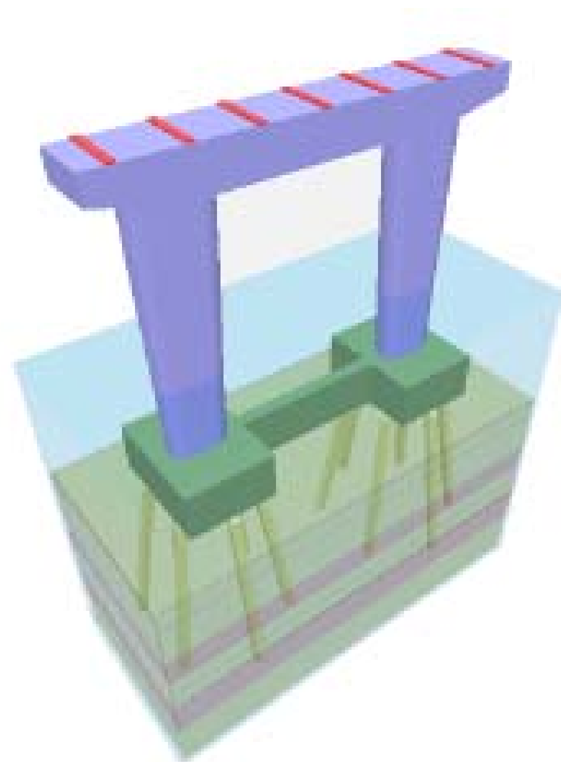
FB-MULTIPIER

Step-By-Step Advanced Example Problems

March 2005



Bridge Software Institute

MP-1. BRIDGE OVER WATER MODEL

Shown in **Figure 1.1** is a two-span portion of a six-span bridge, which will be modeled in **Example MP-1**. The bridge spans a navigable waterway crossing, where the piers are subjected to axial loads and a potential lateral vessel collision load. In addition to dead load and a vessel collision load, the bridge will be subjected to vehicular live loads and wind loads. Loads will be applied and combined using the AASHTO LRFD Design Specification. This example builds upon the pier modeled in Example 2 in the FB-Pier Users Manual. The user is encouraged to review single pier modeling examples before modeling a full bridge using FB-MultiPier.

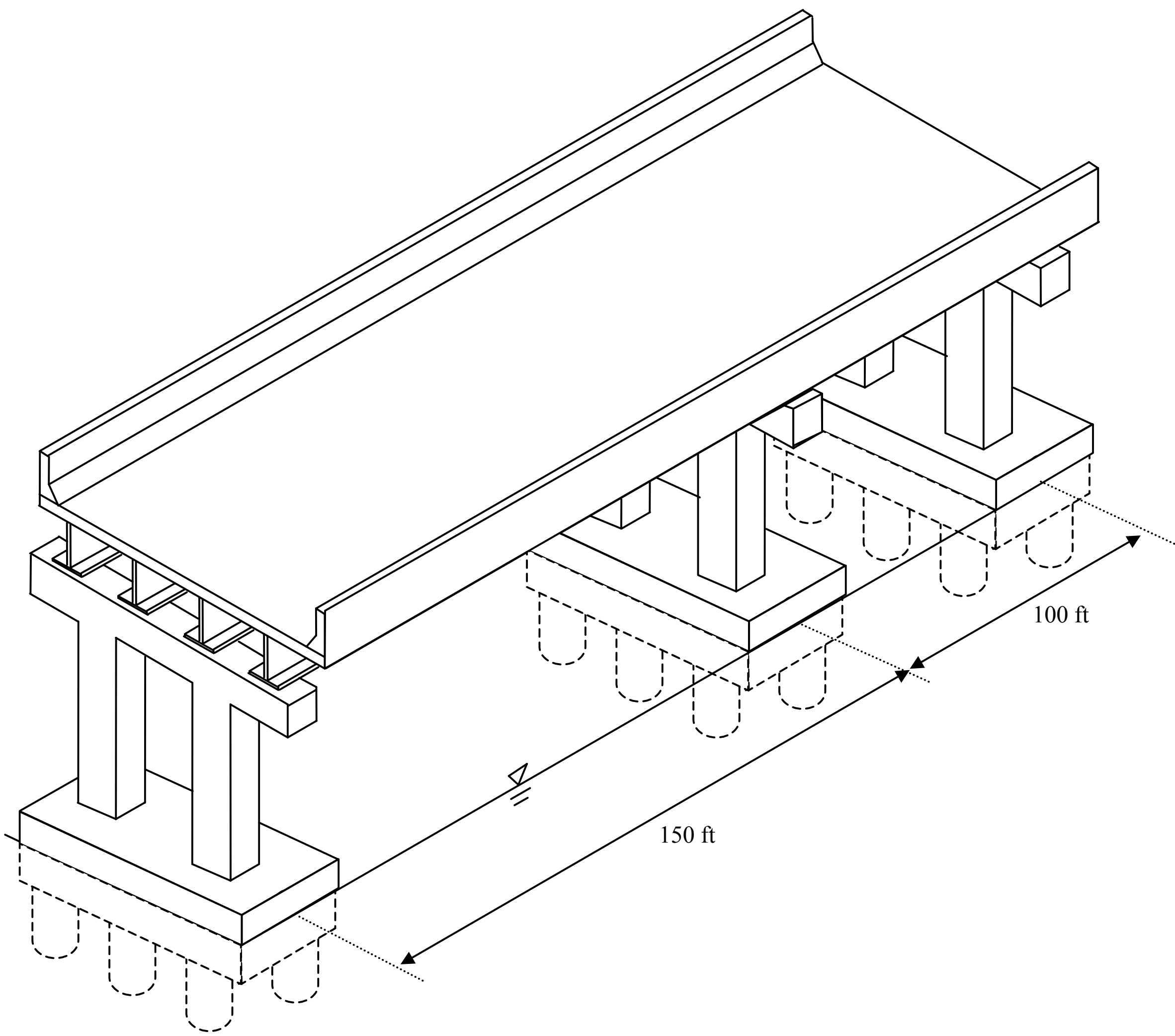


Figure 1.1 Example MP-1, Two-Span Bridge Structure

To recap, the pier modeled in Example 2 consists of 6-54 inch drilled shafts (80 ft long) and two pier columns which are 30 ft tall, 5 ft square and spaced 16 ft apart. The pier cap is 4 ft thick and the drilled shaft cap is 10 ft thick with a 4.5 ft overhang. Due to scour, the sand surface is located 15 ft below mean sea level, and the soft rock is characterized as FHWA's intermediate geomaterial. The properties of the sand and rock are given in **Figure 1.2**. This pier model will be used to support the two spans modeled in this bridge example.

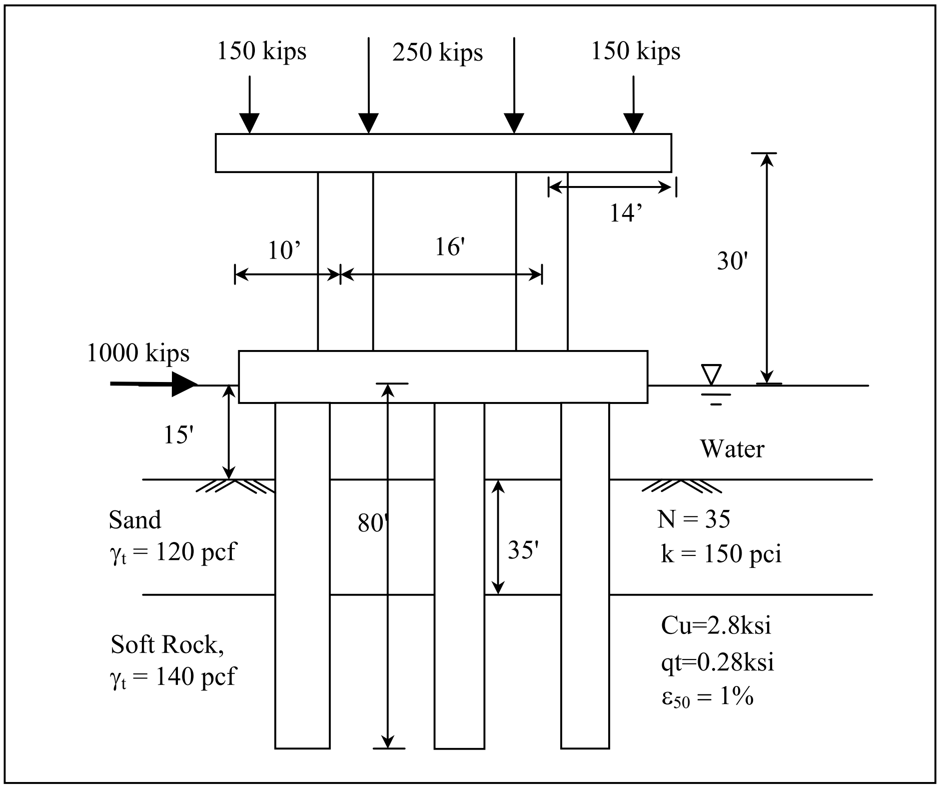


Figure 1.2 Example 2, Pier Structure

This example will consider AASHTO LRFD design load combinations to determine the worst case loading scenario for the drilled shafts and pier. The following LRFD limit states will be checked:

STRENGTH-I, STRENGTH-III, STRENGTH-V, EXTREME-II, SERVICE-I

The load types shown below in Table 1.1 will be considered. The bearing loads shown in the table result from a live load placed on Span #1. The "(L)" marker indicates that the load will be applied in the left bearing row. Each pier has two rows of bearings.

Dead Load (DC)	Automatically generated by the program (self weight)		
Water Load (WA)	Automatically generated by the program (buoyancy)		
Live Load (LL1)	Bearing 1 (L) Bearing 2 (L) Bearing 3 (L) Bearing 4 (L)	80 kips 120 kips 0 kips 0 kips	Vehicular Live Load
Impact Load (IM1)	Bearing 1 (L) Bearing 2 (L) Bearing 3 (L) Bearing 4 (L)	26 kips 40 kips 0 kips 0 kips	Vehicular Dynamic Load Allowance
Braking Load (BR1)	Bearing 1 (L) Bearing 2 (L) Bearing 3 (L) Bearing 4 (L)	15 kips 15 kips 15 kips 15 kips	Y-direction (longitudinal) load
Live Load (LL2)	Bearing 1 (L) Bearing 2 (L) Bearing 3 (L) Bearing 4 (L)	100 kips 110 kips 105 kips 0 kips	Vehicular Live Load
Impact Load (IM2)	Bearing 1 (L) Bearing 2 (L) Bearing 3 (L) Bearing 4 (L)	33 kips 36 kips 35 kips 0 kips	Vehicular Dynamic Load Allowance
Braking Load (BR2)	Bearing 1 (L) Bearing 2 (L) Bearing 3 (L) Bearing 4 (L)	20 kips 20 kips 20 kips 20 kips	Y-direction (longitudinal) load
Wind Load on Structure (WS)	To be generated using wind angles of 0, 30, and 60 degrees		
Wind Load on Live Load (WL)	To be generated using wind angles of 0, 30, and 60 degrees		
Vessel Collision (CV)	Node 38	1000 kips	(Lateral - Applied to center of pile cap)

Table 1.1 Loads Applied to Bridge Model

Select Open from the File menu and choose Example2.in from the program directory (**Figure 1.3**).

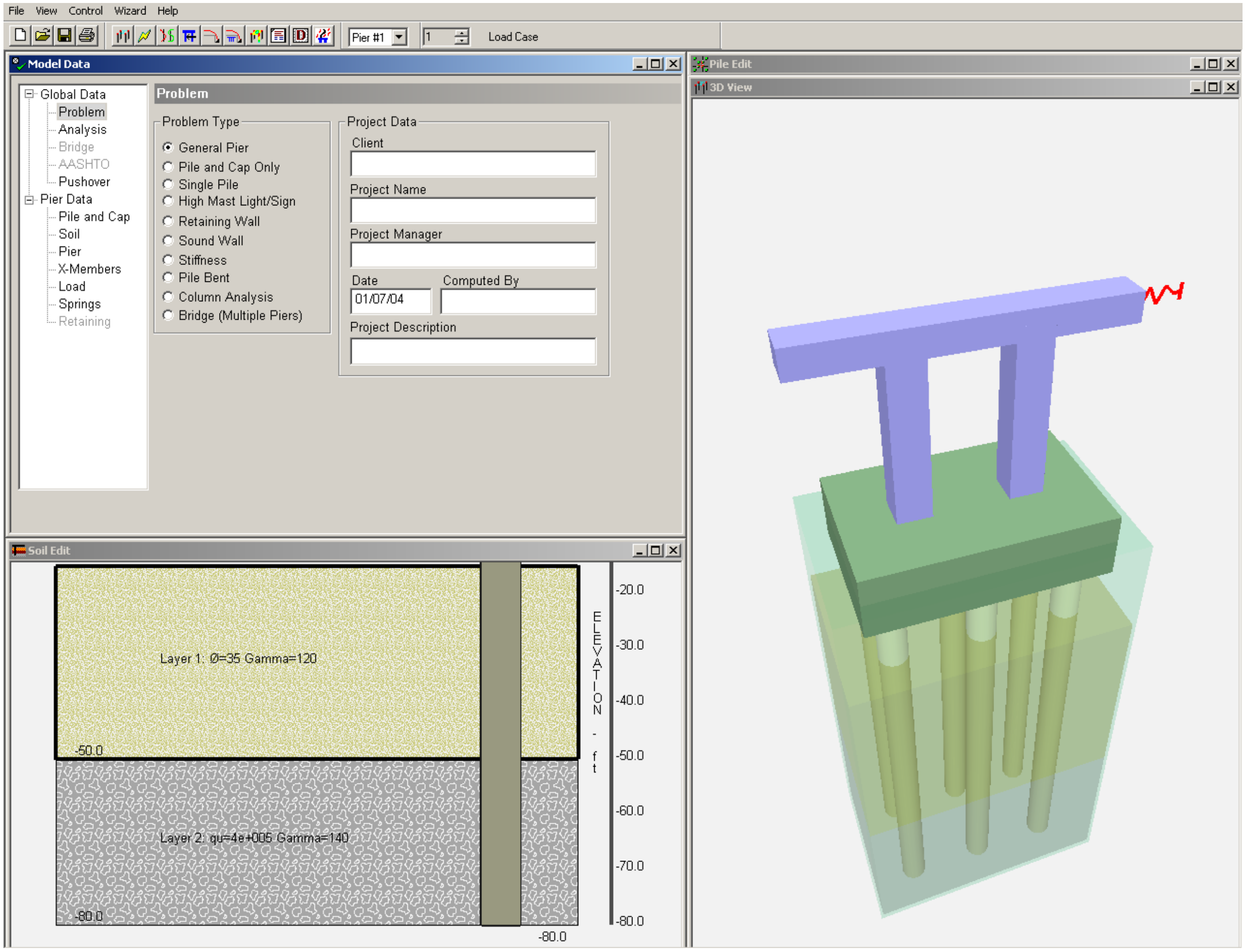


Figure 1.3 Example2.in Model View

The layout of the screens in the FB-MultiPier program is similar to that of FB-Pier to provide a smooth transition between programs. New to FB-MultiPier, however, is the Model Data tree view that allows for easy navigation between the data pages. This navigation can be achieved by using the mouse or by using the arrow keys or the Ctrl-tab key combination. The tree items shown in light grey are either not available for the current model type or can be enabled if certain modeling options are selected.

The first step in modeling the bridge is to modify the single pier that will ultimately be used as a template when generating more piers. The single pier in Example 2 contains concentrated loads applied along the pier cap and a concentrated lateral load applied at the pile cap. The pier cap loads will be replaced by a superstructure dead loads and wind loads generated by the program. The concentrated lateral load will only be applied to one of the three piers to simulate vessel collision. To delete the current loads, select the *Load* data page in the Model Data window and click the “Del” to the left of the load case list. After doing so, only a placeholder for a Preload case will remain. The bridge spring applied to the pier cap should also be removed. When analyzing single piers, a bridge spring is typically used to model the lateral stiffness of the superstructure. This spring is no longer needed since the full bridge model will incorporate the superstructure. To delete the bridge spring, select the *Springs* data page in the Model Data window and click the “Del” button. The pier can now be used to generate a complete bridge model.

The next step in modeling the bridge is to provide bearing locations on the pier cap for the bridge girders. To specify bearing locations, click on the *Pier* item in the Model Data tree view and then click the “Use Bearing Locs.” button to enable bearing location modeling. Click on the “Bearing Locs.” button to enter the bearing data. A single bearing row is modeled by default. Select “Two Rows” for the Bearing Layout. Enter the values shown in the Left Bearing Row and Right Bearing Row Group in **Figure 1.4**.

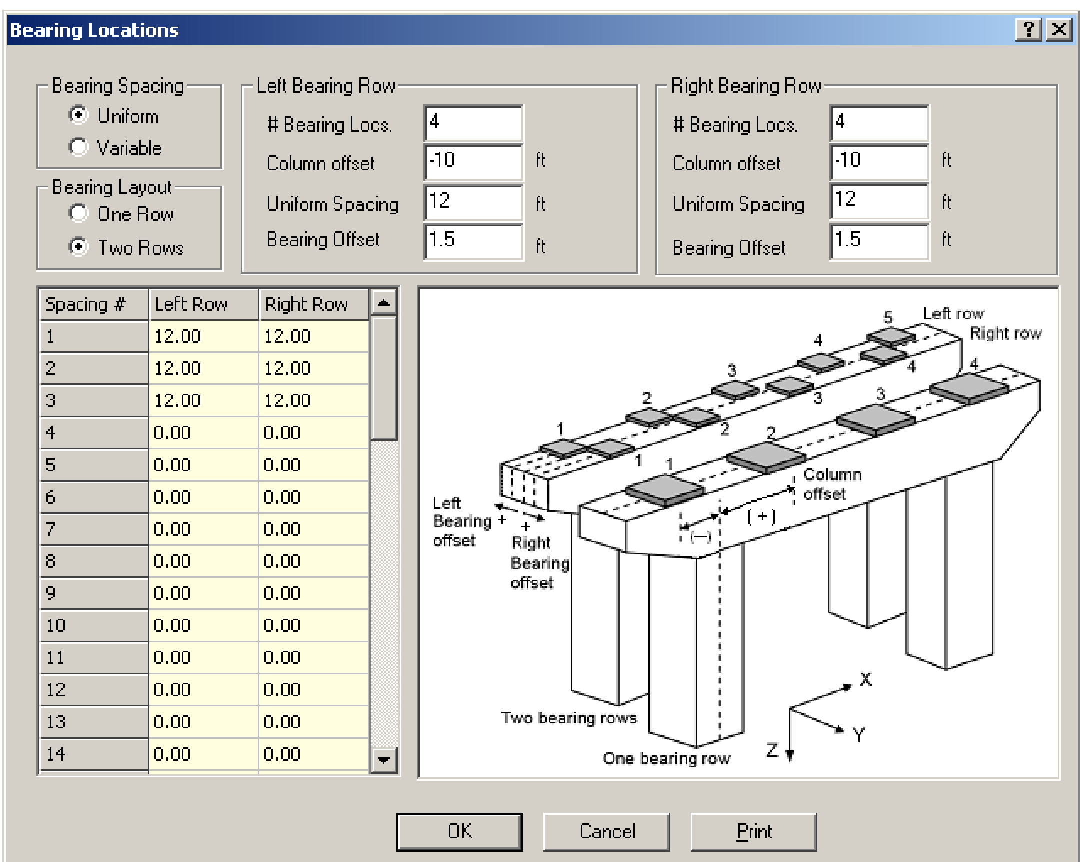


Figure 1.4 Bearing Locations

Bearing locations can be modeled using a uniform or variable spacing. This example calls for 4 bearing locations at a Uniform spacing of 12 ft, offset -10 ft from the first column. The bearings will be offset 1.5 ft from the centerline of the pier cap. In order to establish a naming convention, bridge spans are assumed to be modeled in the positive ‘y’ direction. In doing so, the first row of bearings is called the “Left Row” and the second row of bearings is called the “Right Row.” It is important to note that bearings in the left and right rows do not have to coincide (although they do in this example). That is, there can be a bearing in one row without a corresponding bearing in the other row. This capability allows for more flexibility in laying out the girder locations on the pier.

The pier model, now with bearings, will be used generate the two remaining piers for this example. Before doing so, however, the model must first be converted to a Bridge model type (the model is currently a General Pier model). To do so, click on the *Problem* data page and select the “Bridge” Problem Type and confirm the intended model transformation by clicking “Ok” at the dialog prompt. FB-MultiPier automatically generates a single bridge span and a matching pier at the end of the bridge span. The bridge model is shown in a Plan View and 3D View in **Figure 1.5**.

