displays the Product Model Framing Plan View.

8.2.6.2 Girderline Elevation

Displays the Girderline Elevation View.

8.2.6.3 Pier Elevation

Displays the Pier Elevation View.

8.2.6.4 Analysis Results

Creates a new Analysis Results View.

8.2.6.5 Report...

Displays the Report Selection Dialog, followed by the selected report.

8.2.6.6 Status Center…

Brings up the Status Center dialog as described in Section 7.3.2 above.

8.2.6.7 View Settings

Brings up the view settings dialog for the view in focus.

8.2.6.8 Toolbars...

Displays the Toolbar Dialog. This dialog is used to display or hide toolbars.

8.2.6.9 Status Bar

Displays or hides the status bar.

8.3 BAM Project Menus

These menus provide an interface for manipulating the elements of a QConBridge II BAM-based project.

Table 30 QConBridge II BAM Project Menu

Menu Items	Submenu Items
File	*Standard File commands listed in Section 8.5.1 below, plus:*

Menu Items	Submenu Items
	———————— Export Single Girderline Model… Export TBAM…
Edit	*Refer to Section 8.5.2 below plus:* ———————— Mirror Properties… Swap Supports
Models	Bridge Contractor… Edit Stages… ————————- Add Column… Add Span/Support… ———————— Edit Alignment… Edit Framing Plan… Edit Roadway Widths… Edit Span Hinges… ————————- Move Bridge… Edit Span Lengths… Edit Girder Spacing… ————————- Analysis Settings…
Loads	Add Concentrated Load… Add Distributed Load… Add Support Settlement… Add Temperature Load…
Library	*Refer to Section 8.5.2.9 below.*
Options	*Refer to Section 8.5.5 below*
View	Framing Plan View Girderline Elevation View Pier View ————————- Analysis Results Reports... ———————— View Settings Toolbars... Status Bar

Menu Items	Submenu Items
Window	*Refer to Section 8.5.6 below*
Help	*Refer to Section 8.5.7 below*

8.3.1 File

The File menu provides an interface for the operations that apply to project files. Most commands for the file menu are common to all project types. These common commands are described in Section 8.5.1 below. Only custom commands for Product Models are shown here.

8.3.1.1 Export Single Girderline Model

Displays the standard File|Save As dialog. Allows a single girderline model from the current product model to be saved as a BAM project file.

8.3.1.2 Export Pier Model

Displays the standard File|Save As dialog. Allows a single pier model from the current product model to be saved as a TBAM project file.

8.3.2 Edit

Most editing operations are explained in 8.5.2 below.

8.3.2.1 Mirror Properties

Causes the properties of the selected actor to be mirrored about the actor's center point.

8.3.2.2 Swap Supports

Swaps supports for the selected span/support. Enabled only if a span/support is selected.

8.3.3 BAM

This menu is used to view and edit the current Bridge Analysis Model.

8.3.3.1 Bridge Contractor…

Brings up the Bridge Contractor wizard to edit the current project.

8.3.3.2 Edit Stages

Brings up the Stage Editing dialog

8.3.3.3 Add Column

Adds a column to the left end of the pier model (only active in pier view)

8.3.3.4 Add Span/Support

Adds a default span/support to the left end of the model. (not active in pier view).

8.3.3.5 Edit Alignment

Brings up the Edit Alignment dialog.

8.3.3.6 Edit Framing Plan

Brings up the BAM Framing Plan dialog.

8.3.3.7 Edit Roadway Widths

Brings up the Edit Roadway Widths dialog.

8.3.3.8 Edit Span Hinges

Brings up the BAM Edit Span Hinges dialog.

8.3.3.9 Move Bridge

Bridges up the Move Bridge Dialog to translate entire bridge along alignment.

8.3.3.10 Edit Span Lengths

Brings up Edit Span Lengths Dialog

8.3.3.11 Edit Girder Spacing

Brings up the Edit Girder Spacing dialog.

8.3.3.12 Analysis Settings

Brings up the Analysis Settings Dialog.