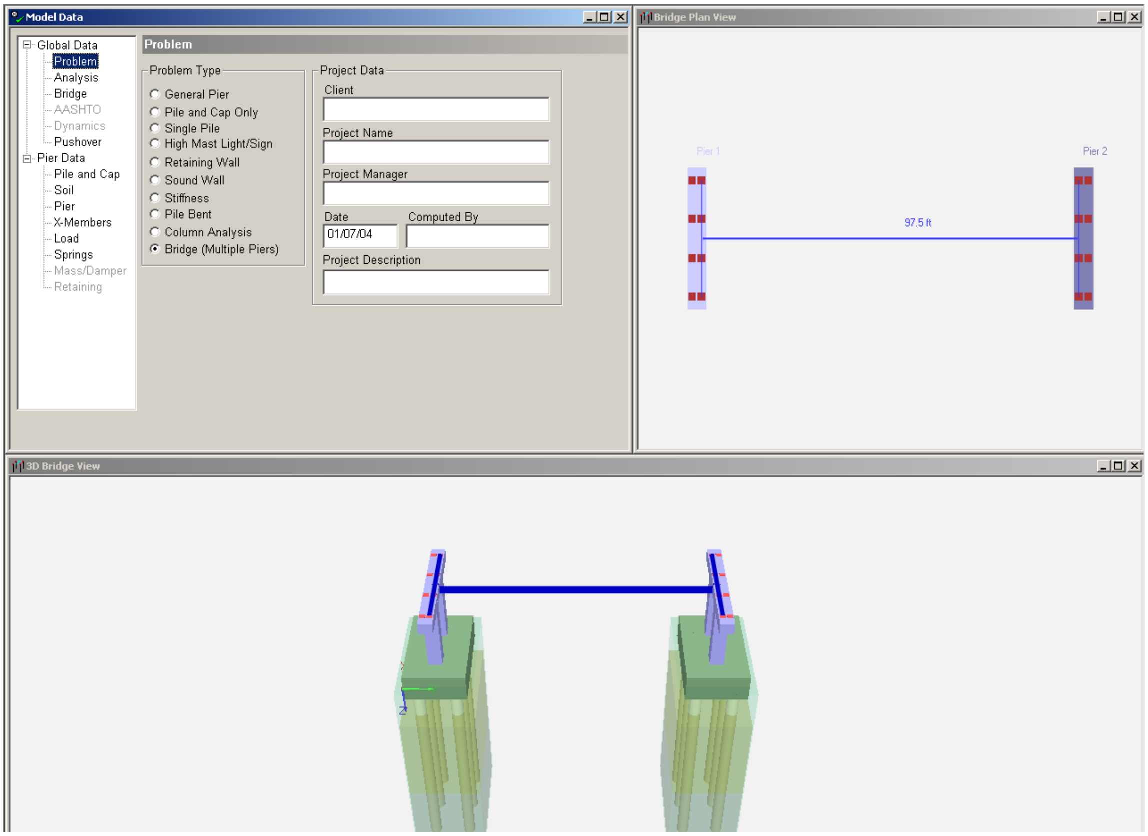


Figure 1.5 Single Span Bridge Model

FB-MultiPier models a bridge span using an equivalent beam element with section properties that represent the actual span. The span element connects to the bearing locations using rigid links in order to transfer the load from the superstructure to the pier. The span element properties will be modified shortly.

The pier generation and bridge layout capabilities are located in the *Bridge* data page. Select the "Bridge" item in the Model data tree view to view this page. **Figure 1.6** shows the options available for modeling a bridge. The page is divided into model data for the Substructure (the pier foundation) and the Superstructure (the bridge span).

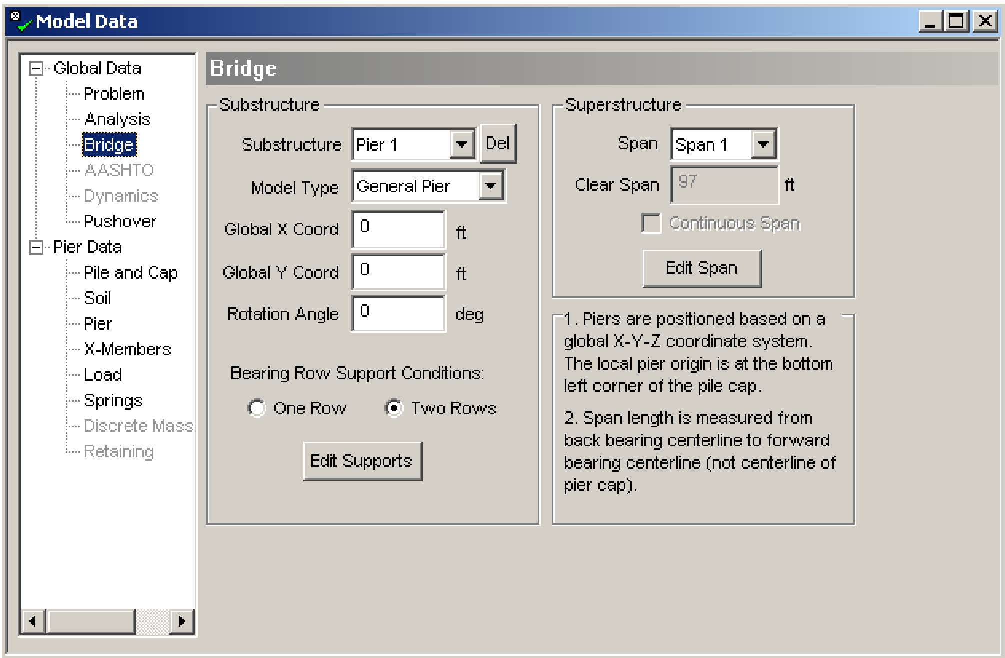


Figure 1.6 Bridge Data Page

In the Substructure group, the Substructure combo box allows the user to select a pier for editing. The Model Type combo box allows the user to select either a General Pier or Pile Bent model type for the current pier. The Global X and Y Coordinates are used to position the pier model and the Rotation Angle is used for rotating the pier about the vertical (global-Z) axis. Pier rotation is typically used in curved alignments or skewed bridges. The Continuous Span option models span continuity over a pier support. The Bearing Row Support Conditions allows for either One Row or Two Rows of bearings.

Clicking the "Edit Supports" button allows the user to verify and/or modify the boundary conditions at the bearing locations as shown in **Figure 1.7**. The Left/Right Bearing Row combo boxes allow the user to specify a pre-defined boundary condition for the bearing row. The program provides options for a Roller, Pin, Integral, or Custom boundary condition for each bearing row. Bearing rows at the beginning and end of the bridge are given the "N/A" designation because there are no girders associated with these bearing rows. Select a "Roller" Support Condition for the Right Bearing Row. Notice that each bearing direction in the Right Bearing Row has a "Constrain" boundary condition except for the Y-translation and the rotation about the X-axis (RX), which have a “Release” condition. This combination of boundary conditions models a roller connection at the first pier.

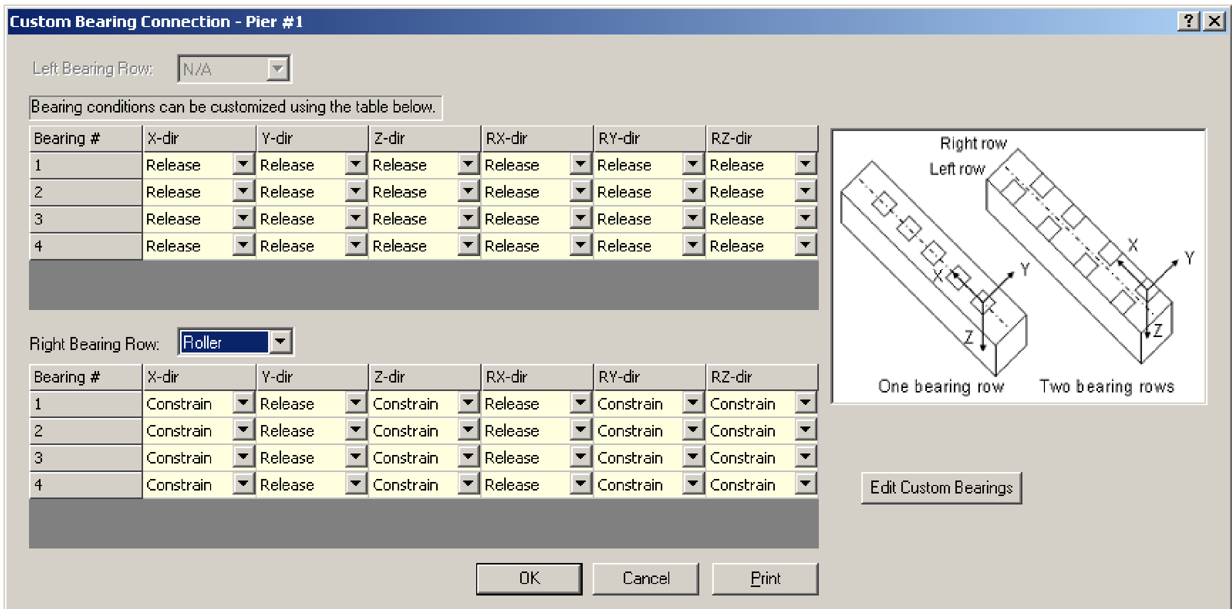


Figure 1.7 Custom Bearing Connection

Although not used in this example, custom boundary conditions can be entered by clicking the “Edit Custom Bearings” button. Custom boundary conditions are described using a load-displacement curve and can be applied to one or more bearing directions. Click “Ok” to close the Custom Bearing Connection dialog.

In the Superstructure group, the Span combo box allows the user to select a bridge span for editing. The “C/C Length” box shows the center-to-center length for the span (measured from the center of the right bearing row to the center of the left bearing row on the next pier). Click the “Edit Span” button to modify the bridge span properties. Enter the values shown in **Figure 1.8**.

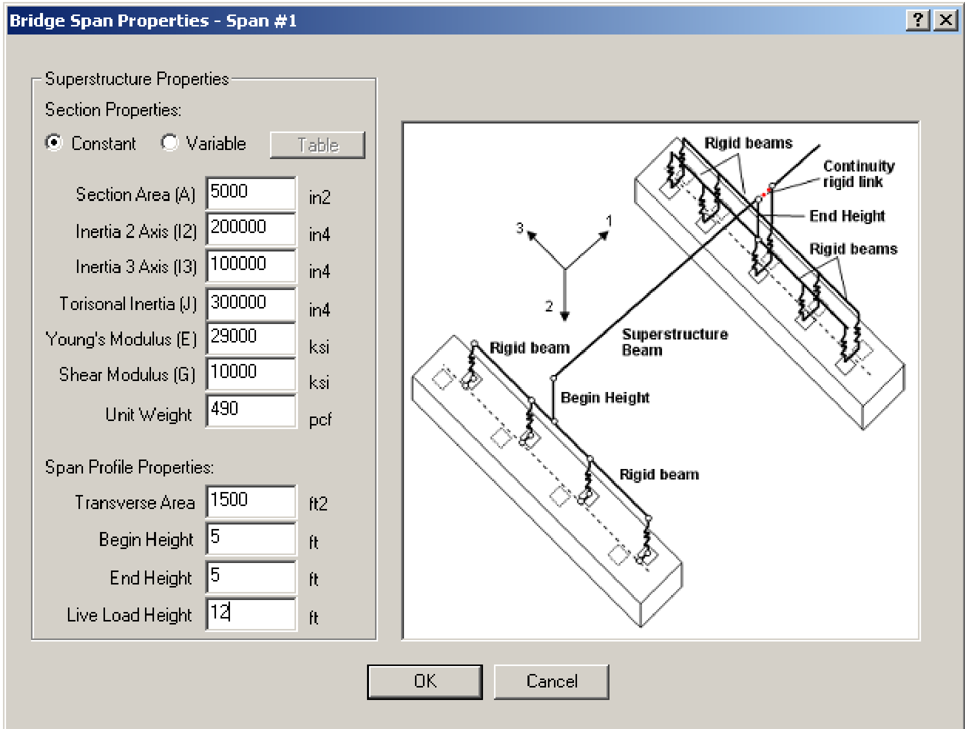


Figure 1.8 Bridge Span Properties

The Bridge Span Properties dialog is divided into two types of data input. The Section Properties group refers to the flexural properties of the equivalent bridge span element. The properties can either be constant or variable along the span. A table is provided for entering variable properties for the 10 equal-length elements along the span. The values enter in the Section Properties group reflect the entire bridge cross-section (i.e. girders, slab, parapets, etc.). Note that the values shown are for demonstration purposes and do not reflect values used in a specific bridge design. The Span Profile Properties group is used described the profile and height of the span. The Transverse Area is the exposed span area that will be used in the wind load application to the structure. The Begin Height and End Height are used to model the equivalent span element through the center of gravity of the span. These values are measure from the center of gravity of the pier cap to the center of gravity of the span. The Live Load Height is used in generating wind on live load cases. Click "Ok" to apply the values and return to the *Bridge* data page.

Bridge spans automatically connect one pier to the next. Therefore, to change the span length, the user must change the global coordinates of the pier. In the *Bridge* data page, first select "Pier 2" from the

Substructure combo box and then change the Global Y Coordinate from 100 ft to 150 ft to reflect the span length for this example. Click the "Edit Supports" button and select "Pin" for the Left Bearing Row in the Custom Bearing Connections dialog. Click "Ok" to apply the changes. The updated bridge model is shown in **Figure 1.9**. Note that the center to center span length is displayed as 147 ft since the span is measured from the bearing centerlines and not the center of the pier cap. Also note that the equivalent span element is modeled above the pier cap based on the center of gravity heights specified in the Bridge Span Properties dialog.

One more pier must be added to the model in order to complete the bridge generation process. To do so, select "Add Pier" from the Substructure combo box. A dialog will appear that gives options for the new pier. Select the Structure Type "General Pier" and select Pier 2 from the "Select Model" menu. A duplicate of pier 2 will automatically be generated and positioned based on the previous span length. This example calls for a span length of 100 ft for the second span. Enter 250 ft for the Global Y Coordinate for Pier 3. The updated two-span bridge model is shown in **Figure 1.10**.

This example calls for a continuous span over the middle pier. Select "Pier 2" from the list of substructures in the *Bridge* data page. Check the "Continuous Span" checkbox to model span continuity over Pier 2. This option will allow the internal bending moment to transfer between bridge spans. Notice that the span continuity option is not available for Pier 1 or Pier 3 because these piers are at the ends of the bridge. Span continuity can be verified by inspecting the span over a pier support in the 3D Bridge View window. The span line will be unbroken if span continuity is modeled.

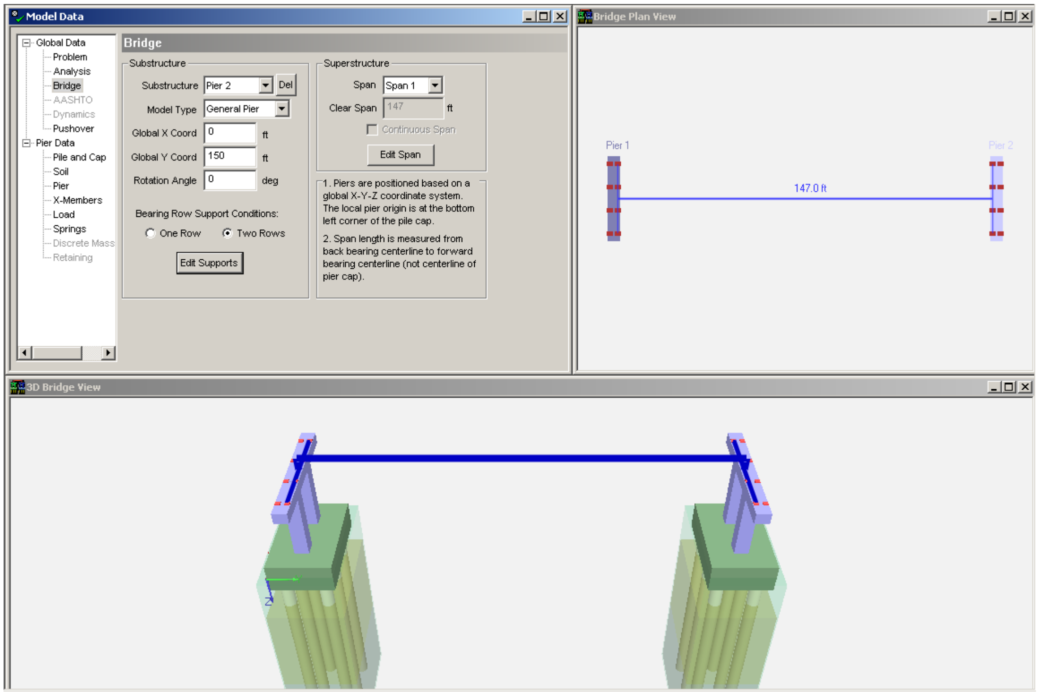


Figure 1.9 Increased Bridge Span

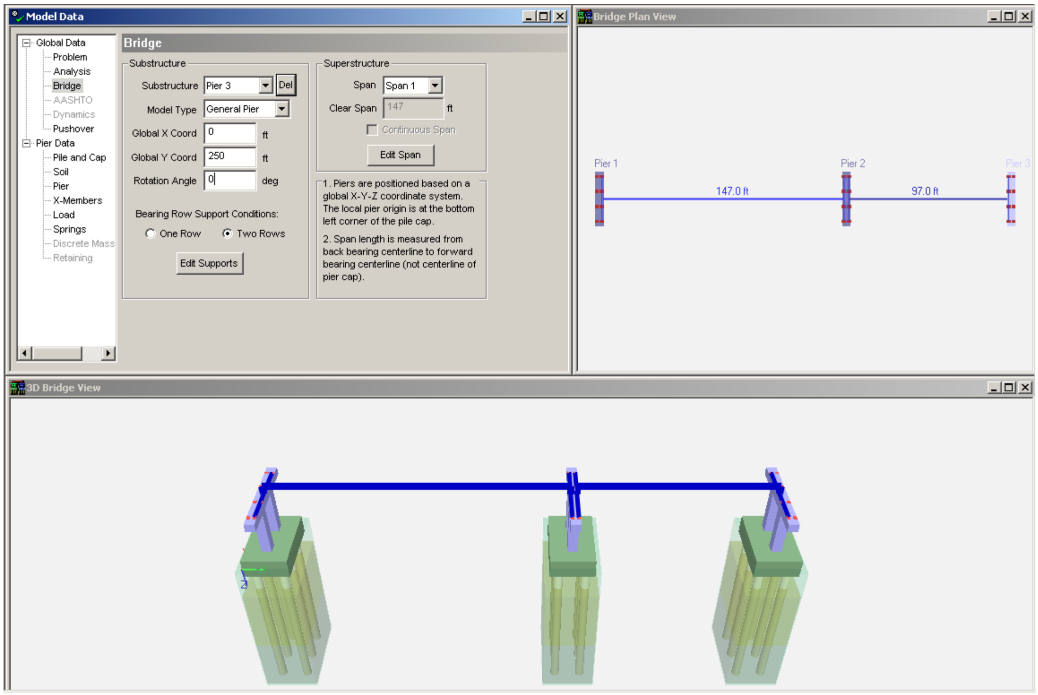


Figure 1.10 Two Span Bridge Model

Several bridge span and bearing details must be updated before the model is complete. First, select "Span 2" from the Span combo box in the Superstructure group. Click the "Edit Span" button and change the Transverse Area to "1000" ft^2 in the Bridge Span Properties dialog since the span is only 100 ft long. Click "Ok" to apply the properties. Second, in the Substructure Group, Pier #3, change the Bearing Row Support Conditions to "Roller" for the Left row. Note that the bearing conditions for the Pier #2 are set at "Integral" for both the Left and Right rows. Change both bearing rows in Pier #2 to "Pin."

Now that the bridge model has been created, it is easy to apply load cases to the full bridge. In this example, AASHTO LRFD load combinations will be used to determine the governing loads for the bridge. To begin, select the *Analysis* data page in the Model Data window and then check the "AASHTO Combinations" check box as shown in **Figure 1.11**. Doing so will enabled AASHTO load combination capabilities for the entire bridge.

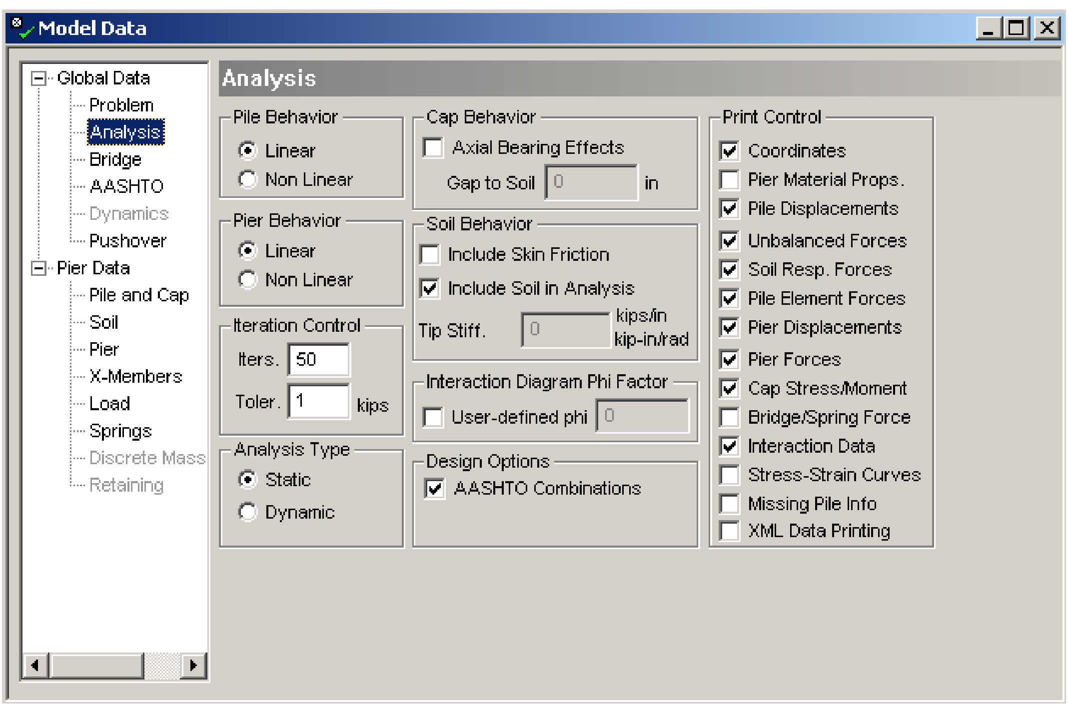


Figure 1.11 Analysis Data Page – Enabling AASHTO Load Combinations

AASHTO load cases are managed using the *AASHTO* data page. Select this page to view the various loading option, as shown in **Figure 1.12**. The AASHTO Load Factors group allows the use to edit the load factors used in the load combination equations. This can be done for either an LRFD analysis or an LFD analysis. The Limit States to Check group is used to specify which limit states to consider in the analysis as well as whether to reverse the loads and check maximum/minimum load factors (LRFD only). Finally, the Automated AASHTO Loads group is used to specify the types of loads to consider in the analysis. A Load Case Manager is provided in this group to add, modify, or delete AASHTO load cases for the model. This group also includes options for automatic generation of self weight, buoyancy, and wind load cases.