8.3.4 Loads

This menu supports load editing commands for the Current LBAM and TBAM. Menu items are disabled except when loadable actors are selected in the BAM and TBAM views.

8.3.4.1 Add Concentrated Load

Adds a concentrated load to the currently selected LBAM or TBAM actor. Brings up the Edit Point Load dialog.

8.3.4.2 Add Distributed Load

Adds a distributed load to the currently selected LBAM or TBAM actor. Brings up the Edit Distributed Load dialog.

8.3.4.3 Add Support Settlement

Adds a concentrated load to the currently selected LBAM or TBAM support actor. Brings up the Edit Support Settlement dialog.

8.3.4.4 Add Temperature Load

Adds a temperature load to all LBAMS. Brings up the Edit Temperature dialog.

8.3.5 View

The View menu provides an interface for the operations that display and hide elements of the user interface, access the various views the program offers, and that control how information is presented in the views.

8.3.5.1 Framing Plan

Displays the Analysis Model Framing Plan View.

8.3.5.2 Elevation View

Displays the Girderline Elevation View.

8.3.5.3 Pier View

Displays the TBAM Elevation View.

8.3.5.4 Analysis Results

Creates a new Analysis Results View.

8.3.5.5 Report...

Displays the Report Selection Dialog, followed by the selected report.

8.3.5.6 View Settings

Brings up the view settings dialog for the view in focus.

8.3.5.7 Toolbars...

Displays the Toolbar Dialog. This dialog is used to display or hide toolbars.

8.3.5.8 Status Bar

Displays or hides the status bar.

8.4 TBAM Project Menus

These menus provide an interface for manipulating the elements of a QConBridge II TBAM-based project.

Table 31 QConBridge II TBAM Project Menu

Menu Items	Submenu Items
File	*Standard File commands listed in Section 8.5.1 below.*
Edit	*Refer to Section 8.5.2 below plus:* _____- Mirror Properties…
Model	Bridge Contractor… _____ Edit Framing Plan… Edit Roadway Width… _____- Analysis Settings…
Loads	Add Concentrated Load… Add Distributed Load… Add Support Settlement…
Library	*Refer to Section 8.5.2.9 below.*
Options	*Refer to Section 8.5.5 below*
View	Pier View _____- Analysis Results Reports... _____ View Settings Toolbars... Status Bar
Window	*Refer to Section 8.5.6 below*

Menu Items	Submenu Items
Help	*Refer to Section 8.5.7 below*

8.4.1 File

The File menu provides an interface for the operations that apply to project files. Most commands for the file menu are common to all project types. These common commands are described in Section 8.5.1 below.

8.4.2 Edit

Most editing operations are explained in 8.5.2 below.

8.4.2.1 Mirror Properties

Causes the properties of the selected actor to be mirrored about the actor's center point.

8.4.3 TBAM

This menu is used to view and edit the current TBAM.

8.4.3.1 Bridge Contractor…

Brings up the Bridge Contractor wizard to edit the current project.

8.4.3.2 Edit Framing Plan

Brings up the TBAM Framing Plan dialog.

8.4.3.3 Edit Roadway Width

Brings up the Edit Roadway Widths dialog.

8.4.3.4 Analysis Settings

Brings up the Analysis Settings Dialog.

8.4.4 Loads

This menu supports load editing commands for the Current TBAM Menu items are disabled except when loadable actors are selected in the TBAM view.

8.4.4.1 Add Concentrated Load

Adds a concentrated load to the currently selected TBAM actor. Brings up the Edit Point Load dialog.

8.4.4.2 Add Distributed Load

Adds a distributed load to the currently selected TBAM actor. Brings up the Edit Distributed Load dialog.

8.4.4.3 Add Support Settlement

Adds a concentrated load to the currently selected TBAM support actor. Brings up the Edit Support Settlement dialog.

8.4.5 View

The View menu provides an interface for the operations that display and hide elements of the user interface, access the various views the program offers, and that control how information is presented in the views.