Clicking the "Load Case Manager" displays the AASHTO Load Manager dialog (**Figure 1.13**). The dialog shows the AASHTO load types that are available and the load types are currently defined for the model. Initially, the "Defined Load Cases" list is empty as no AASHTO load types have been added to the model in this example.

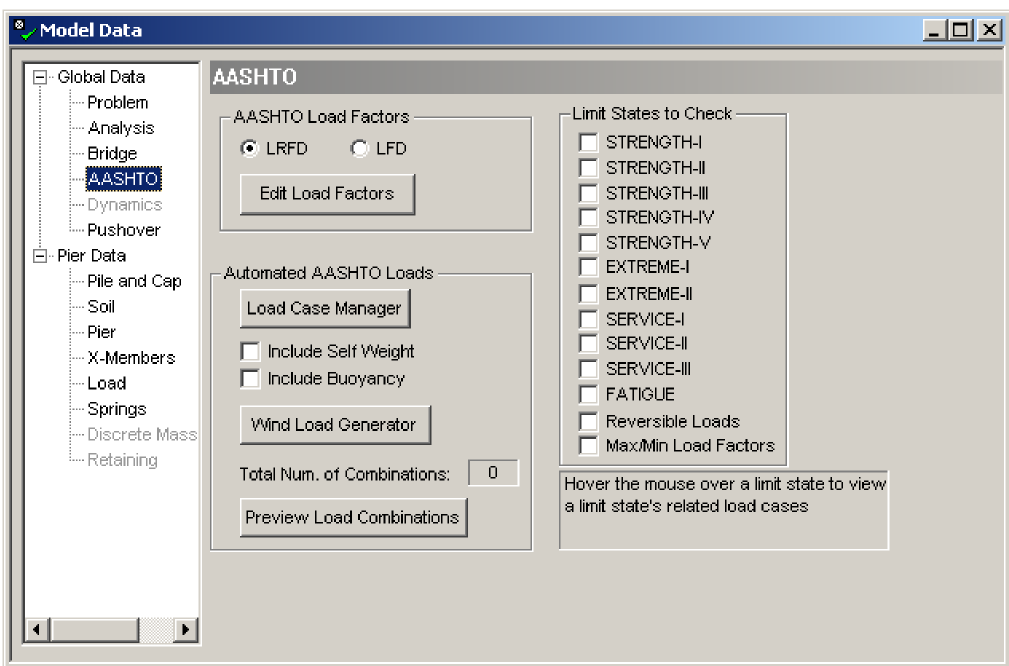


Figure 1.12 AASHTO Data Page

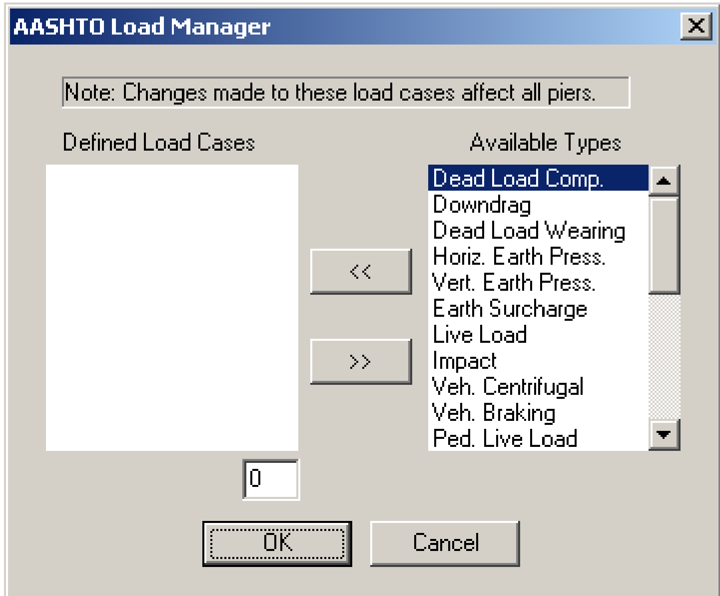


Figure 1.13 AASHTO Load Case Manager

This example will consider a simple bridge loading consisting of dead (self weight and buoyancy), live, and wind loads. Load cases are added to the model by selecting a load case from the "Available Types" list and then clicking the "<<" button to add the load case to the "Defined Load Cases" list. Do this for "Dead Load Comp.", "Water Load", "Live Load", "Wind on Structure", and "Vessel Collision" load cases. The resulting screen is shown in **Figure 1.14**.

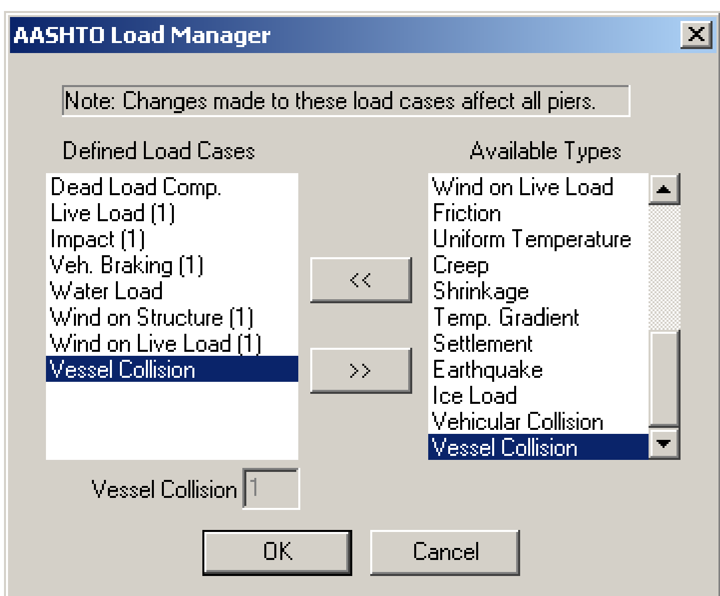


Figure 1.14 AASHTO Load Case Manager – With Dead, Live, and Wind Loads

Several observations can be made at this point. First, some load case types can have multiple load cases. These types have the number of currently defined load cases shown in parenthesis. Multiple load cases are enabled for live loads and wind loads. To change the number of load cases defined for a given load type, enter a new value in the box shown below the Defined Load Cases list. Second, certain loads are automatically generation based on the existence of either live loads or wind loads. For live loads, Impact and Vehicle Braking cases are automatically included with the live load. For wind loads, Wind Load on Structure and Wind Load on Live Load are added in tandem. Finally, when the number of live loads is changed, the number of Impact and Vehicle Braking cases is automatically updated. The same is true for wind load cases.

This example will consider the effect of 2 live load cases and 3 wind load cases. Click on the "Live Load" item in the Defined Load Cases list. Enter 2 in the edit box to change the number of live (and impact and braking) cases to 2. Next, click on the "Wind on Structure" item in the Defined Load Cases list and enter 3 in the edit box to change the number of wind load cases. The resulting screen is shown in **Figure 1.15**.

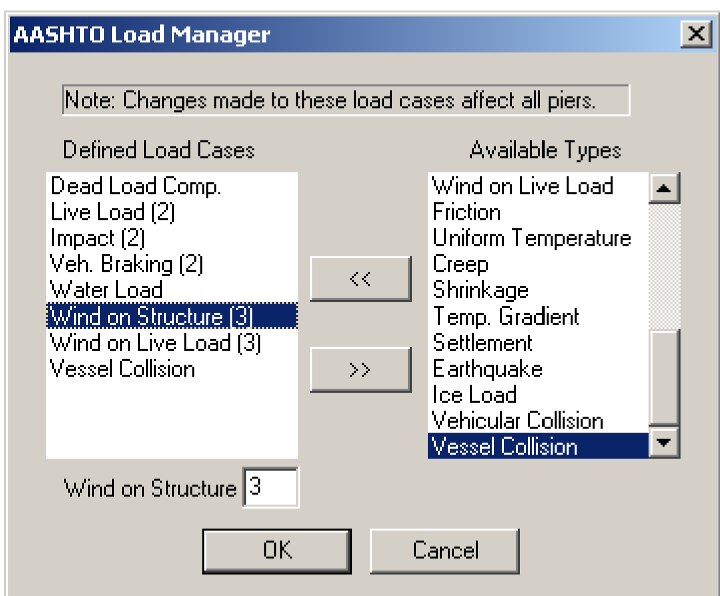


Figure 1.15 AASHTO Load Case Manager – Multiple Load Cases

After defining the load case types for the model, the loads can then be applied to the entire bridge. Click Ok to process the load changes to the model. A dialog will be displayed (**Figure 1.16**) to inform the use of the changes that will be made to the model. This summary window is important and requires the user to confirm the changes made to the model. Clicking "Yes" will modify the load cases for the *entire* bridge. Clicking "No" will abandon the proposed load case modifications and return the user to the AASHTO Load Manager. For this example, click "Yes" to confirm the changes and apply the load types to the entire bridge.

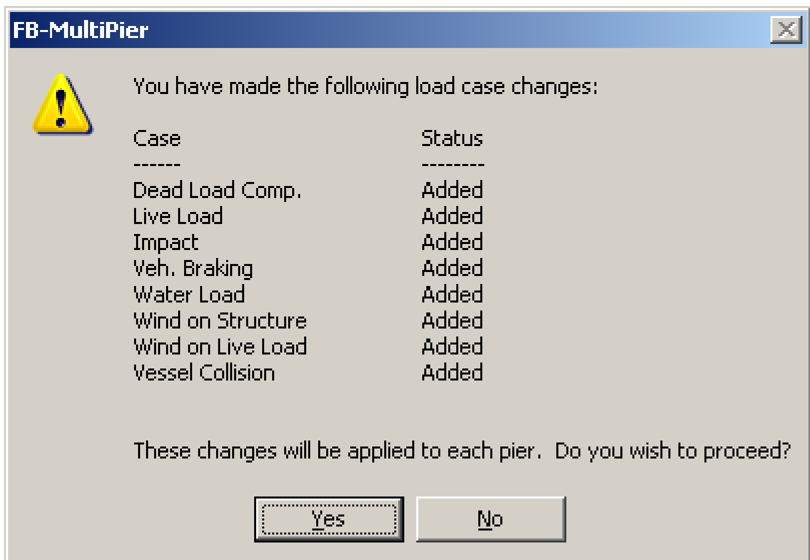


Figure 1.16 Confirming Load Case Modification for Entire Bridge

In addition to synchronizing the load types for all piers in the model, the AASHTO Load Manager also ensures that concentrated load placeholders are created at the bearing locations to accommodate loads applied from the superstructure. In this example, all of the defined load types except for the "Water Load" and "Vessel Collision" type, are expected to have loads applied at the bearing locations. The Dead Load of Components (self weight) is applied internally; therefore, no additional load needs to be applied to the bearings. The concentrated load placeholders are still provided; however, in case the user wishes to apply additional load. Wind loads will be automatically applied to the pier bearings using the Wind Load Generator on the *AASHTO* data page. The live loads will manually be applied to the bearing locations since the program does not currently contain a live load generator. The vessel collision load will be manually applied to the pier as well.

Wind Loads

As stated, FB-MultiPier can automatically apply wind loads to the entire bridge using the AASHTO Wind Load Generator. This can be done by clicking the “Wind Load Generator” button. This example will consider 3 wind load cases, with wind applied at 0, 30, and 60 degrees. Specify “3” for the Number of Cases and select “0”, “30”, and “60” for the wind angles as shown in **Figure 1.17**. The table of wind pressures shows the default values given in the AASHTO LRFD code. These values should be modified if necessary. Click the “Generate Wind Load Cases” button to apply the wind to the bridge and transfer the loads to the pier bearings. Click “Yes” and then “Ok” to confirm the wind load case generation.

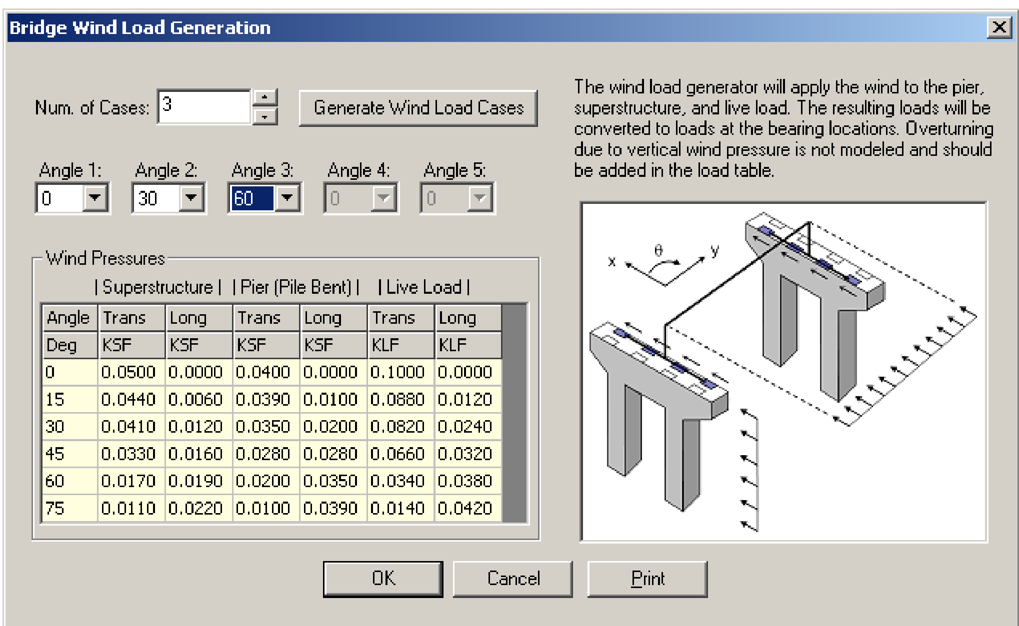


Figure 1.17 AASHTO Wind Load Generator

Before proceeding to the remaining load types, it is important to brief discuss the methodology used by the Wind Load Generator. As Figure 1.17 demonstrates, a uniform distributed load is applied perpendicular to each span for Wind Load on Structure cases. The magnitude of this load is based on the span profile area and the wind pressures in the Wind Pressures table and spans are assumed to be simply supported when applying the uniform load. A line load is also applied perpendicular to the each span for the Wind Load on Live Load cases. The magnitude of this load is based on the span length and wind pressures in the Wind Pressures table. Both the Wind Load on Structure and Wind Load on Live Load are applied at the center of gravity of the superstructure (as defined in the Bridge Span Properties dialog – Figure 1.8). A wind load is also applied to the pier using the cross-section properties of the pier column and pier cap and the wind pressures in the Wind Pressures table. All of the wind loads on the bridge are transformed into concentrated loads acting at the bearing locations for each pier. This can be confirmed by viewing the loads generated for the wind load cases. Note that only the generated wind loads are shown as bearing loads at the piers. **Figure 1.18** shows the wind loads generated at Pier #2 for the zero wind angle case.

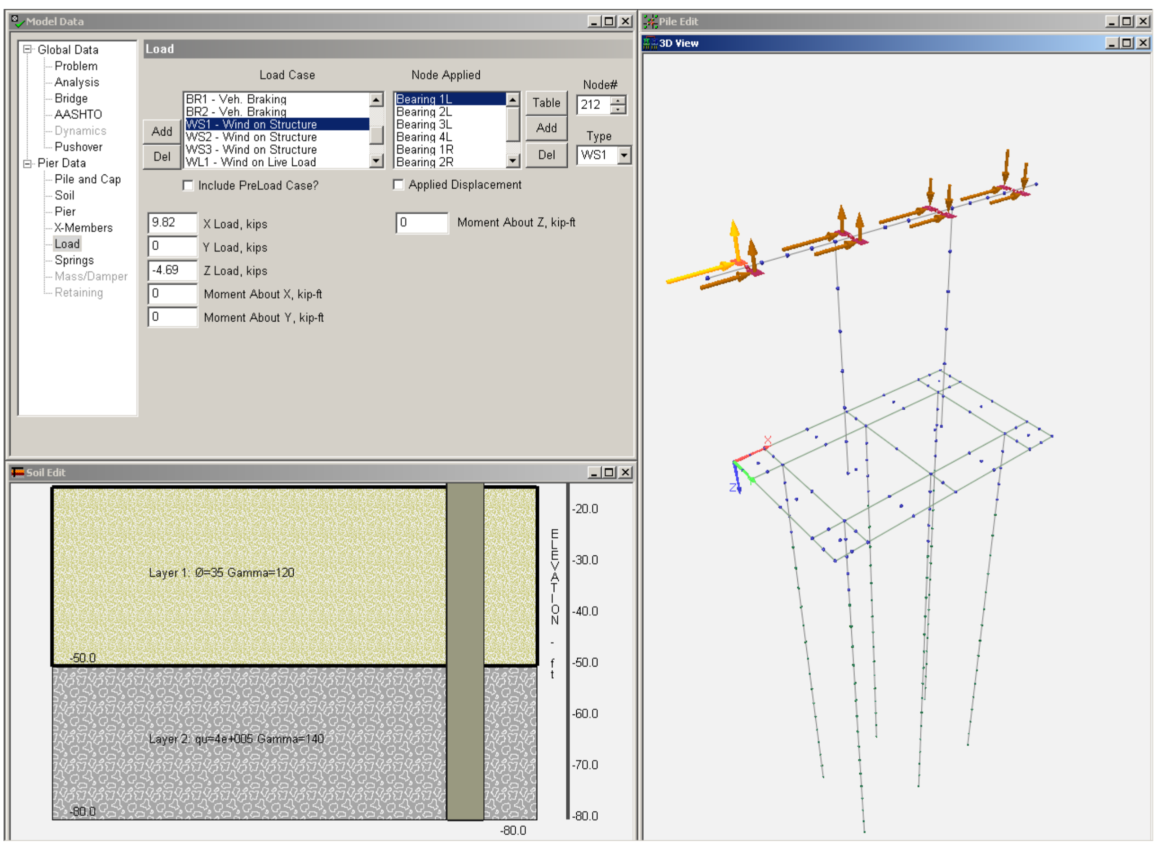


Figure 1.18 Wind Loads on Pier #2, 0^{o} Wind Angle

Live Loads

Live loads can be applied to the pier using the *Load* data page. The user can either select each live load case from the Load Case list or click on "Table" button to enter the loads in a spreadsheet format. **Table 1.2** provides the Live Load, Impact, and Vehicle Braking loads used in this example. These loads are applied to the interior pier support (Pier #2). Similar loads could also be applied to the other piers and is left as an exercise to the user.

Bearing	Live Load (kip) (Force Z)	Impact (kip) (Force Z)	Braking (kip) (Force Y)
(Case 1 – One Lanes Loaded)			
1L	120	40	15
2L	80	26	15
3L	0	0	15
4L	0	0	15
(Case 2 – Two Lanes Loaded)			
1L	100	33	20
2L	110	36	20
3L	105	35	20
4L	0	0	20

Table 1.2 Live Load, Impact, Braking

Select the *Load* data page in order to begin entering the live load data. Next, select "Pier #2" from the pier selection combo box in the toolbar. The AASHTO Load Table provides a quick method for entering the data if it is already in a tabular format. Click the "Table" button to view all load case data. First, verify that the list cases listed in the AASHTO Load Table match the load cases generated using the AASHTO Load Manager. After doing so, click on the "+" to expand and modify the load case "Live Load 1". Enter the live loads as shown in **Figure 1.19**. Next, enter the data given in Table 1.2 for Live Load 2 and the Impact and Vehicle Braking load cases. Note that the Live Load and Impact cases have vertical loads and the Vehicle Braking cases have horizontal (longitudinal) loads. The final AASHTO Load Table is shown in **Figure 1.20**. Two additional items are worth mentioning. First, the Wind on Structure and Wind on Live Load cases can be examined in this table to verify the results of the AASHTO Wind Load Generator. Second, the Load Types list on the right side of the dialog is grayed out.

This list is only accessible for single pier problems. For full bridge problems, the AASHTO Load Manager must be used to add and remove load cases.