8.4.5.1 Pier View

Displays the TBAM Elevation View.

8.4.5.2 Analysis Results

Creates a new Analysis Results View.

8.4.5.3 Reports...

Displays the Report Selection Dialog, followed by the selected report.

8.4.5.4 View Settings

Brings up the view settings dialog for the view in focus.

8.4.5.5 Toolbars...

Displays the Toolbar Dialog. This dialog is used to display or hide toolbars.

8.4.5.6 Status Bar

Displays or hides the status bar.

8.5 Common Menus

This section defines menus that are common to all project types.

8.5.1 File Menu

The file menu is used to open, close, create, print, and set properties related to project files. Some custom commands exist for different project types and are noted in the sections above for individual project types.

Menu Items	Submenu Items
File	New Open... Close Save Save As... ————————- Import Library Entries… ————————-- *(Custom Commands Here)* ————————- Properties... Program Settings… ————————- Print... Printer Setup... Plotter Setup… ————————- Send ————————- *MRU* ————————-

Menu Items	Submenu Items
	Exit

8.5.1.1 New…

Creates a new project. Before a new project can be created any open projects must be closed. See description of File | New for the Application Menu for additional information.

8.5.1.2 Open…

Opens an existing project workspace. The current project workspace must be closed before another workspace can be opened. At the completion of this operation all windows will be restored to their previous size and location.

8.5.1.3 Close

Closes the project workspace. If the project has been modified, a request to save the changes is made. If the project is unnamed, a request for a project name is made. At the completion of this operation the program will revert back to the program startup state.

8.5.1.4 Save

Saves the current project workspace. If the project is unnamed, this operation becomes Save As. At the completion of this operation all modifications to the project since the last time it was saved are committed to disk. This operation is only available when a project workspace is modified.

8.5.1.5 Save As

Saves the current project workspace with a name and location provided by the user. At the completion of this operation the project workspace is committed to disk with the name and location provided by the user.

8.5.1.6 Import Library Entries

Displays the standard File|Open dialog. Allows reading of library entries from another project into the currently open project. For duplicate entries, a dialog is used to ask whether to Overwrite or Ignore the duplicate entry from the import file.

8.5.1.7 Custom Commands

Placeholder for custom commands for a given project type.

8.5.1.8 Program Settings

Displays the Program Settings dialog.

8.5.1.9 Properties

Displays the properties of the project in the Project Properties Dialog.

8.5.1.10 Print

Print the contents of the current window. This operation is only available when a window with printable attributes is active.

8.5.1.11 Print Setup

Used to select and configure a printer. Displays the standard print setup dialog.

8.5.1.12 Send

Sends a mail message with the project as an attachment.

8.5.1.13 MRU

List of the 4 most recently used projects. Selection of one of the most recently used projects will cause that project to be opened. The behavior is similar to that of File | Open.

8.5.1.14 Exit

Exits the program. If the project has been modified, the user is given an opportunity to save the changes before the program exists.

8.5.2 Edit Menu

The Edit menu supports context-based editing of the currently selected actor. It also allows undo and redo of recent commands.

Menu Items	Submenu Items
Edit	Undo Redo ————————- Cut Copy Paste Paste Special… Delete Properties… Select All

8.5.2.1 Undo

Undoes the effect of the most recent command.

8.5.2.2 Redo

Undoes the last undo (only if undo was the last command, otherwise disabled).

8.5.2.3 Cut

Deleted the currently selected actor from the model and copy it to the paste buffer.

8.5.2.4 Copy

Copy the currently selected actor to the paste buffer.

8.5.2.5 Paste

Paste the data in the paste buffer to the currently selected location.

8.5.2.6 Paste Special…

Brings up a Paste Special dialog showing options for the current paste.

8.5.2.7 Delete

Delete the currently selected actor.

8.5.2.8 Properties…

Show the properties dialog for the currently selected actor and make them editable if appropriate.

8.5.2.9 Select All

Selects all actors in the view.