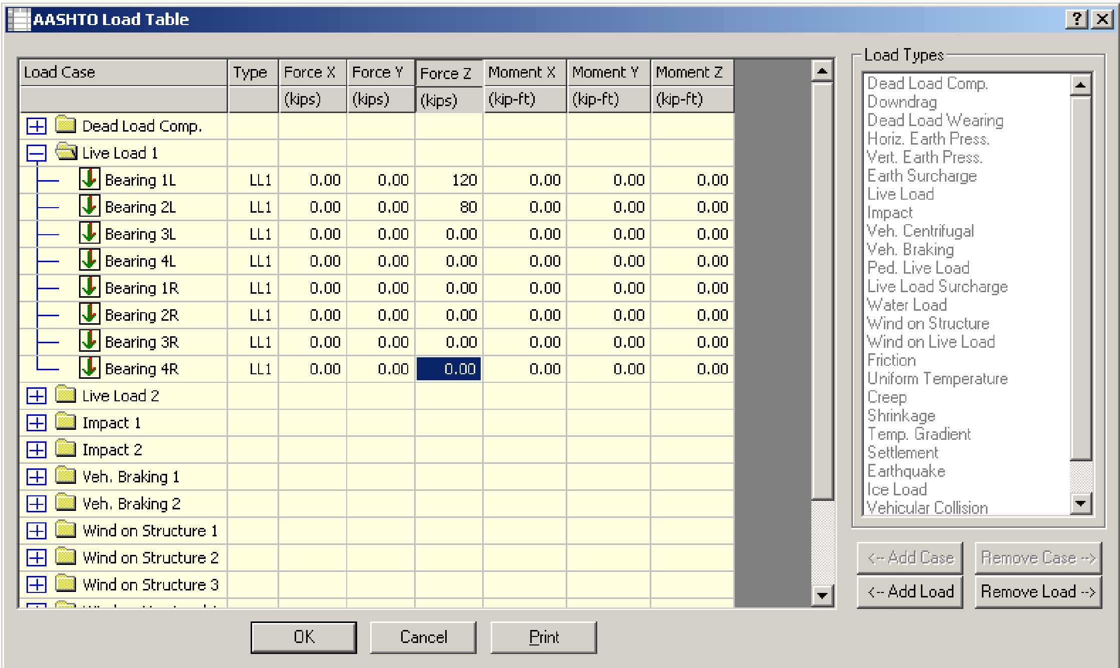


Figure 1.19 AASHTO Load Table – Live Loads

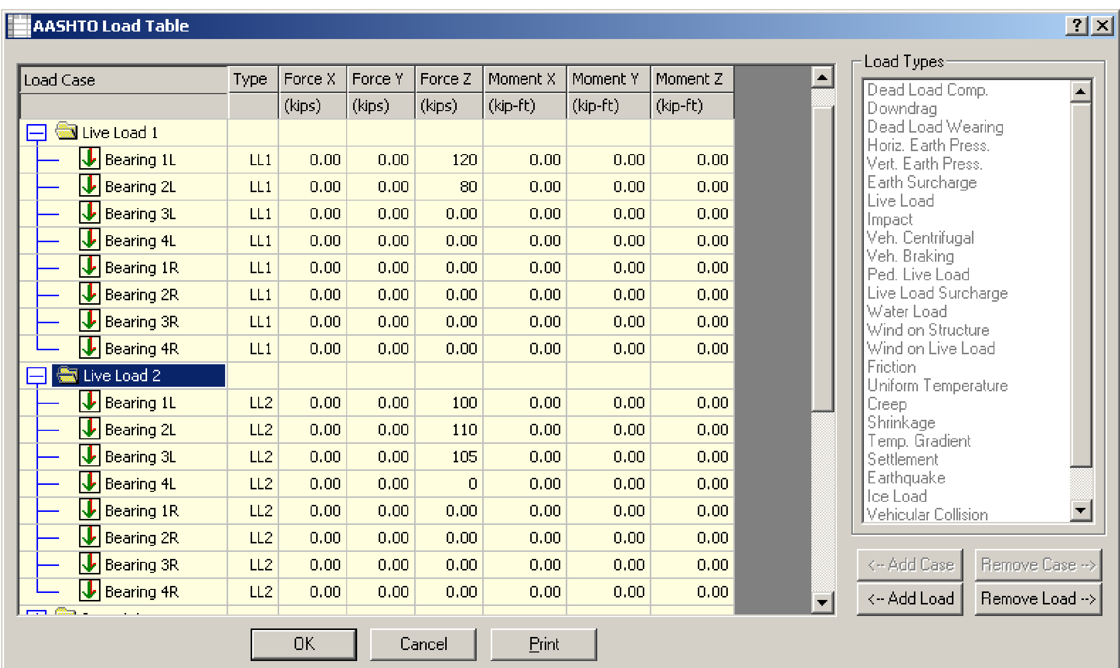


Figure 1.20 AASHTO Load Table – Live Load, Impact, Vehicle Braking

Vessel Collision Load

The vessel collision load value can also be specified in the AASHTO Load Table. Scroll down to the bottom of the table and expand the "Vessel Collision" load case. A default placeholder called "Node 1" is provided. Click on this placeholder to edit the node number. Change the number "1" to "38" to apply the vessel collision load at Node 38 (center of the pile cap). Enter "1000" kips for the Force X as shown in **Figure 1.21**. Note that although not demonstrated in this example, more concentrated loads can be added by clicking the "<- Add Load" button. Click "Ok" to apply the loads and return to the *Load* data page.

The final step before analyzing the model is to select the AASHTO Limit States to check. For this example, first select the *AASHTO* data page and then check "STRENGTH-I", STRENGTH-III", "STRENGTH-V", "EXTREME-II", and "SERVICE-I" as shown in **Figure 1.22**.

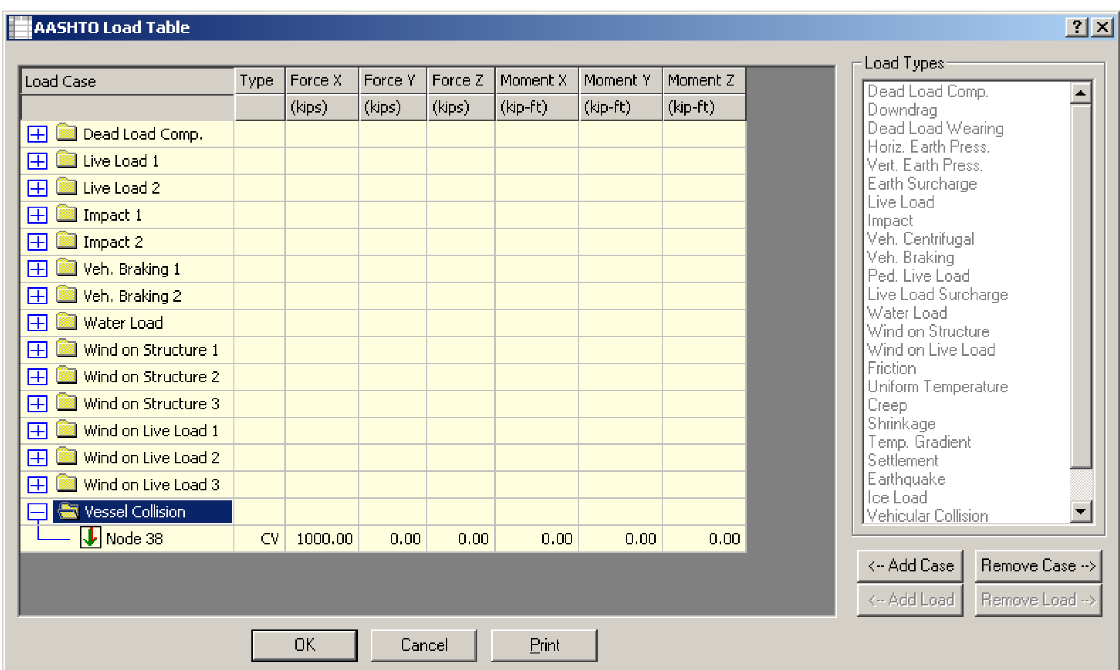


Figure 1.21 AASHTO Load Table – Vessel Collision

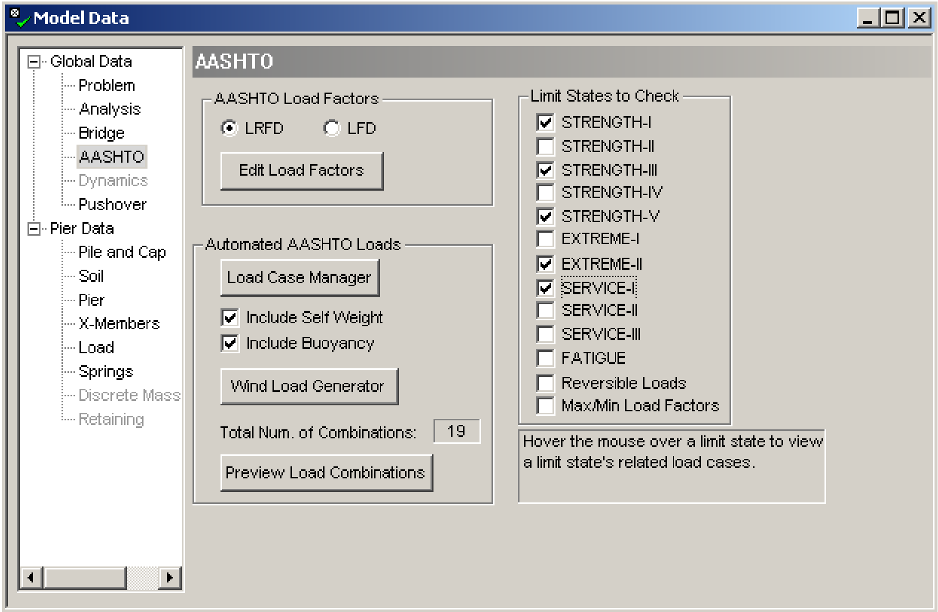
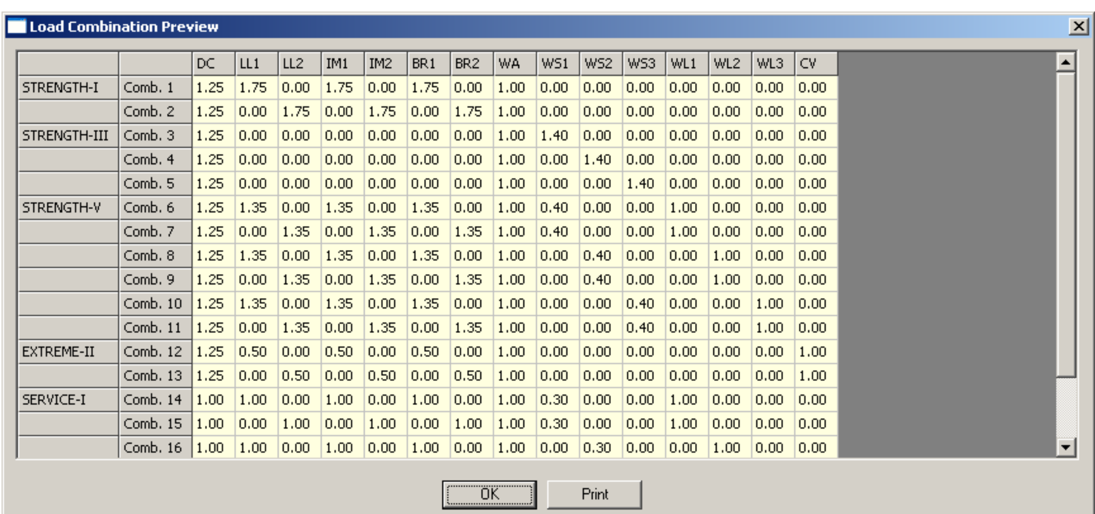


Figure 1.22 AASHTO Limit States

The AASHTO data page automatically updates the Number of Load Combinations count as the user selects Limit States to consider in the analysis. As shown in Figure 1.22, the current model analysis will consider 19 load combinations. The equations for each of these combinations can be previewed in advance of the analysis by clicking the "Preview Load Combinations" button. **Figure 1.23** shows the current load combinations.


Load Combination Preview

		DC	LL1	LL2	IM1	IM2	BR1	BR2	WA	WS1	WS2	WS3	WL1	WL2	WL3	CV
STRENGTH-I	Comb. 1	1.25	1.75	0.00	1.75	0.00	1.75	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
	Comb. 2	1.25	0.00	1.75	0.00	1.75	0.00	1.75	1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
STRENGTH-III	Comb. 3	1.25	0.00	0.00	0.00	0.00	0.00	0.00	1.00	1.40	0.00	0.00	0.00	0.00	0.00	0.00
	Comb. 4	1.25	0.00	0.00	0.00	0.00	0.00	0.00	1.00	0.00	1.40	0.00	0.00	0.00	0.00	0.00
	Comb. 5	1.25	0.00	0.00	0.00	0.00	0.00	0.00	1.00	0.00	0.00	1.40	0.00	0.00	0.00	0.00
STRENGTH-V	Comb. 6	1.25	1.35	0.00	1.35	0.00	1.35	0.00	1.00	0.40	0.00	0.00	1.00	0.00	0.00	0.00
	Comb. 7	1.25	0.00	1.35	0.00	1.35	0.00	1.35	1.00	0.40	0.00	0.00	1.00	0.00	0.00	0.00
	Comb. 8	1.25	1.35	0.00	1.35	0.00	1.35	0.00	1.00	0.00	0.40	0.00	0.00	1.00	0.00	0.00
	Comb. 9	1.25	0.00	1.35	0.00	1.35	0.00	1.35	1.00	0.00	0.40	0.00	0.00	1.00	0.00	0.00
	Comb. 10	1.25	1.35	0.00	1.35	0.00	1.35	0.00	1.00	0.00	0.00	0.40	0.00	0.00	1.00	0.00
	Comb. 11	1.25	0.00	1.35	0.00	1.35	0.00	1.35	1.00	0.00	0.00	0.40	0.00	0.00	1.00	0.00
EXTREME-II	Comb. 12	1.25	0.50	0.00	0.50	0.00	0.50	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00	1.00
	Comb. 13	1.25	0.00	0.50	0.00	0.50	0.00	0.50	1.00	0.00	0.00	0.00	0.00	0.00	0.00	1.00
SERVICE-I	Comb. 14	1.00	1.00	0.00	1.00	0.00	1.00	0.00	1.00	0.30	0.00	0.00	1.00	0.00	0.00	0.00
	Comb. 15	1.00	0.00	1.00	0.00	1.00	0.00	1.00	1.00	0.30	0.00	0.00	1.00	0.00	0.00	0.00
	Comb. 16	1.00	1.00	0.00	1.00	0.00	1.00	0.00	1.00	0.00	0.30	0.00	0.00	1.00	0.00	0.00

OK Print

Figure 1.23 Load Combination Preview

This completes the modeling phase for the bridge. To analyze the model under the applied loads, click the (Analyze) button in the toolbar.

Results can be viewed after reaching a converged solution to the problem. Results are presented per pier and the pier selection combo box in the toolbar is used to switch between piers. Click the (Pile Results) button to view the maximum results for the piles. The pile group for the interior pier support is of particular interest since it has the live load application at the support. Select "Pier #2" from the pier selection combo box in the toolbar to view the results for the interior pier. The load combinations with the vessel collision load are most likely to produce the maximum results. On this assumption, select "EXTREME-II" from the limit state combo box in the Plot Display Control window. This will highlight the pile with the maximum demand/capacity ratio and show the force results for the governing load combination in the EXTREME-II limit state. The resulting plots are shown in **Figure 1.24**.

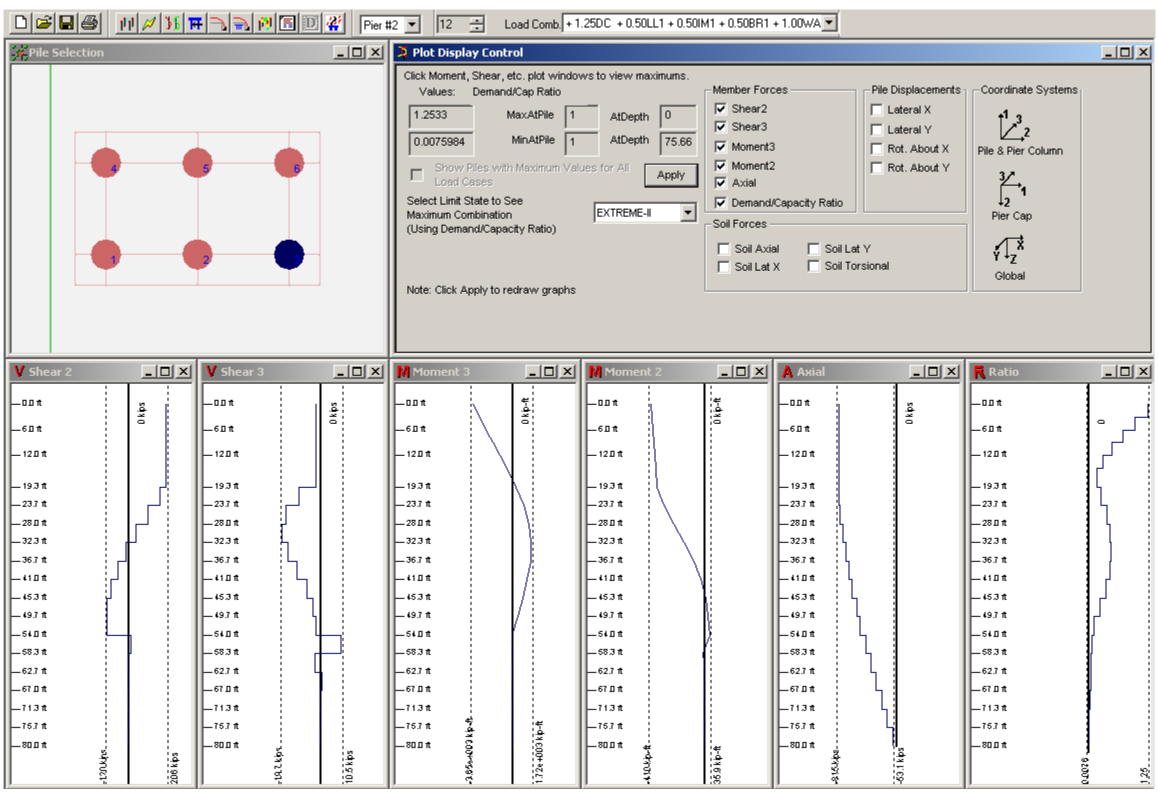
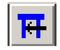


Figure 1.24 Maximum Pile Results – EXTREME-II

To view the pier column results, click the  (Pier Results) button. The limit state combo box can again be used determine the maximum load combination result for each limit state. In this example, the EXTREME-II limit state has the maximum demand/capacity ratio for the pier columns, although not by much. The other limit states have similar results. This is due to the fact that the vessel collision load is applied at the pile cap, so most of the load transfers into the foundation and not into the pier columns. The resulting plots are shown in **Figure 1.25**.

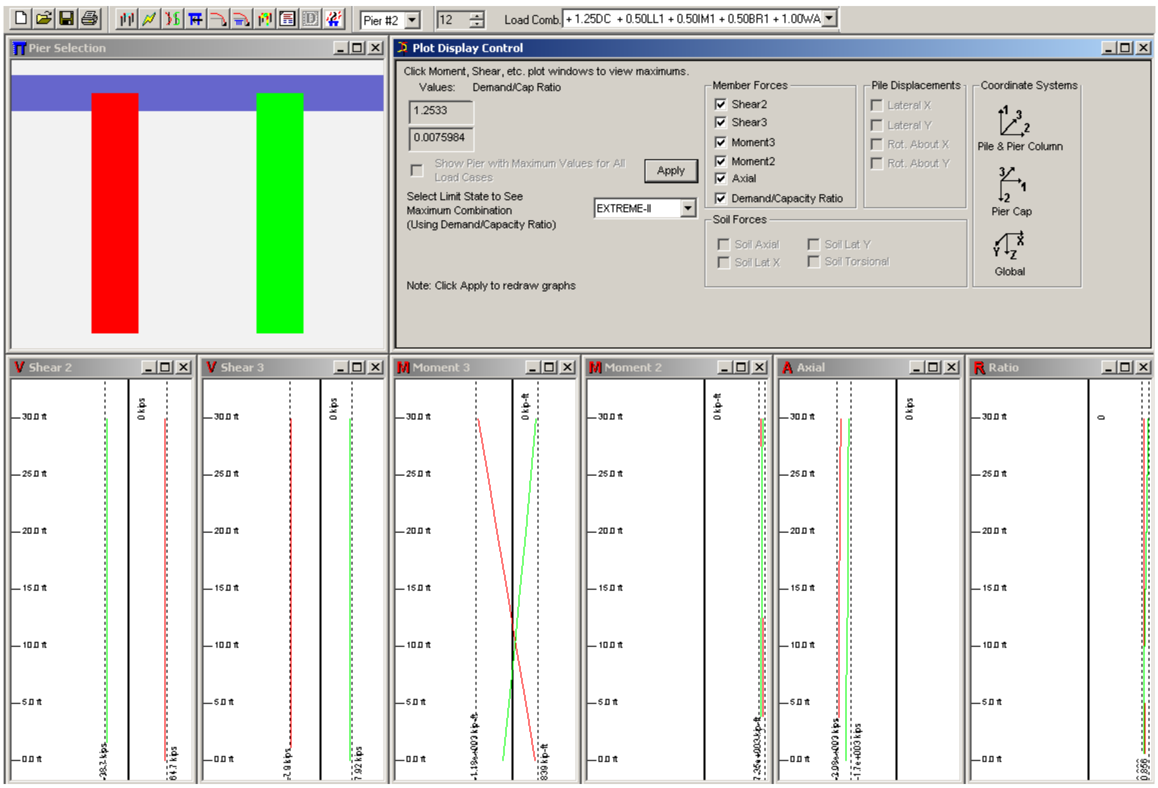


Figure 1.25 Maximum Pier Column Results – EXTREME-II

The deformed shape of the pier and bridge model can provide additional insight into the model behavior. Click the  (3D Results) button to view the deformed shape for Pier #2. The deformed pier is shown in **Figure 1.26**. Note that the red markers at the pile heads indicate plastic hinging zones. The pile capacity has been exceeded at these plastic hinge zones. Right-clicking the mouse in the 3D Results window and selecting “Bridge View” from the popup menu will show the 3D view of deformed shape for the entire bridge. This view is shown in **Figure 1.27**.