8.5.3 Loads Menu

This menu allows editing of user-defined load information. The only item that is common among all model types is load groups.

Menu Items	Submenu Items
Loads	Edit Load Groups… Edit Superstructure Load Modifiers… Edit Substructure Load Modifiers…

8.5.3.1 Edit Load Groups…

Add load groups to the project or modify project-specific load groups.

8.5.3.2 Edit Superstructure Load Modifiers…

Edit eta factors for superstructure components

8.5.3.3 Edit Substructure Load Modifiers…

Edit eta factors for substructure components

8.5.4 Library Menu

The Library menu provides an interface for the operations that pertain to managing library entries.

Menu Items	Submenu Items
Library	Edit Libraries Add New Entry Edit Entry Delete Entry Duplicate Entry

8.5.4.1 Edit Libraries

Opens the Library Editor Window.

8.5.4.2 Add New Entry

Creates a new library entry.

8.5.4.3 Edit Entry

This is a generic command that displays the editing dialog for the currently selected library entry.

8.5.4.4 Delete Entry

Deletes the selected library entry

8.5.4.5 Duplicate Entry

Duplicates the selected library entry

8.5.5 Options

The Options menu is used to select miscellaneous program settings.

Menu Items	Submenu Items

Menu Items	Submenu Items
Options	Turn AutoCalc On Update Now Ctrl+U ——————— Units…. ——————— Design Criteria… Load Modifiers… (Should go on the loads menu???)

8.5.5.1 Turn AutoCalc On/Off

Toggles AutoCalc Model on and off. When AutoCalc mode is on, the menu text is Turn AutoCalc Off.

8.5.5.2 Update Now

Causes the program to update all views. This option is only available if AutoCalc is off and there are pending view updates.

8.5.5.3 Units...

Displays the System of Units dialog. This dialog is used to set the active system of units for the program.

8.5.5.4 Design Criteria...

Displays the Specification dialog. This dialog is used to set the active specification for the program.

8.5.5.5 Load Modifier...

Displays the Load Modifiers dialog. This dialog is used to set LRFD η factors for this bridge project.

8.5.6 Window

The Window menu provides an interface for the operations that manipulate the size and location of child windows.

Menu Items	Submenu Items
Window	Cascade Tile Arrange Icons ——————— Large Icons Small Icons List

8.5.6.1 Cascade

Rearranges all display windows into a cascade pattern. The toolbars, and status bars are not affected by this operation.

8.5.6.2 Tile

Tiles all open windows. The toolbars, and status bars are not effected by this operation.

8.5.6.3 Arrange Icons

Arranges all minimized window icons.

8.5.6.4 Large Icons

Only enabled if focus view is a list view. Sets view to large icon mode.

8.5.6.5 Small Icons

Only enabled if focus view is a list view. Sets view to small icon mode.

8.5.6.6 List

Only enabled if focus view is a list view. Sets view to list mode.

8.5.7 Help Menu

The Help menu provides an interface for the operations pertaining to user assistance.

Menu Items	Submenu Items
Help	Help Topics Tip of the Day Online Resources Legal Notices About QConBridge

8.5.7.1 Help Topics

Provides access to the Help Topics Browser

8.5.7.2 Tip of the Day

Displays the Tip of the Day dialog.

8.5.7.3 Online Resources

This menu cascades to a submenu containing a “Visit WSDOT” option and “QConBridge II Home Page” option. The first option opens the user’s default web browser and goes to the WSDOT home page. The second option opens the user’s default web browser and goes to the QConBridge II Home Page where the most up to date information about QConBridge II exists.

8.5.7.4 Legal Notices

Displays the Alternate Route License

8.5.7.5 About QConBridge

Displays copyright and version information in the About Dialog.

9 TOOLBARS

The program will have a set of built in toolbars that the user may hide or display. The program will keep track of the positions and sizes of active toolbars so that the settings can be restored each time the program is executed. All toolbars will be dockable.

9.1 Standard Toolbar

The Standard Toolbar contains file commands and other commands available to all QConBridge II project types.