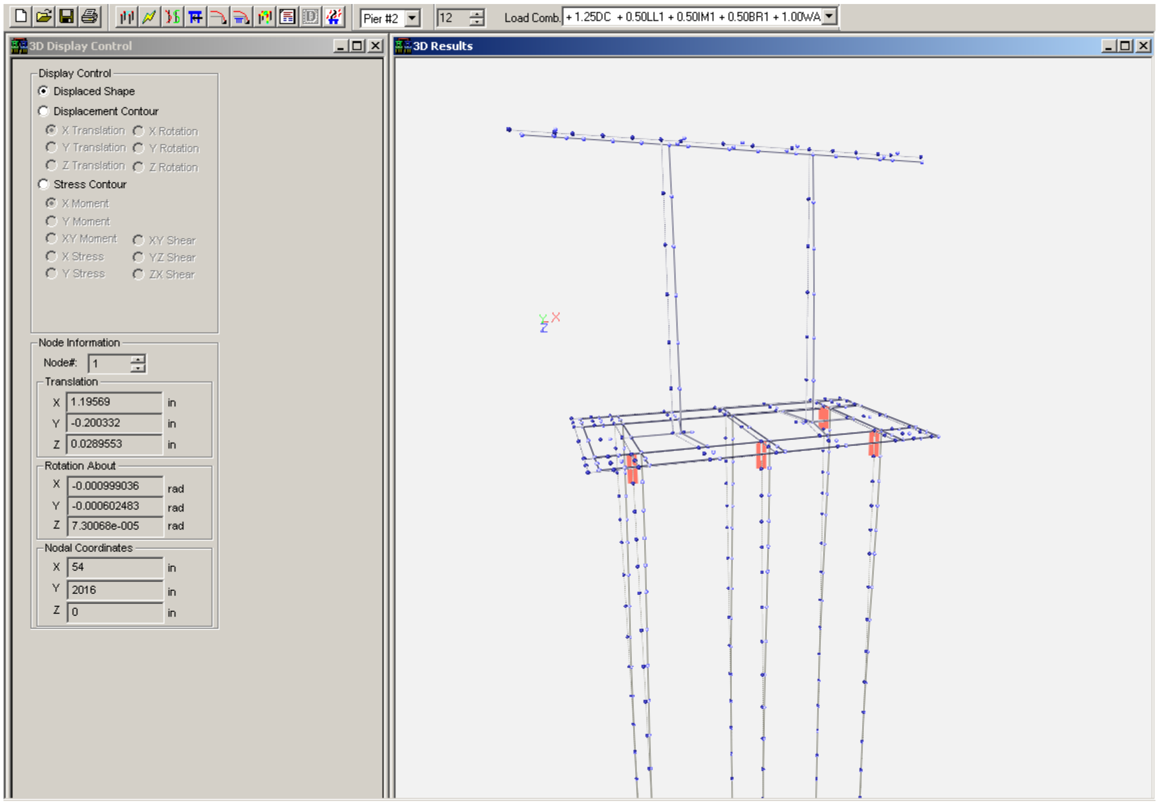


Figure 1.26 Pier #2 Deformed Shape – EXTREME-II

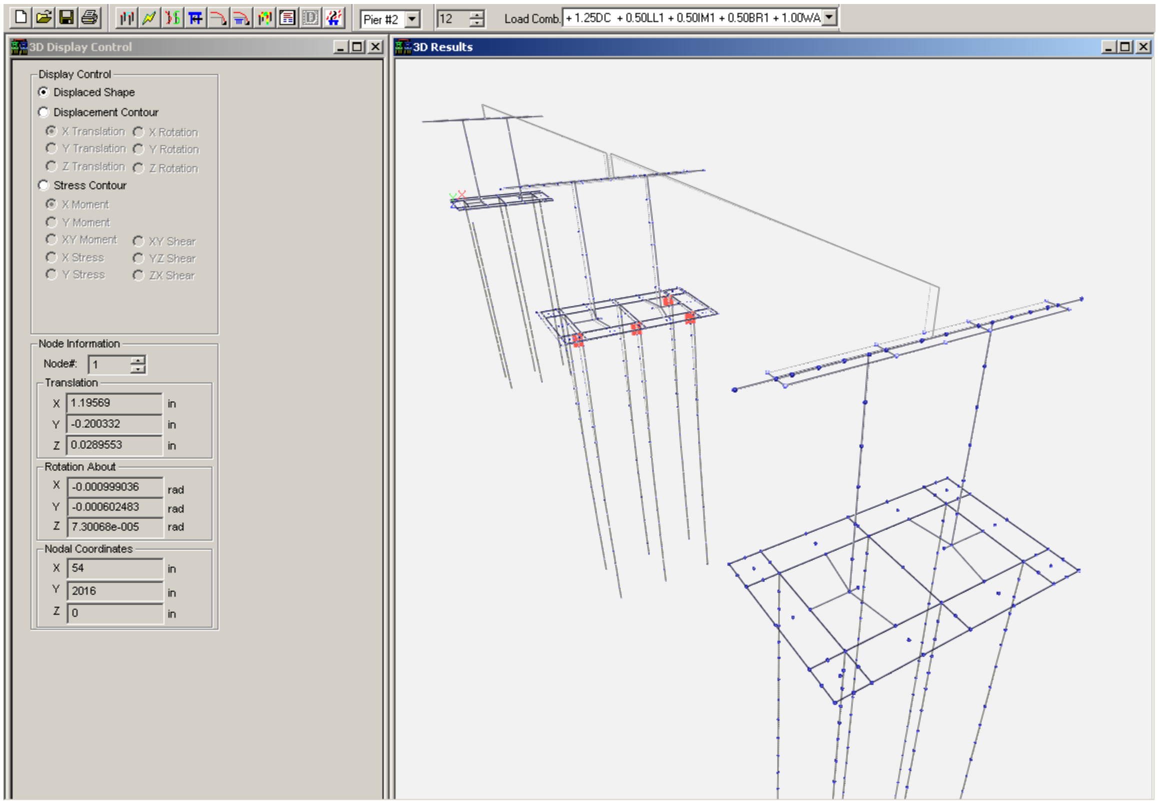


Figure 1.27 Bridge Deformed Shape – EXTREME-II

This concludes Example MP-1.

MP-2. HIGHWAY OVERPASS BRIDGE MODEL

Shown in **Figure 2.1** is a two-span highway overpass bridge, which will be modeled in **Example MP-2**. The bridge is supported by pile bent abutments and a general pier interior support. The bridge is primarily subjected to vertical loads, but there is some concern about a possible vehicle collision force on one of the columns in the pier. The default Bridge model type will be used in creating the bridge model for this example.

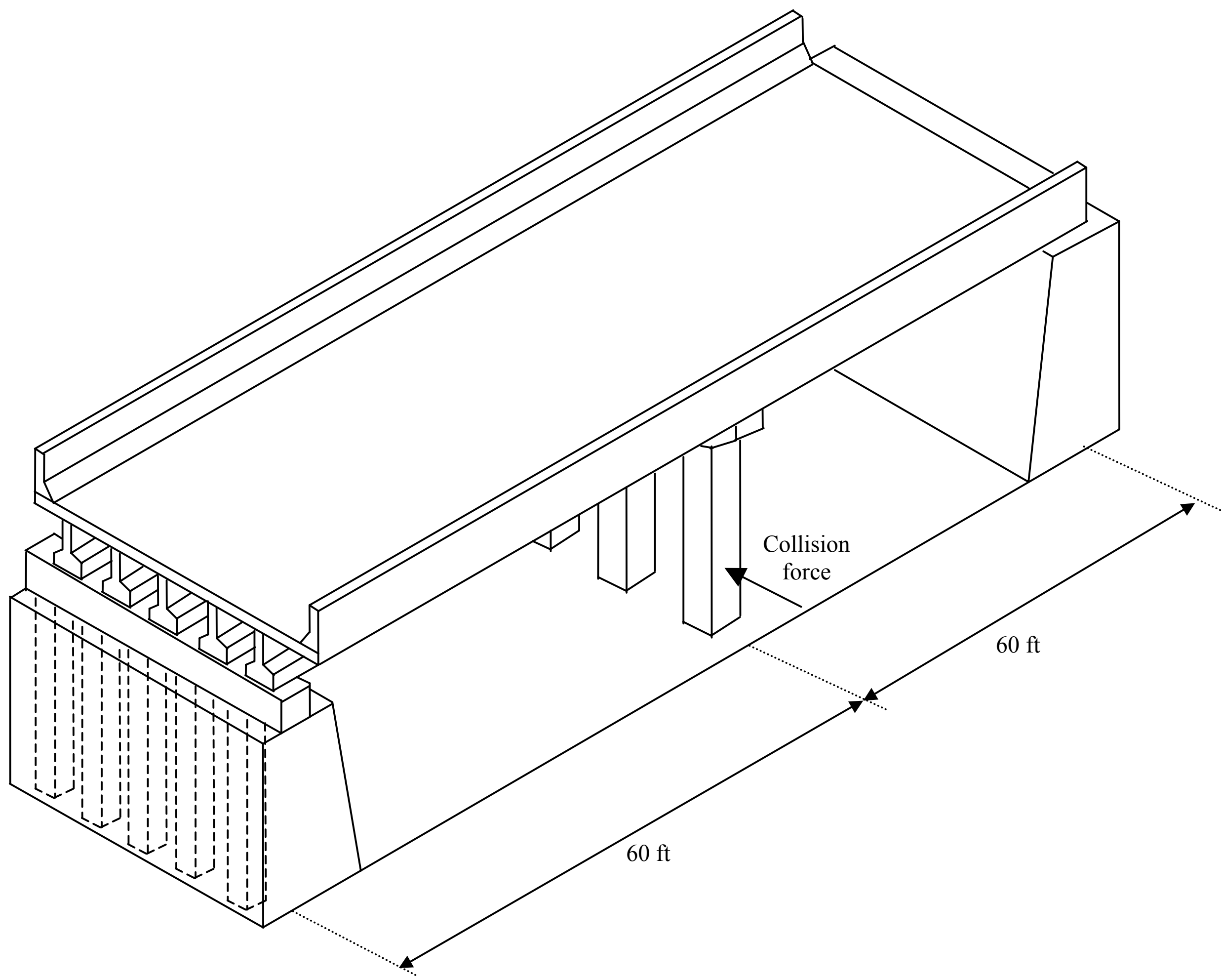


Figure 2.1 Example MP-2, Two-Span Highway Overpass Structure

Select New from the File menu and choose Bridge (Multiple Piers) from the New Problem Type screen as shown in **Figure 2.2** and click "Ok"**.** The program will provide a default single-span Bridge model, as shown in **Figure 2.3**.

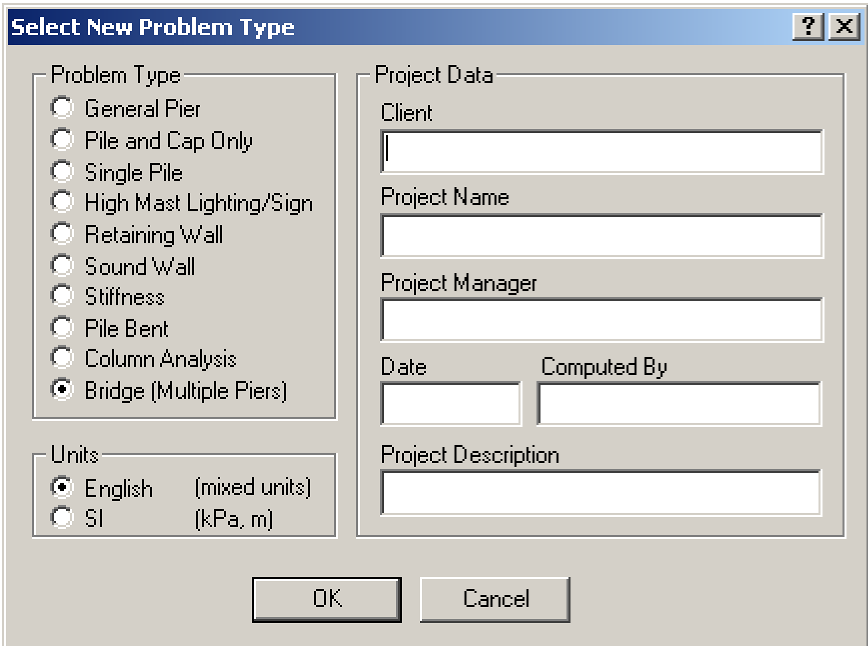


Figure 2.2 Example MP-2, Two-Span Highway Overpass Structure

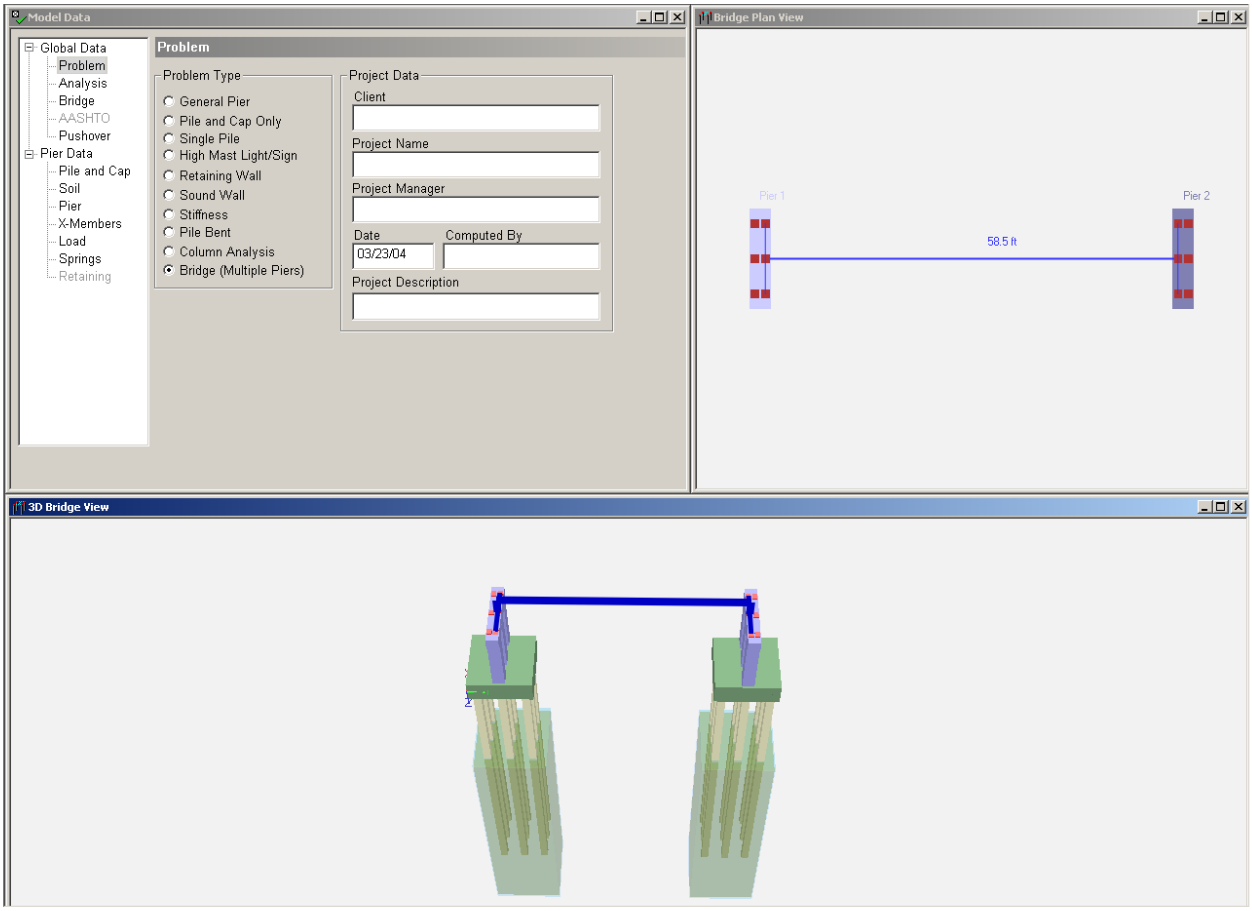


Figure 2.3 Default Bridge Model

The bridge in this model consists of two pile bent abutments and a general pier interior support. To convert the first pier to a pile bent, click on the *Bridge* data page in the Model Data window. The first pier is automatically selected as the active pier. Select "Pile Bent" from the Model Type combo box to change the pier to a pile bent. Click "Ok" to confirm the model transformation. Select "Default Pile Bent" when prompted to Select Model. The resulting model type transformation looks a little unusual in the 3D Bridge view window at first because the bent cap and pier cap elevations are not the same. Also note that the span has been removed because the number of bearing locations do not match. This will be corrected shortly. To create the third pier, select "Add Pier" from the Substructure combo box. In the Substructure dialog that appears select "Pile Bent" for structure type and "Pier 1 Bent" from the Model type menu. The program will automatically copy the pier data from the first pier to the third pier. Note that when adding a pier, the new pier automatically becomes the active pier. To change the currently selected (third) pier to a pile bent model type, select “Pile Bent” from the Model Type combo box. The resulting changes are shown in **Figure 2.4**.

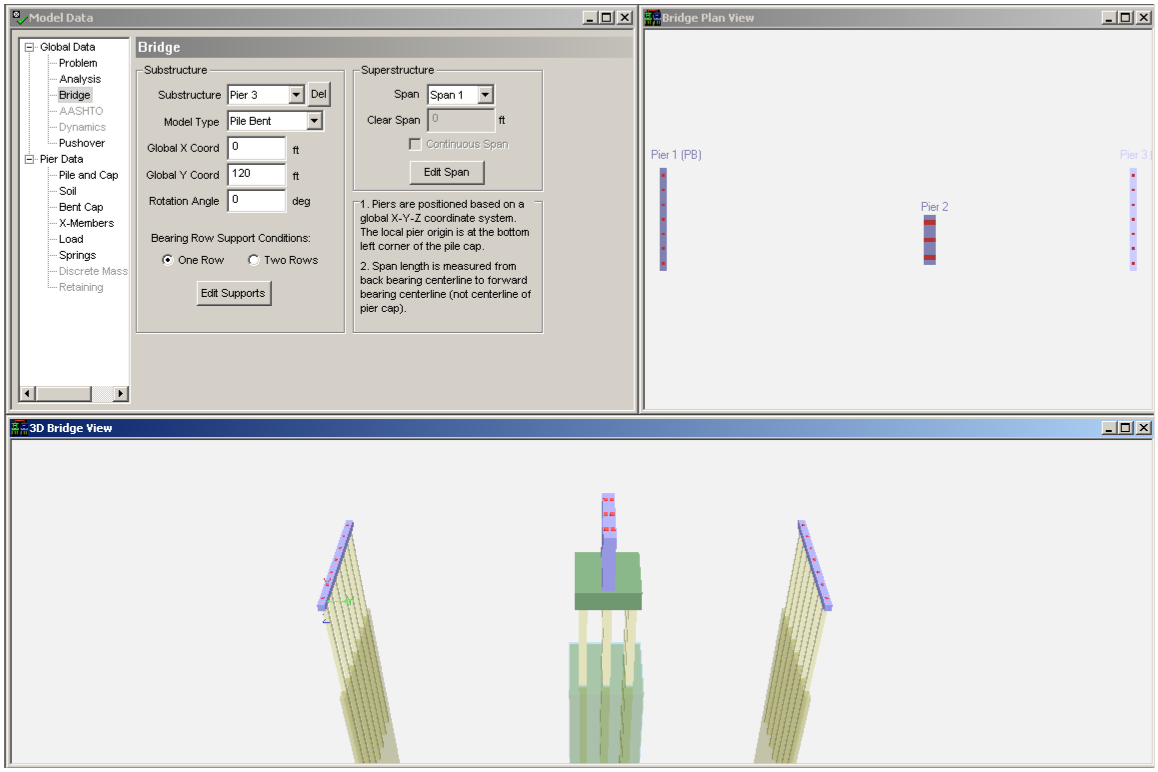


Figure 2.4 Bridge Model with Pile Bent Abutments

The next step in the bridge modeling is to adjust the elevations of each foundation to model a level bridge span. This can be accomplished by changing the elevation of the interior pier support (or by changing the elevations of the abutments). To do so, select “Pier 2” from the Substructure combo box in the *Bridge* data page. Now select the *Pile and Cap* data page in the Model Data tree view. The pier cap is currently 18 feet higher than the bent caps at the abutments. Enter “-18” feet for the Head/Cap Elevation in the Pile Cap Data section. This will move the entire pier downward 18 feet. Finally, click on the *Soil* data page to modify the soil layer elevation to bury the pile cap. Change the Top of Layer elevation to “-15” feet. Click on the *Bridge* data page again to view the entire bridge model. When done the screen should like **Figure 2.5**.

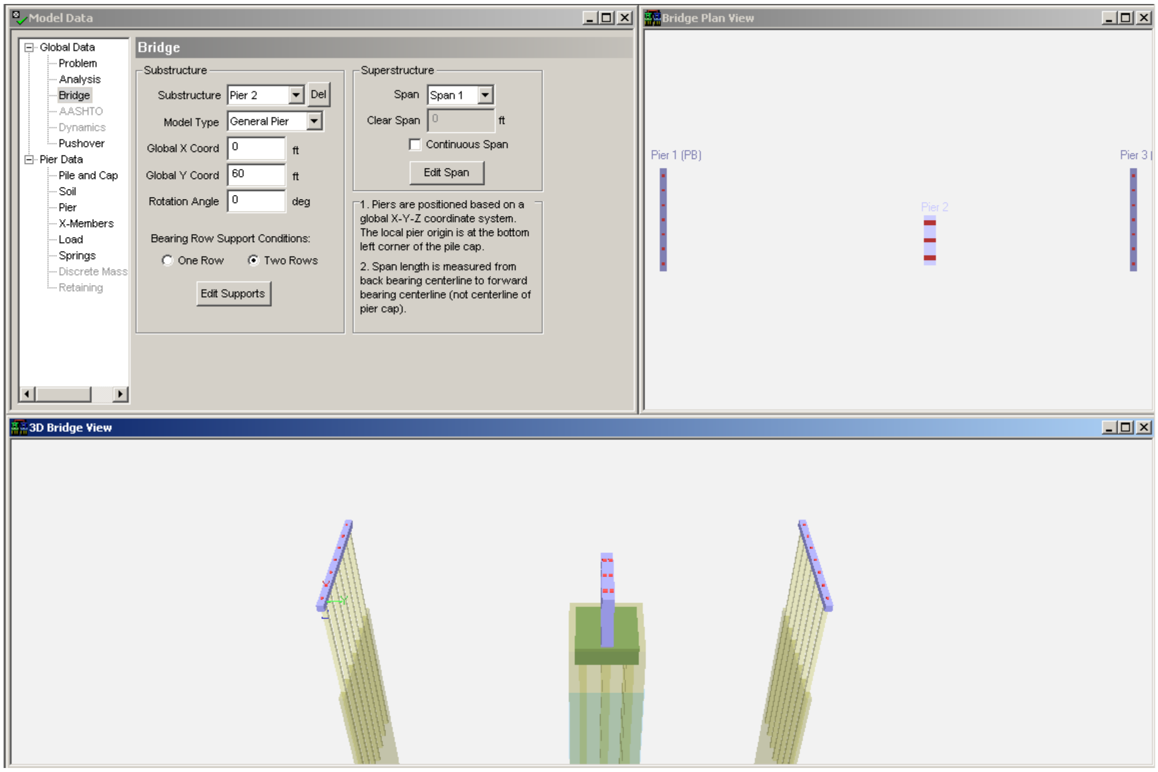


Figure 2.5 Pier with Modified Elevation

The soil layer elevations for the abutments should also be modified in this example to incorporate the soil-structure interaction behavior for the soil backfill. First, select “Pier 1” from the Substructure combo box in the *Bridge* data page and then select the *Soil* data page to change the soil layer elevation for the first abutment. Change the Top of Layer elevation to “0” feet to model the soil up to the bent cap elevation. Select “Pier #3” from the pier selection combo box in the toolbar or click on Pier 3 in the Bridge Plan View window. Now that the last abutment is selected, again change the Top of Layer elevation to “0” feet. When done the screen should like **Figure 2.6**.

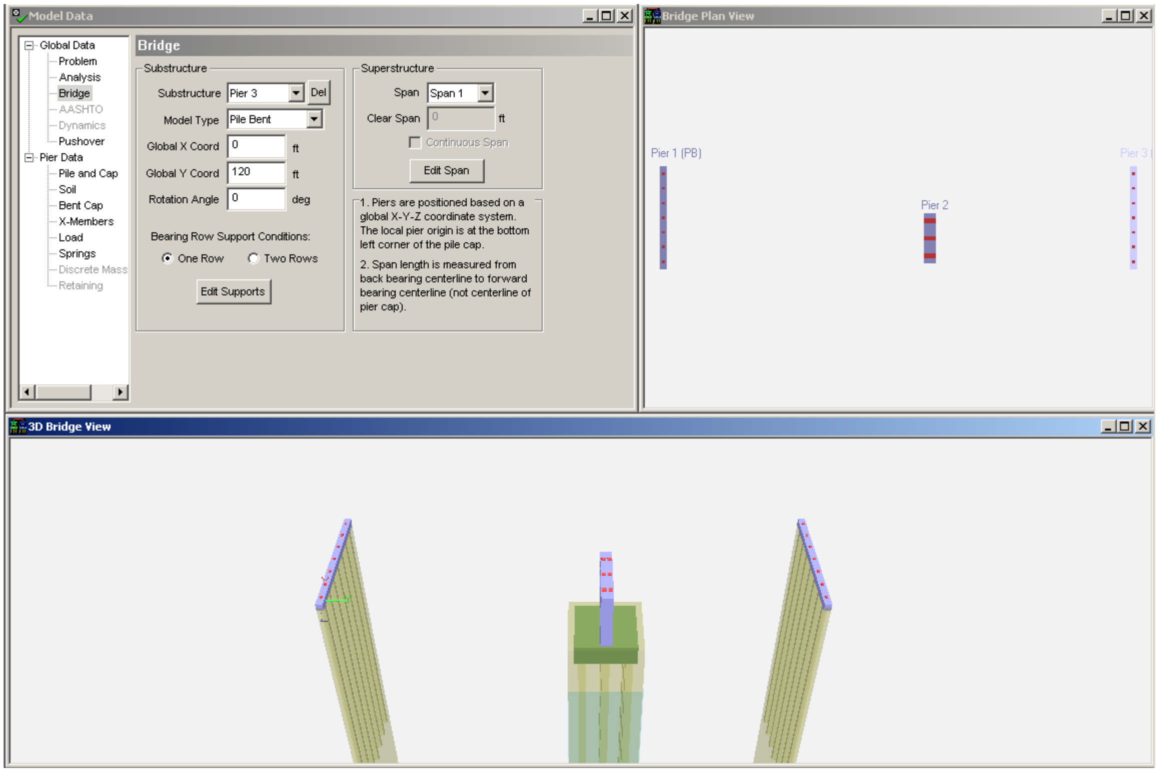


Figure 2.6 Bridge with Modified Elevations

The dimensions of the abutments and pier foundation can now be modified to accommodate the girder positions for the bridge spans. This example has 5 girders spaced at 6 ft on center. To specify the bearing locations for the first abutment, first select "Pier #1" from the pier selection combo box in the toolbar. Once the first abutment is selected, select the *Bent Cap* data page. Click the "Bearing Locs." to specify the bearing locations. Enter the values shown in **Figure 2.7**, the Bearing Locations dialog. Click "Ok" to apply the changes to the bearing locations. To specify the bearing locations for the last abutment, first select "Pier #3" from the pier selection combo box in the toolbar. Again, enter the values shown in Figure 2.7.

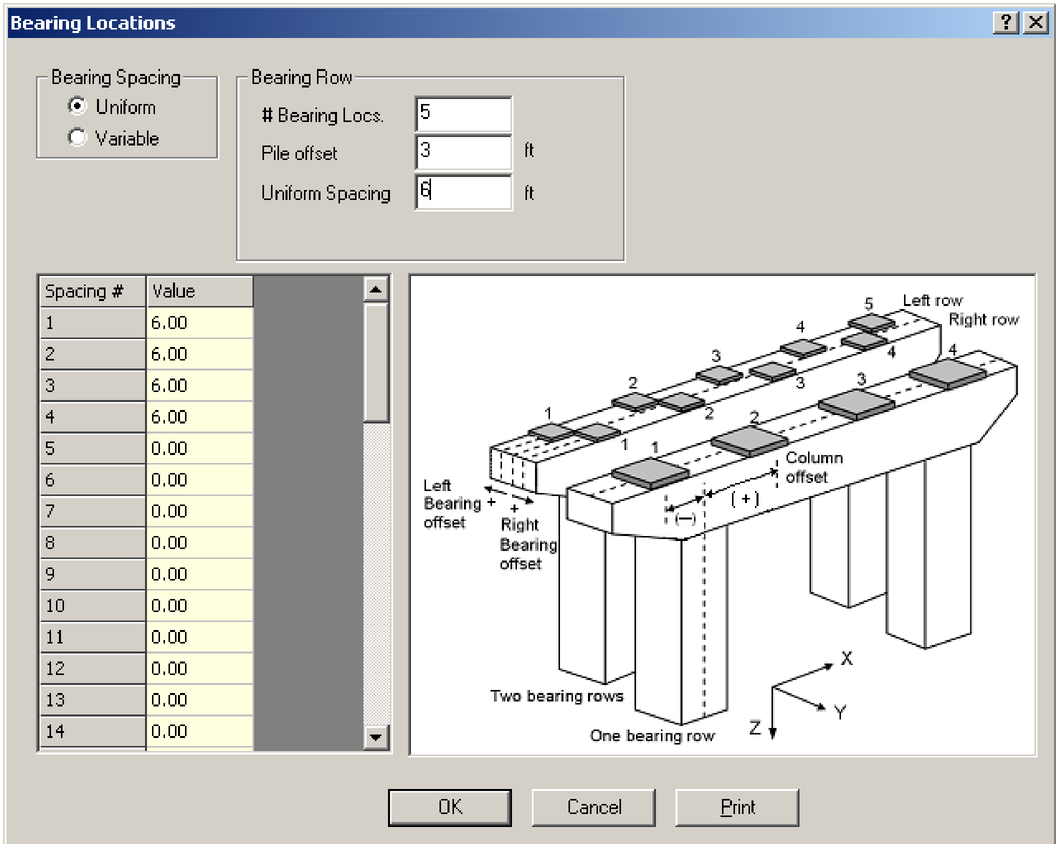


Figure 2.7 First Abutment Bearing Locations

The program will warn the user if the number of bearings at the beginning and end of the span do not match. This is currently the case, since the interior pier support only has three bearing locations. To change the number of bearing locations, select “Pier #2” from the pier selection combo box in the toolbar. The pier column spacing must first be adjusted to accommodate five bearings at a spacing of 6 feet on center. Select the *Pier* data page and enter “12” feet for the Col 1-2 spacing as shown in **Figure 2.8**. Also, enter “12” feet for the Col 2-3 spacing. Finally, click the “Bearing Locs.” to specify the bearing locations. Enter “5” Bearing Locations, “0” Pile Offset, “6” Uniform Spacing and “0.75” Bearing Offset for both bearing rows.

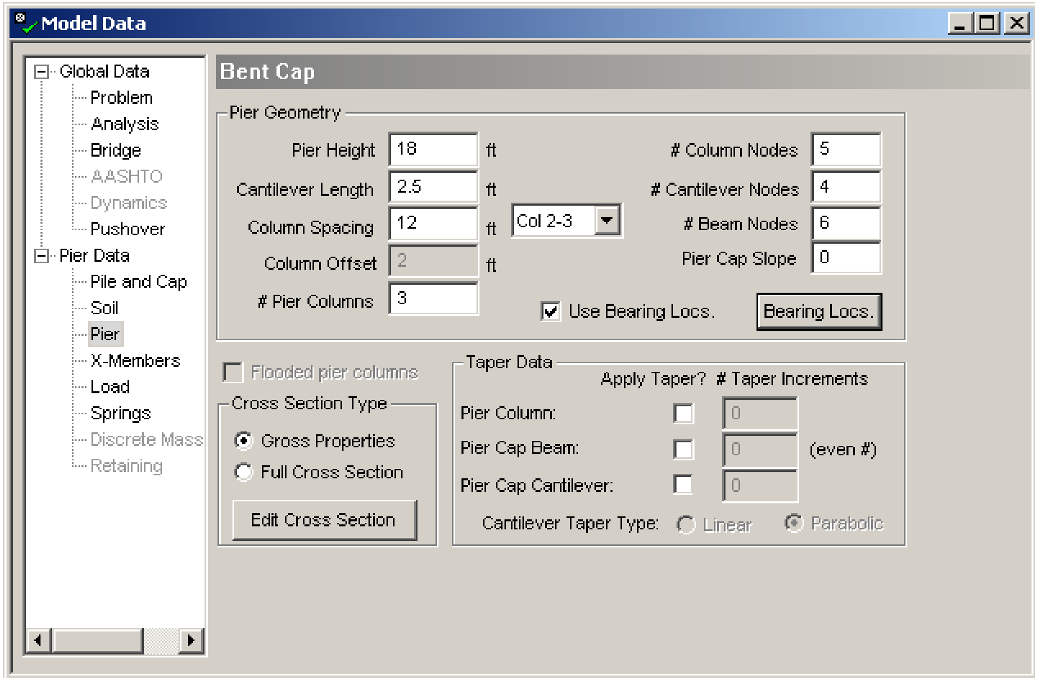


Figure 2.8 Modifying Pier Column Spacing

The pile group geometry must also be modified to accommodate the increase in pier column spacing. To do so, select the *Pile and Cap* data page and then change the X-Direction Grid Points to 7. Click "Yes" to include piles at all of the new grid points.

The foundation layout must now be adjusted before completing the full bridge model. The span length in the default bridge model is 60 feet. This value must be modified for the first span to match the example problem description. For pile bent models, the global coordinate is referenced to the center of the bent cap (since there is no pile cap). To model a 60 foot span, the interior pier support must be moved closer to the first abutment so that the center-to-center distance from the bent cap to pier cap is 60 feet. The amount of movement is equal to the half the width of the pile cap, a distance of 8 ft. To update the layout, select the *Bridge* data page and "Pier 2" and then enter "52" feet for the Global Y Coordinate. For Pier 3, the Global Y Coordinate does not need to be modified since the distance from the bottom left corner of the pile cap in the interior support to the center of the bent cap (in the last abutment) is 60 feet. Also, in the current model, the interior pier support is not exactly aligned with the end abutments due to the difference in the foundation layouts. To align the foundations (to make a straight bridge), the interior pier support must be moved 1 foot in the global X direction. To align the two spans, enter “1” foot for the Global X Coordinate. The aligned bridge model is shown in **Figure 2.9**. Note the gap in the superstructure element model over the interior pier support. This gap is present to indicate that span continuity is not present over the pier support. Upon further inspection (in the *Bridge* data page), Integral supports are used at the pier support.

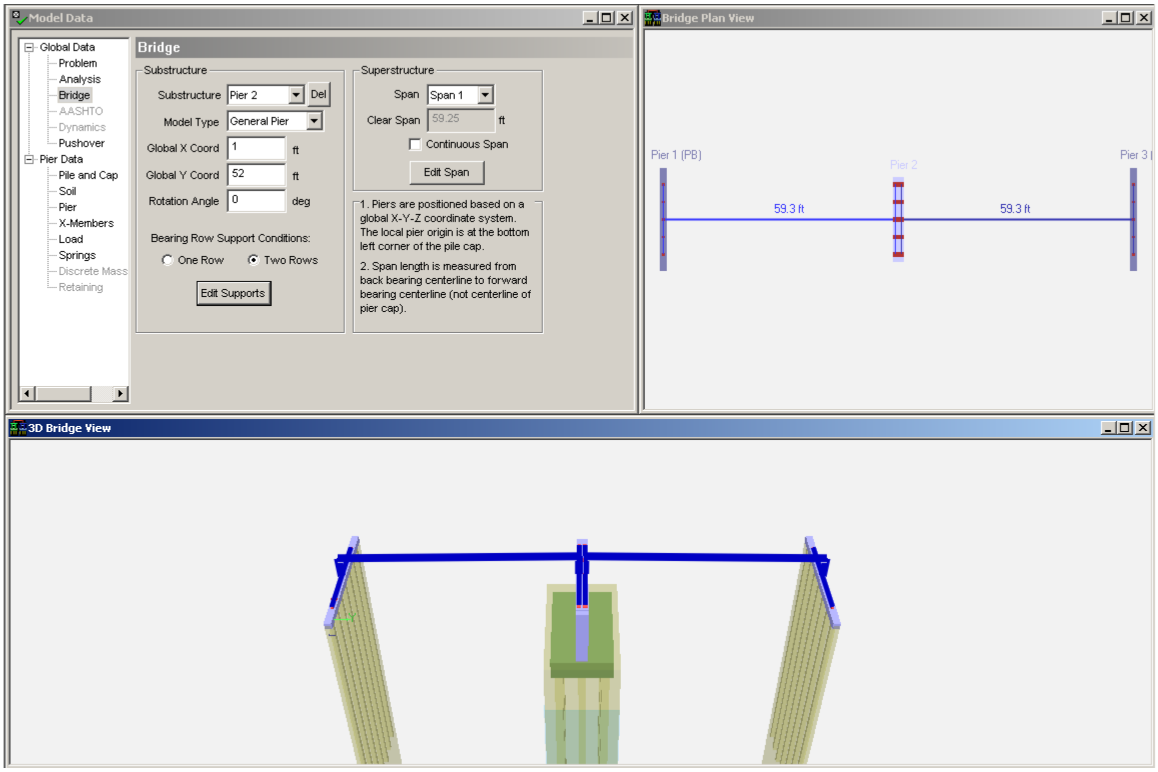


Figure 2.9 Aligned Bridge Model

This example will focus on a simple loading scenario in order to concentrate on the effect of the vehicle collision load on the interior pier support. The default Bridge model type automatically includes self weight of the abutments, pier, and bridge spans. The only load that needs to be added to the current model is the vehicle collision force. In this example, 500 kips will be applied at 6 feet above the rightmost pier column. To apply this load, first make sure that “Pier 2” is the selected pier and then select the *Load* data page. Click on Node 151 in the 3D View window. This node is the third node in the rightmost pier column. Click the “Add” button to the right of the Node Applied list and then enter “-500” kips for the applied load as shown in **Figure 2.10**. Note that the negative load value is used to apply the load in the negative x-direction.

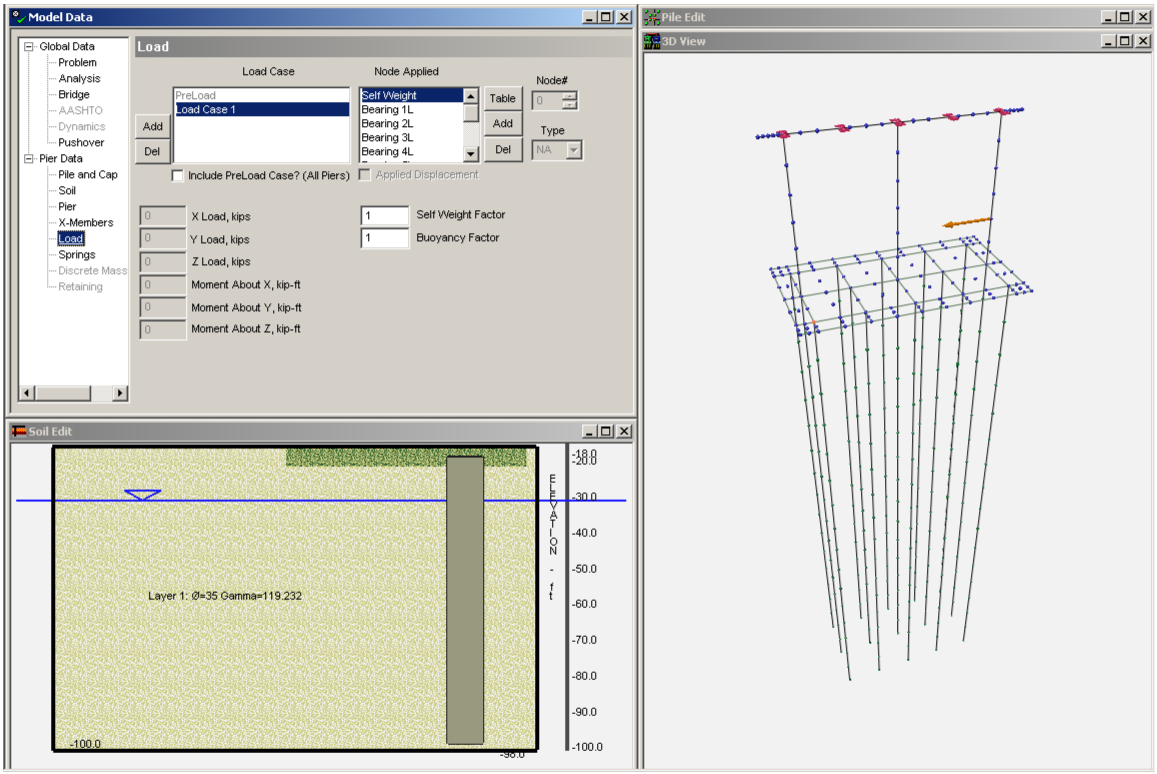
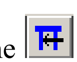


Figure 2.10 Applying Vehicle Collision Load

This completes the modeling phase of this bridge model. To analyze the model under the applied loads, click the  (Analyze) button in the toolbar. Results can be viewed after reaching a converged solution to the problem. Results are presented per pier and the pier selection combo box in the toolbar is used to switch between piers.

To view the force results for the interior pier support, click the  (Pier Results) button in the toolbar and then select “Pier #2” from the pier selection combo box. Select the third pier column to view the internal forces in the column as shown in **Figure 2.11**.