Button	Corresponding Command
	File | New
	File | Open
	File | Save
	File | Print (Need a similar icon for plotting)
	Edit | Cut
	Edit | Copy
	Edit | Paste
	Edit | Properties
	View | Analysis Results
	View | Report
	Options | Units. SI Units in the Units Dialog
	Options | Units. US Units in the Units Dialog
	Project | Update Now

Table 32 Standard Toolbar Buttons

9.2 Product Model Project Toolbar

This toolbar is for custom viewing and editing of Product Models

Table 33 - Product Model Toolbar

Button	Corresponding Command
	Bridge Contractor
	Edit Framing Plan
	Edit Alignment
	Edit Slab
	Edit Span Hinges
	View Framing Plan
	View Girderline Elevation
	View Pier

9.3 BAM Project Toolbar

This toolbar is for editing only in BAM projects.

Table 34 - BAM Model Toolbar

Button	Corresponding Command
	Bridge Contractor
	Edit Framing Plan
	Edit Alignment
	Edit Roadway
	Edit Span Hinges
	View Framing Plan
	View Girderline Elevation
	View Pier

9.4 TBAM Project Toolbar

This toolbar is used only in TBAM projects.

Table 35 - TBAM Project Toolbar

Button	Corresponding Command
	Bridge Contractor
	Edit Framing Plan
	Edit Roadway
	View Pier

9.5 Library Toolbar

This toolbar contains buttons for the library editing commands.

Table 36 - Library Toolbar

Button	Corresponding Command
	Library | Edit Entry
	Library | Add New Entry
	Library | Delete Entry
	Library | Duplicate Entry
	Library | Edit Entry

Table 37 Library Toolbar Buttons

9.6 Help Toolbar

The help toolbar is shown in Figure 108. This toolbar contains buttons for the help commands.

Figure 108 Help Toolbar

Button	Corresponding Command
	Help | Online Resources | PGSuper Home Page
	No corresponding menu item. This is the context help tool.

Table 38 Help Toolbar Buttons

10 STATUS BARS

The program will have one status bar along the bottom of the main window. The status bar will display menu hints, and state information for project. The state information is described below.

10.1 Status Center Color Bar

This will be the far right entry in the status bar. The color bar is explained in Section 7.3.1 above.

10.2 Modification State Information

The status bar will display *Modified* if the project has been modified.

10.3 AutoCalc Mode State Information

The status bar will display *AutoCalc ON* or *AutoCalc OFF* depending upon the state of the AutoCalc mode.

10.4 Unit State Information

The status bar will display the name of the currently selected unit category.

11 MISCELLENEOUS INTERFACE ELEMENTS

11.1 Program Settings

This dialog is used to set properties associated with the QConBridge II application.
The Engineer and Company names are used to fill in project properties on new project files.
The master library path can be changed to allow users to customize different libraries for their needs.

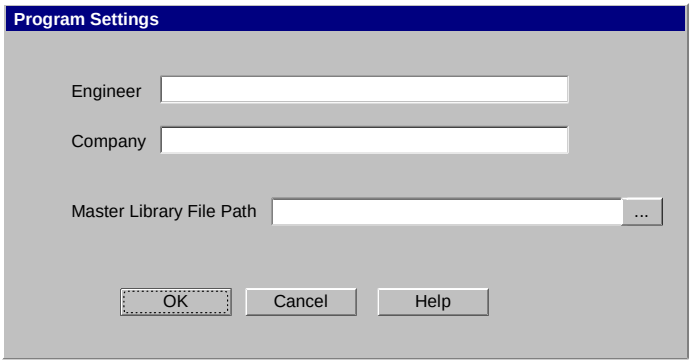


Figure 109 - Edit Program Settings

11.2 Project Properties

This dialog captures some basic information about the project and the engineer.

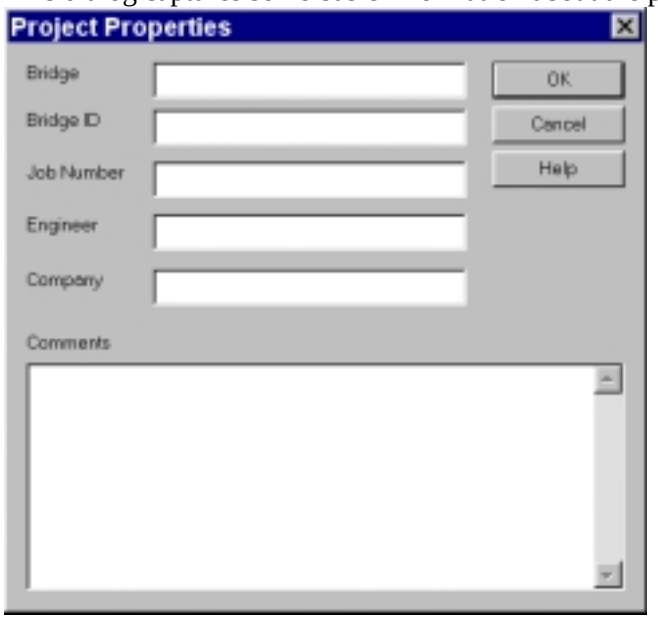


Figure 110 - Edit Project Properties

11.3 Specification Selection

Specifications are made available as Library Entries. However, selection of the current library is a system-level command and is available via the main menu Options|Design Criteria.

11.4 Load Modifiers

Load Modifiers (eta factors) are input separately for substructure and superstructure components. The editing interface shall be the same as for PGSuper.

12 INTEGRATION WITH THE OPERATING SYSTEM

This section describes the specific requirements for integrating the application with the *Windows® 95 and Windows NT®* operating systems. See *The Windows® Interface Guidelines for Software Design* for general integration requirements.

12.1 Install/Uninstall

The program shall be installed through the use of a modern, graphical installation program. The program shall also be able to be uninstalled, which will completely remove all traces of the previous installation.

12.2 The Registry

The program must update the registry every time it is executed. The program must have an unregister facility. This facility will be implemented with a command line argument.

12.3 Executable File Properties

The *Windows®* Property Sheet Shell shall be extended to include the information displayed in the About dialog for the executable file. See pg. 126 of *The Windows® Interface Guidelines for Software Design*, and *Windows® 95 Property-Sheet Shell Extensions*, Jeff Prosise, PC Magazine, April 25, 1995.

12.4 Project File Properties

The *Windows®* Property Sheet Shell shall be extended to include the project properties. See *Windows® 95 Property-Sheet Shell Extensions*, Jeff Prosise, PC Magazine, April 25, 1995.

13 GLOSSARY OF TERMS

Bridge Analysis Model (BAM)

An idealization of a bridge for structural analysis purposes. The model is designed for use with the modeling approach described in LRFD 4.6.2, and shall work with all bridge types described in that section.

Plug-in

An application that is embedded within the main application. Well done plug ins appear to be an integral part of the QconBridge II application. They share the main menu, toolbars, and views available from the program. Data for plug-ins is also saved within QconBridge II input files.

Product Model

A model that describes a thing, as it exists in the real world. A complete (full-fidelity) product model of a bridge would represent all of the information required by a contractor to build a bridge. In QConBridge II, we don't need this level of detail, so we will only describe enough information required for our purposes.

Transverse Bridge Analysis Model (TBAM)

An idealization of a pier structure for structural analysis purposes.

14 SPECIAL CONSIDERATIONS

14.1 Dealing with models that can't be analyzed

Describe the following situations

14.1.1 Model Doesn't Meet LRFD Requirements for Simplified Analysis

Same as PGSuper, except give some advise on how to adjust the model and how to create a BAM only model.

14.1.2 Degrading Stiffness

QconBridge II will just go ahead and analyze these. The status center will show a warning when this situation occurs. We will also put a big note in the reports that highlights the problem, explains why the analysis results are wrong, and how the user might correct the problem.