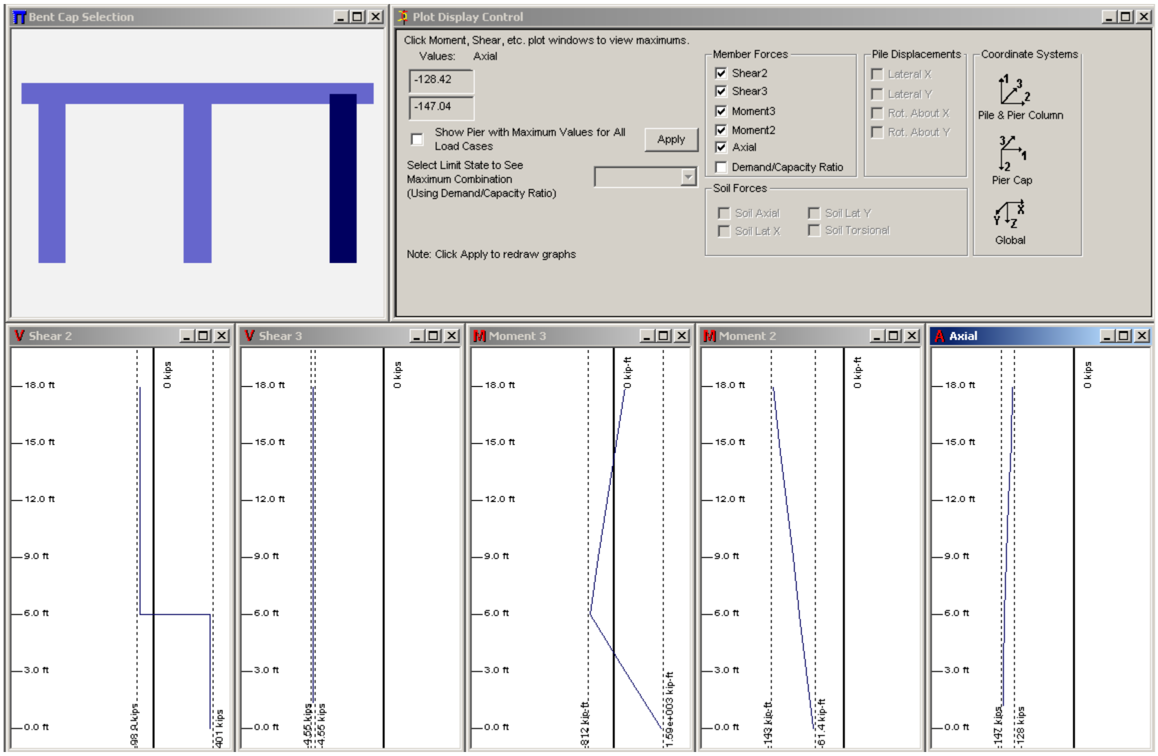
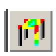


Figure 2.11 Plotting Internal Forces in the Pier Column

The deformed shape of the pier and bridge model can provide additional insight into the model behavior. Click the  (3D Results) button to view the deformed shape for the interior pier. The deformed pier is shown in **Figure 2.12**. Nodal displacement data can be view by clicking on any node in the pier model. The deformed shape for the entire bridge can be viewed by right-clicking the mouse in the 3D Results window and selecting “Bridge View” from the popup menu. This view is shown in **Figure 2.13.**

This concludes Example MP-2.

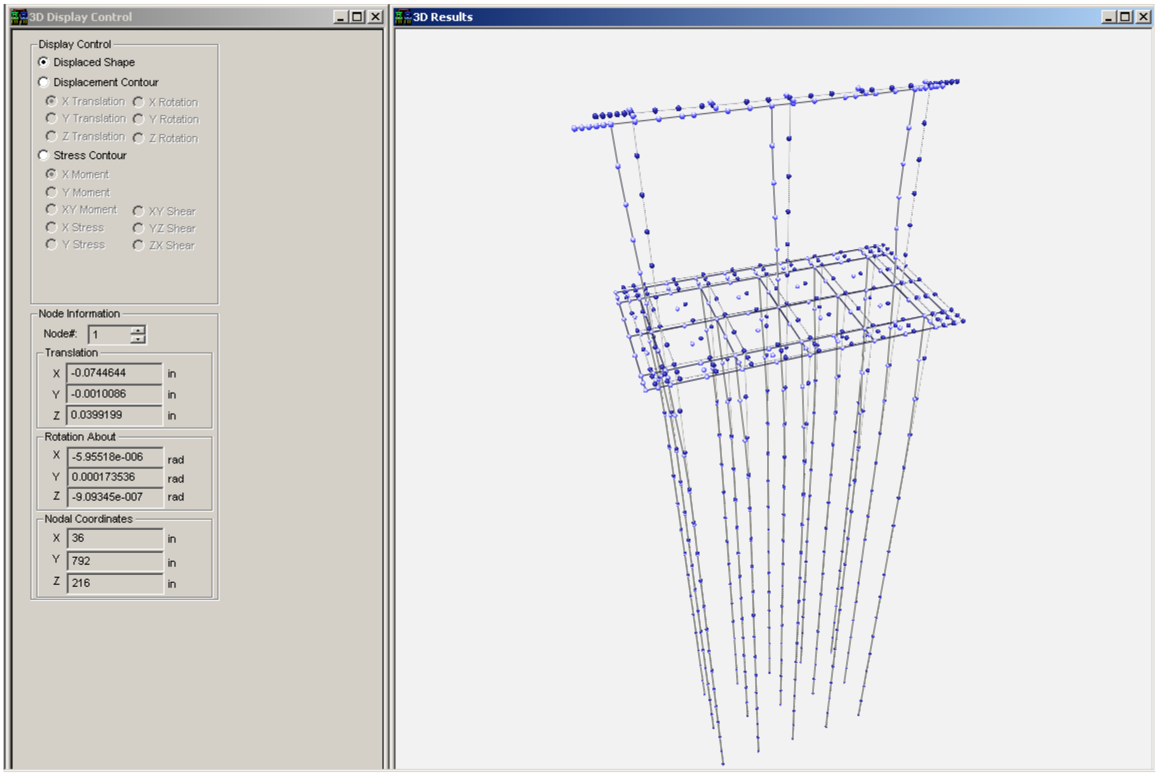


Figure 2.12 Pier Deformed Shape

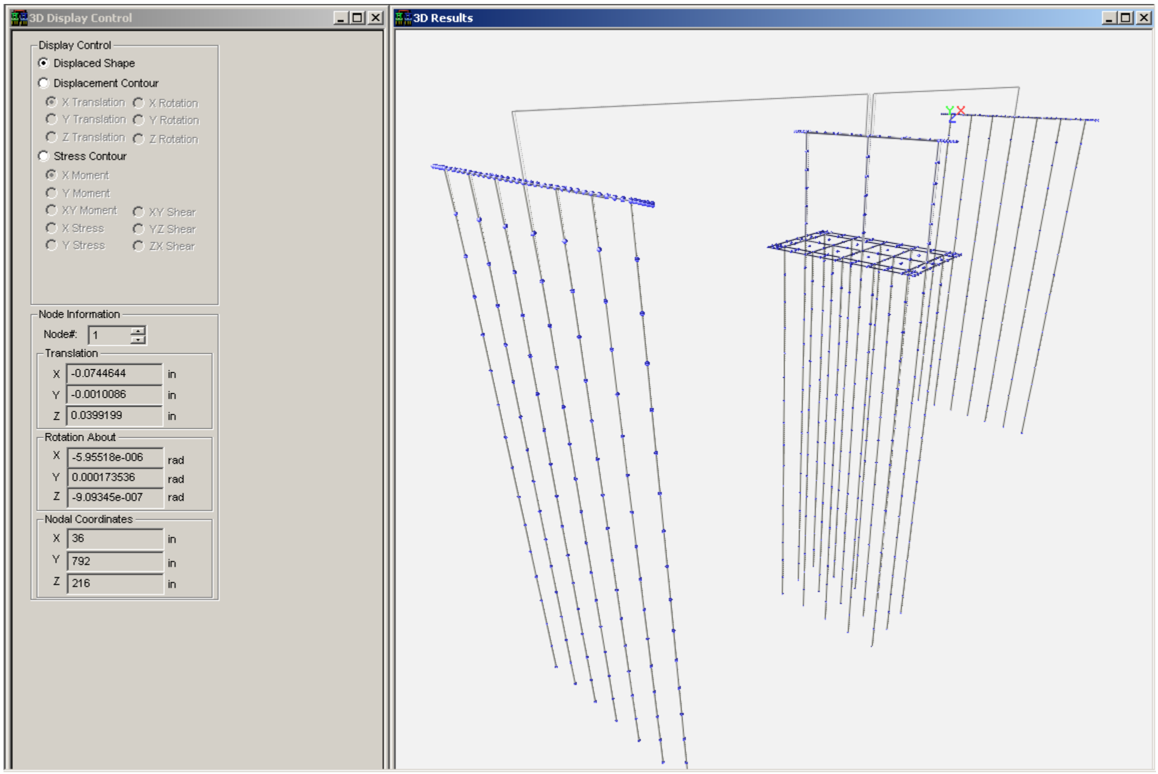


Figure 2.13 Bridge Deformed Shape

MP-3. HIGHWAY BRIDGE IMPACT ANALYSIS

Shown in **Figure 3.1** is a two-span highway overpass bridge, which was modeled in **Example MP-2**. The bridge is supported by pile bent abutments and a general pier interior support. The bridge is primarily subjected to vertical loads, but there is some concern about a possible vehicle collision force on one of the columns in the pier. The bridge model developed in Example MP-2 will be modified to study the effect of a time-dependent collision load on the pier column.

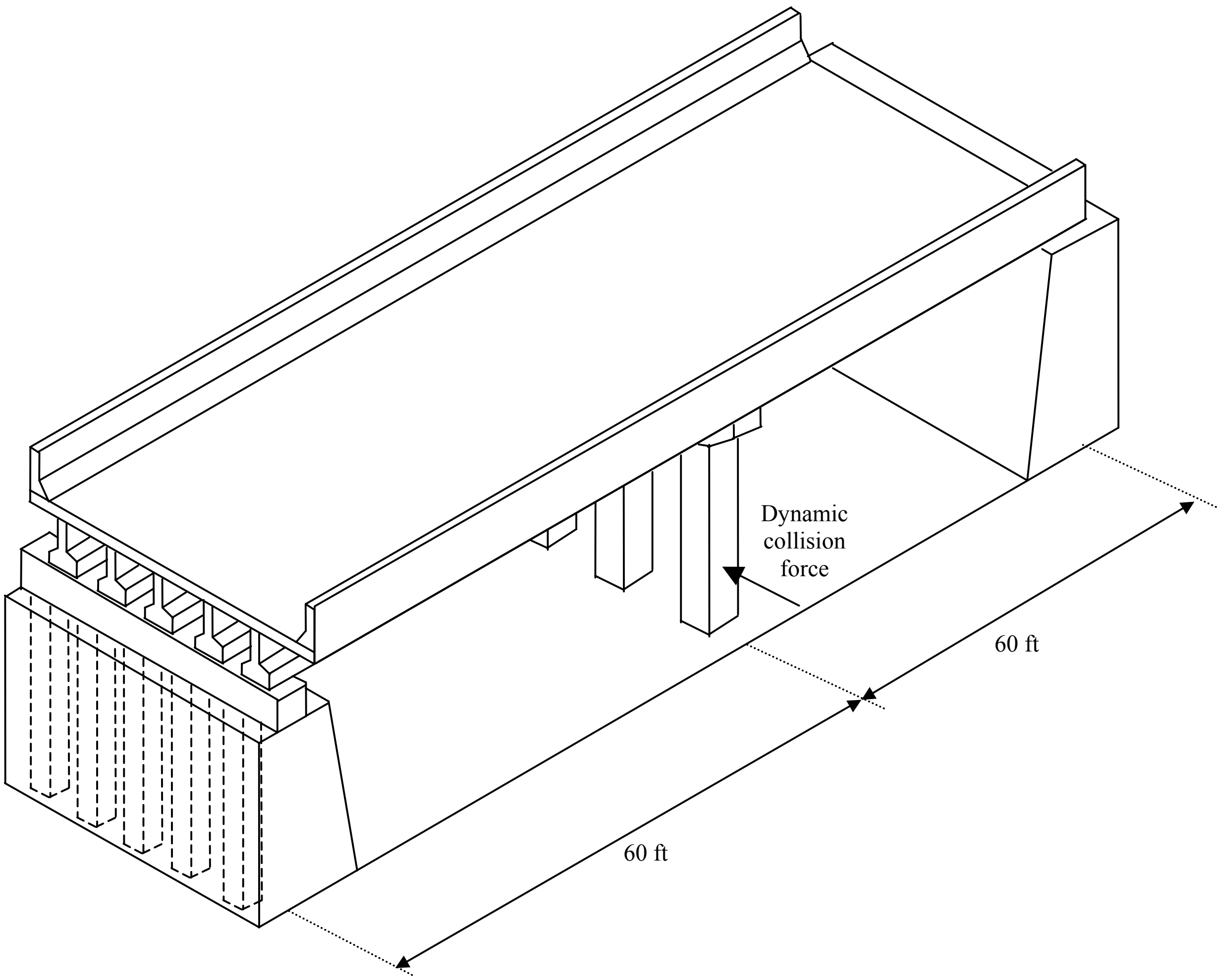


Figure 3.1 Example MP-2, Two-Span Highway Overpass Structure

Select Open from the File menu and choose Example MP-2.in from the program directory (**Figure 4.2**).

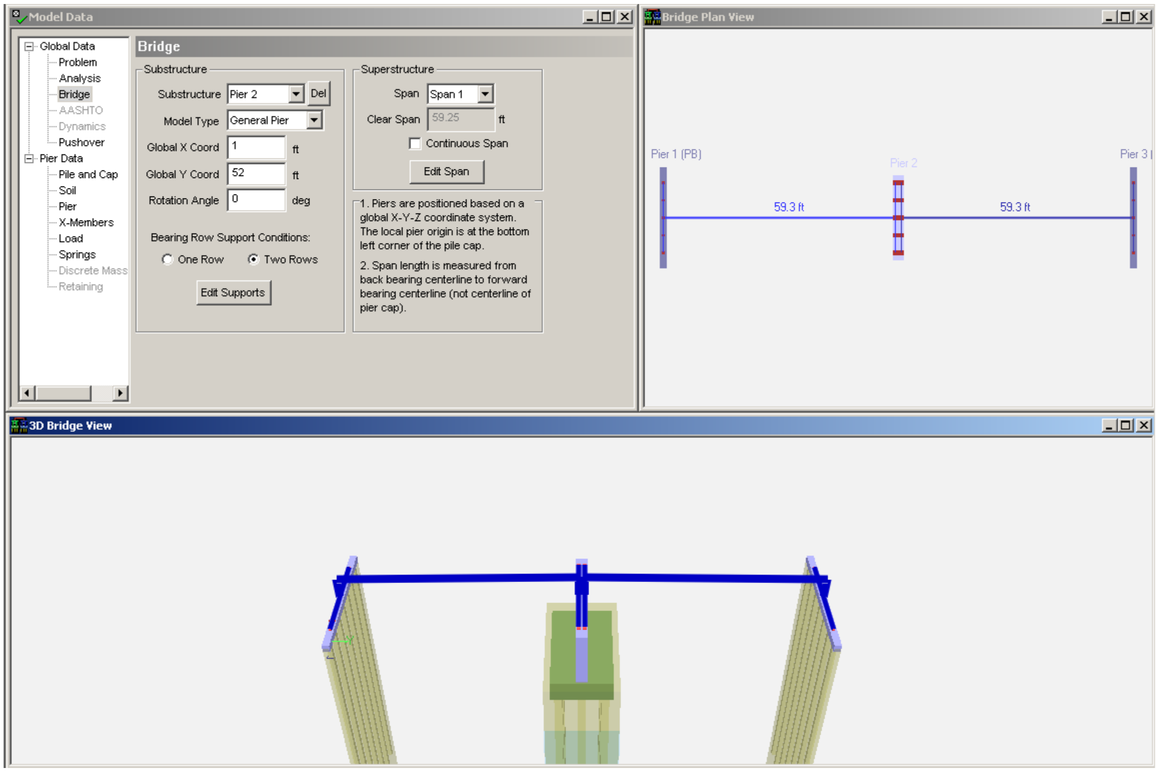


Figure 3.2 Example MP-2, Bridge Model

In addition to a static analysis, FB-MultiPier offers dynamic analysis capabilities. The current example considers only static loads. To switch to a dynamic analysis, select the *Analysis* data page and choose “Dynamic” for the Analysis Type. Click “Yes” to remove all static load cases beyond load case 1 (a dynamic analysis can only contain a single load case). Notice that after selecting a dynamic analysis, the “Dynamics” tree item becomes enabled. Select the *Dynamics* data page to specify the dynamic parameters to use in the modeling and analysis. The *Dynamics* data page is shown in **Figure 3.3**.

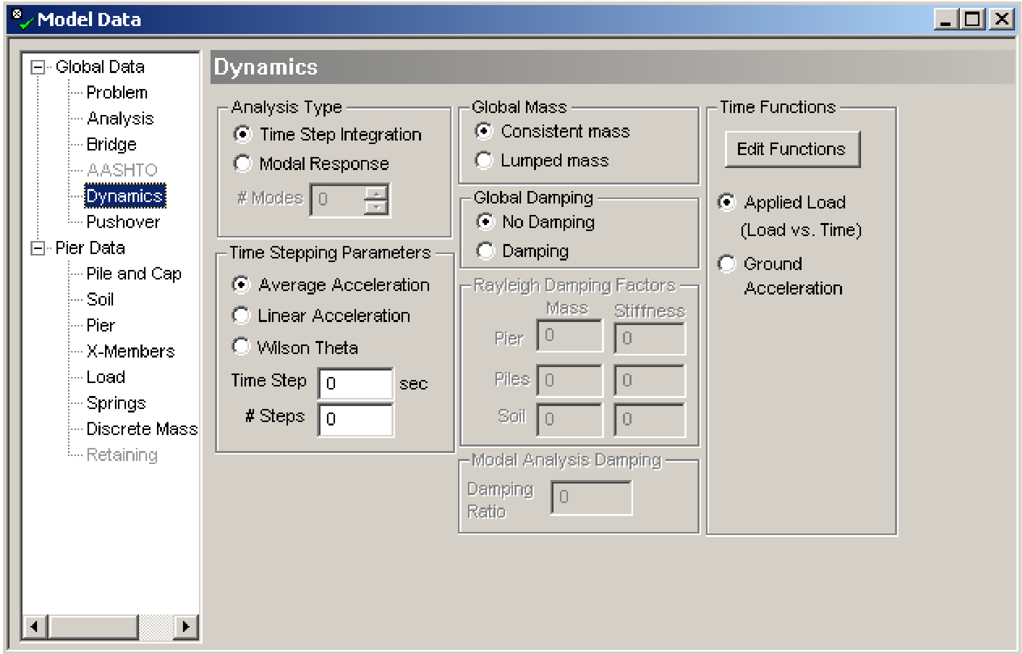


Figure 3.3 Dynamics Data Page

The *Dynamics* data page contains a number of options for controlling the dynamic analysis. Among these options are two types of dynamic analysis: 1) Time Step Integration and 2) Modal Response Spectrum Analysis. This example will demonstrate the program capabilities for a time step integration analysis. The *Dynamics* data page also contains options for including damping in the analysis using mass and stiffness proportional damping factors as well as an option for modeling consistent or lumped mass behavior. Three implicit time step iteration methods are also available and each requires parameters indicating the time step and number of time steps to consider in the analysis. For this example, enter "0.02" for the Time Step and "50" for the Number of Steps. The dynamic load function can be either time-dependent loads or a ground acceleration record.

Click on the "Edit Functions" button to define a time-dependent load function. In the Edit Load Functions dialog, select "Add Load Function" from the Load Function combo box. After doing so, "Load Function 1" will be added to the combo box. To define the load function values, click on the "Edit Function Values" button. Type "Vehicle Impact" for the name of the load function. Enter the 7 values shown in **Figure 3.4** and click “Ok”. The user-defined load function is shown in the plot in **Figure 3.5**. Click “Ok” to close the plot dialog.

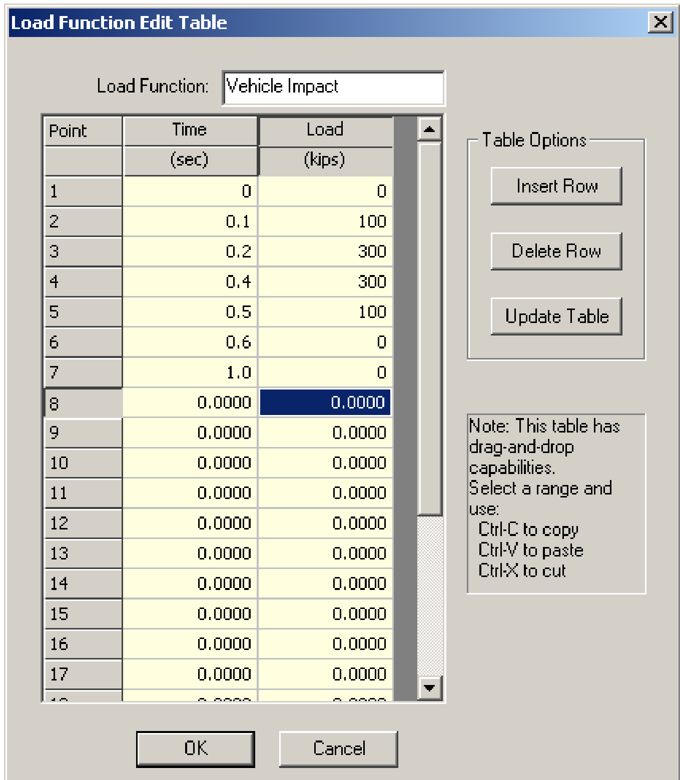


Figure 3.4 Load Function Values

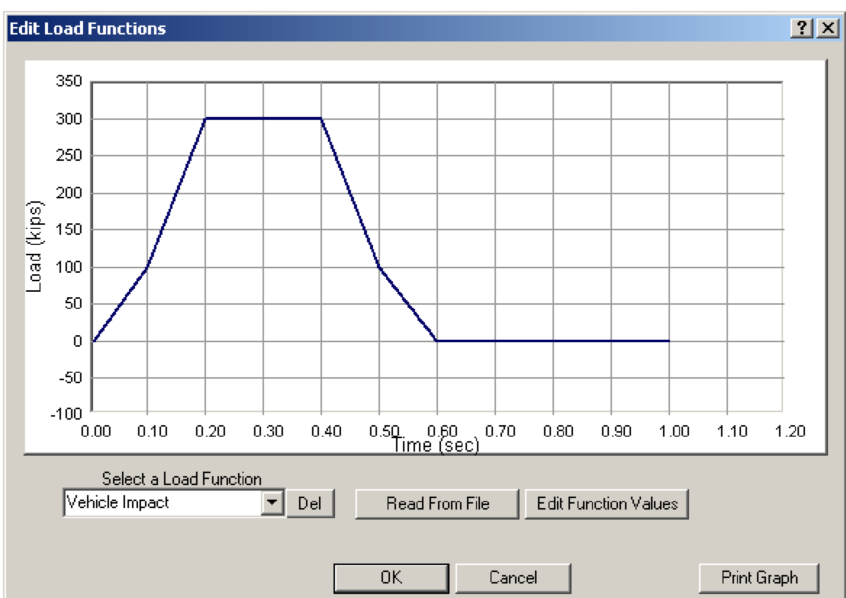


Figure 3.5 Plotted Load Function

Now that the load function is defined, select the *Load* data page to apply the load to the model. To apply the load to the interior pier, select "Pier #2" from the pier selection combo box in the toolbar. This pier currently has a static load applied to it. This load will be replaced with the dynamic load function. The letter "**S**" in front of Node 151 is used to indicate a static load. Click on the "**S**" to switch the load to a dynamic load. Notice that the letter changes to a "**D**" to indicate a dynamic load and the options on the page change as well. To apply the load in the transverse direction, enter "1" for the X Direction factor. Also select the "Vehicle Impact" load function from the load function combo box. The current state of the Load data page is shown in **Figure 3.6**. This completes the load definition and application process. Note that both static and dynamic loads can be applied to a model, but not at the same node. This example demonstrates dynamic load capabilities with the only static load being the self weight of the bridge.

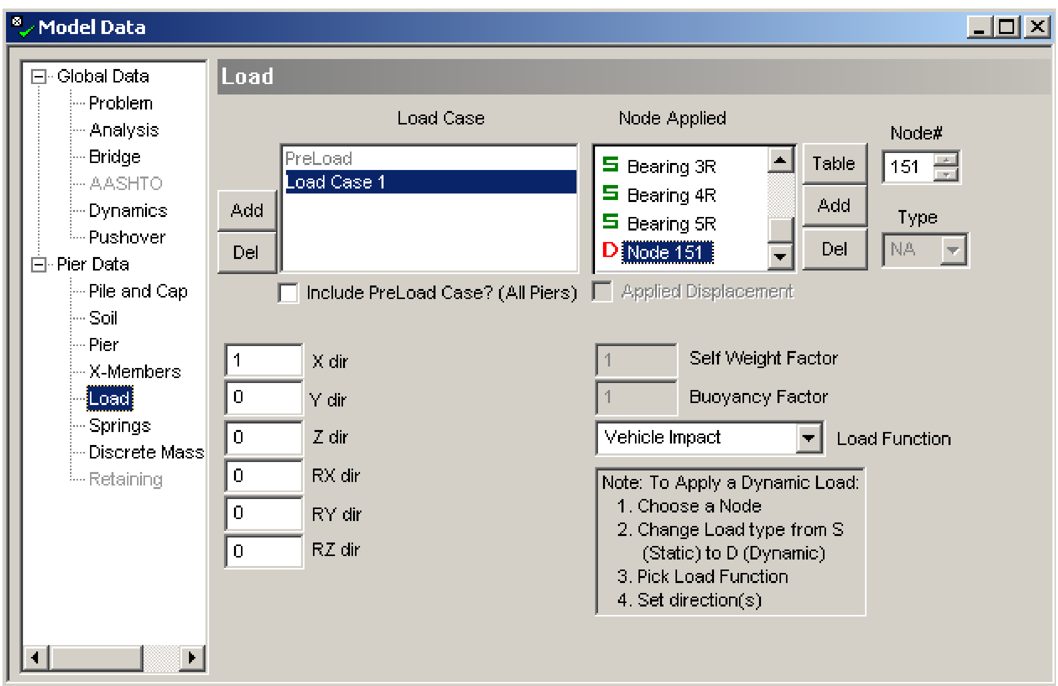
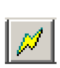


Figure 3.6 Load Data Page

This completes the modeling phase of this bridge model. To analyze the model under the applied loads, click the  (Analyze) button in the toolbar.

Results can be viewed after reaching a converged solution to the problem. Results are presented per time step per pier and the pier selection combo box in the toolbar is used to switch between piers. Click the (Pile Results) button to view the maximum results for the piles. The pile group for the interior pier support is of particular interest since it must resist the collision force. Select “Pier #2” from the pier selection combo box in the toolbar to view the results for the interior pier. FB-MultiPier allows the user to view the maximum pile member force results and the corresponding time step (in a dynamic analysis). To view the maximum pile shear force in the transverse direction, select “Shear2” from the Member Force combo box as shown in **Figure 3.7**. Notice that the corresponding time step in which the maximum Shear2 occurred is shown next to the pier selection combo box in the toolbar. In this example, the maximum occurred in Time Step 16 (16 x 0.02 sec = 0.328 seconds into the analysis).

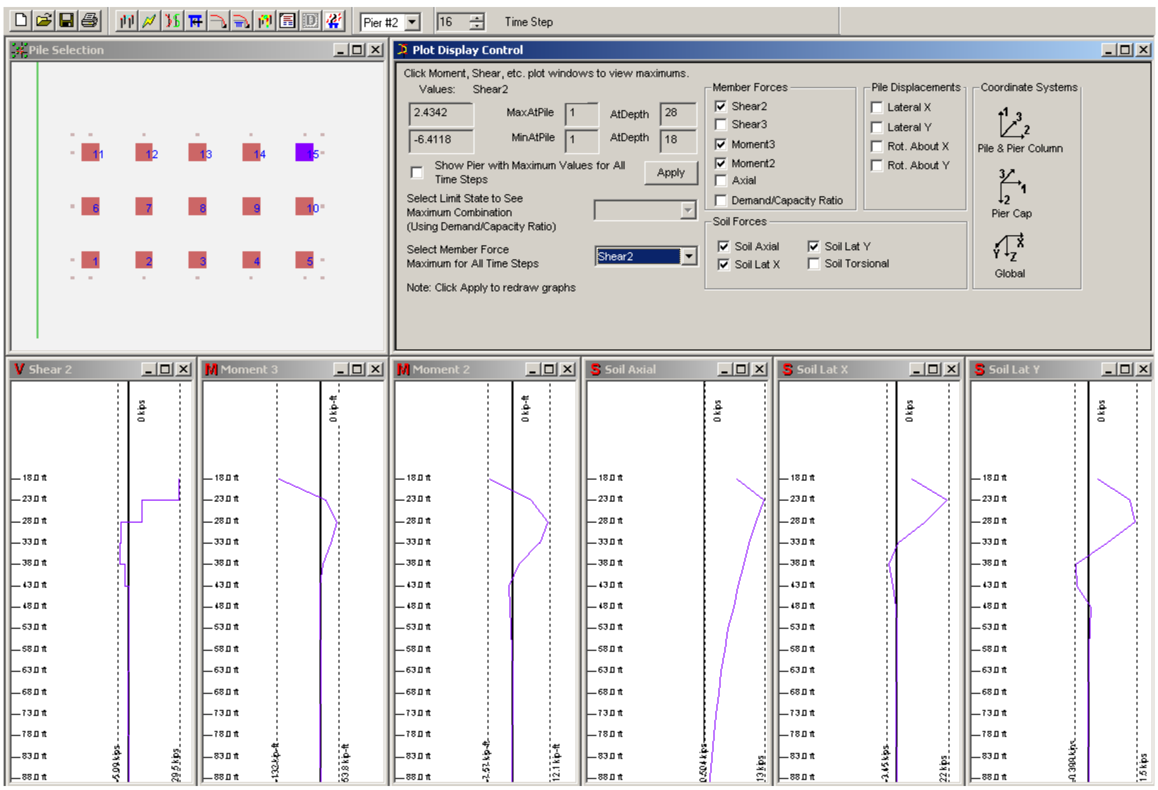


Figure 3.7 Maximum Pile Shear2 and Corresponding Time Step

To view the pier column results, click the (Pier Results) button. The outside pier column is of particular interest since it must resist the collision force. FB-MultiPier allows the user to view the maximum column and pier cap member force results and the corresponding time step (in a dynamic analysis). To view the maximum column shear force in the transverse direction, select “Shear2” from the Member Force combo box as shown in **Figure 3.8**. Notice that the corresponding time step in which the maximum Shear2 occurred is shown next to the pier selection combo box in the toolbar. In this example, the maximum occurred in Time Step 14 (14 x 0.02 sec = 0.28 seconds into the analysis).

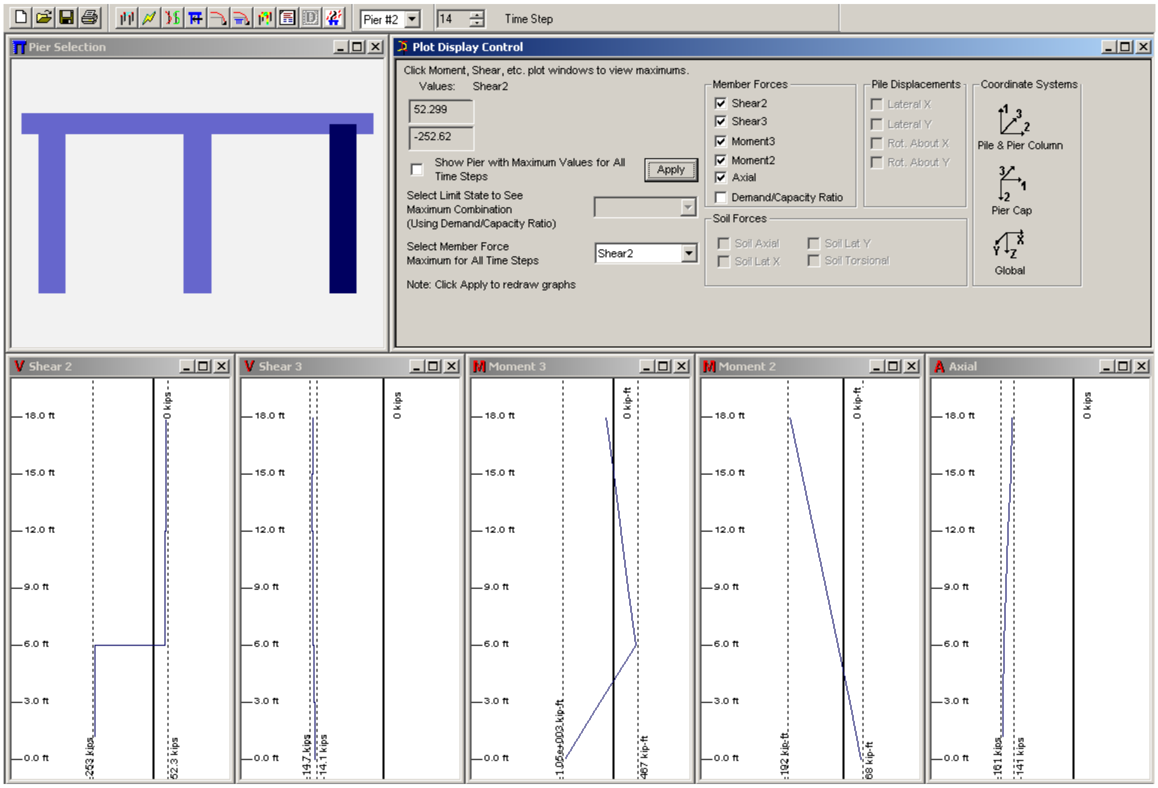


Figure 3.8 Maximum Pier Column Shear2 and Corresponding Time Step

The maximum moment in the pier columns is also of interest for this example. To view the maximum moment about the 3-axis, select “Moment3” from the Member Force combo box as shown in **Figure 3.9**. Notice that the maximum moment also occurred in the same column at time step 14.

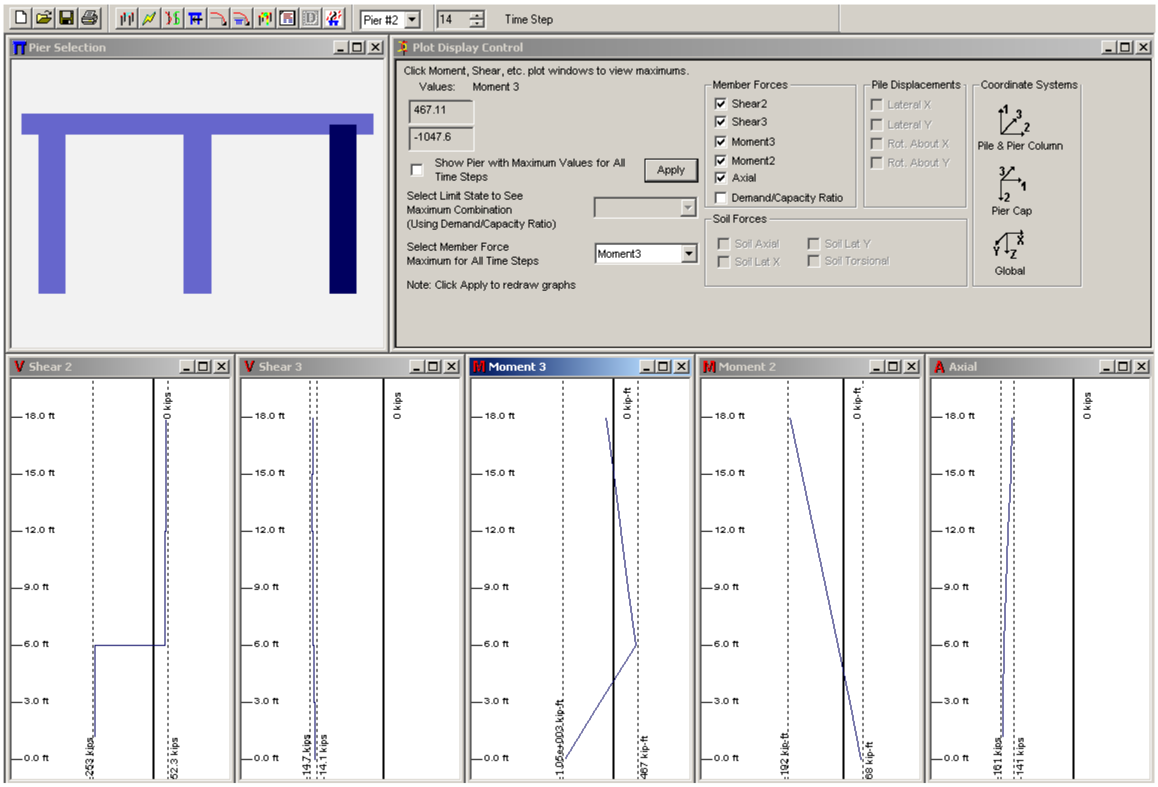
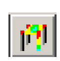


Figure 3.9 Maximum Pier Column Moment3 and Corresponding Time Step

To view the deformed shape of the pier, click the  (3D Results) button. Next, click on Node 151 in the 3D model to view the displacement results at the impact point. After doing so, select the X (transverse) translation option under “Results Plotting” and click the “Plot” button to view a displacement time history. The resulting plot is shown in **Figure 3.10**. Other displacement directions can be plotted as well. An animated movie of the time history response can be viewed by clicking the “Animate!” button. Click the button again to stop the animation.

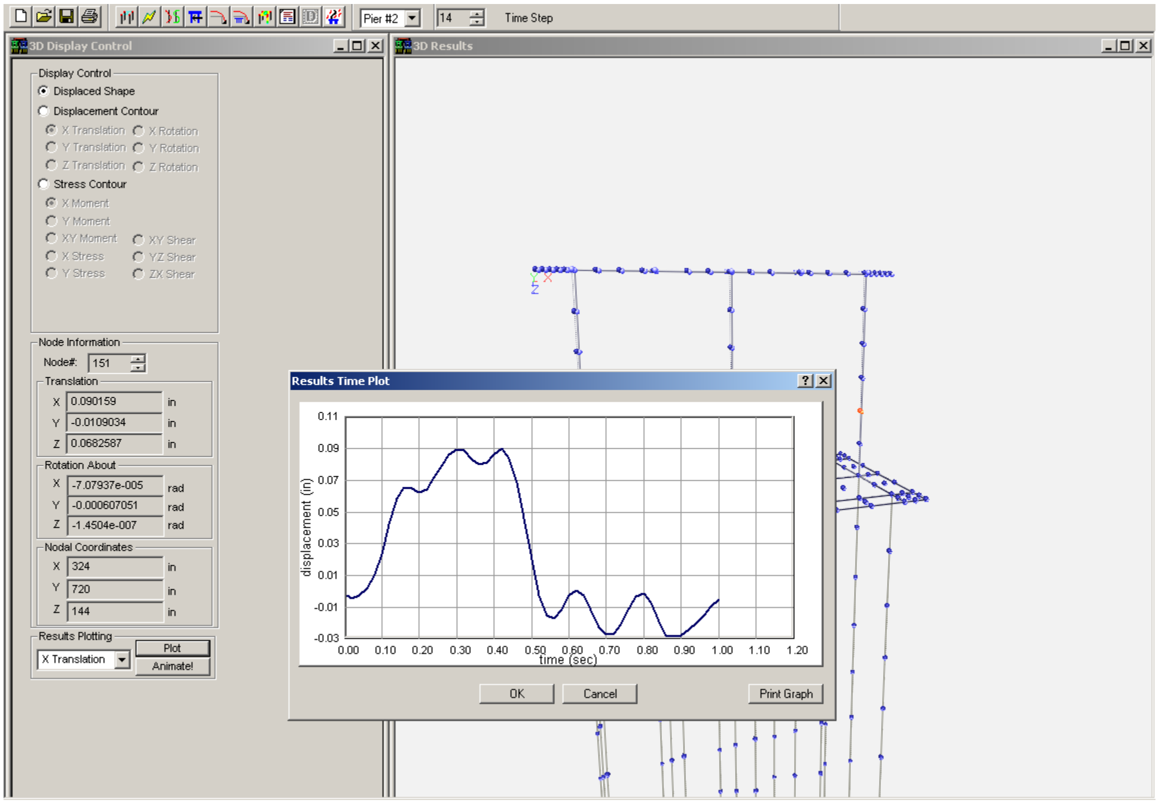


Figure 3.10 Displacement History at Impact Point

Before concluding this example it is important to note that the bearing reactions are also available (in the output file) at every time step during the analysis. By viewing the bearing reactions at each pier, the user can determine the amount of load that is shed to adjacent piers. **Figure 3.11** shows the bearing reactions at Time Step 14, where the shear force in the pier column reached a maximum value. Refer to the FB-MultiPier Help Manual for the sign convention for the bearing reactions. The bearing reactions are given an “L” or “R” designation, indicating the left or right row, respectively. The transverse bearing reactions are located in the column labeled “FX.” Summing the reactions for Pier #1 (the first abutment) indicates that only 19 kips of the dynamic impact (approximately is transferred to the first pier. Pier #3 also receives the same load from symmetry. In this example, the interior pier absorbs most of the impact.

BEARING REACTIONS

PIER #1

LOC	L/R	FX (Kips)	FY (Kips)	FZ (Kips)	MX (Kip-ft)	MY (Kip-ft)	MZ (Kip-ft)
1	R	9.1	24.1	-9.8	-97.6	120.7	-16.6
2	R	-0.4	19.0	-87.9	-67.4	34.0	4.9
3	R	9.5	48.2	-158.8	-144.0	227.5	179.4
4	R	0.0	0.0	0.0	0.0	0.0	0.0
5	R	0.0	0.0	0.0	0.0	0.0	0.0

PIER #2

LOC	L/R	FX (Kips)	FY (Kips)	FZ (Kips)	MX (Kip-ft)	MY (Kip-ft)	MZ (Kip-ft)
1	L	-28.1	10.0	-13.0	0.0	-166.0	10.5
2	L	-10.6	-23.3	-49.2	0.0	-1.8	6.9
3	L	-17.1	1.8	-64.1	0.0	-148.4	6.5
4	L	9.8	23.3	-63.3	0.0	3.2	-8.3
5	L	40.6	3.5	-42.8	0.0	84.7	-8.8
1	R	-5.6	2.3	-59.8	-130.6	-141.6	9.0
2	R	11.5	23.6	57.4	-80.0	2.2	9.5
3	R	-16.3	11.7	-106.6	-147.6	-157.2	-3.4
4	R	-8.9	-23.2	72.0	-101.5	-3.1	-5.7
5	R	12.7	11.0	-90.0	-156.7	58.1	-9.5

PIER #3

LOC	L/R	FX (Kips)	FY (Kips)	FZ (Kips)	MX (Kip-ft)	MY (Kip-ft)	MZ (Kip-ft)
1	L	9.9	-7.7	3.1	52.1	99.2	-0.6
2	L	-1.1	-10.2	-41.8	44.3	13.6	-3.6
3	L	0.4	-25.1	-50.9	66.8	78.4	-64.9
4	L	0.0	0.0	0.0	0.0	0.0	0.0
5	L	0.0	0.0	0.0	0.0	0.0	0.0

Figure 3.11 Bearing Reactions at Time Step 14

This concludes Example MP-3.

MP-4. HIGHWAY BRIDGE SEISMIC ANALYSIS

Shown in Figure 4.1 is a two-span typical highway overpass bridge, which was modeled in Example MP-2. The bridge is supported by pile bent abutments and a general pier interior support. The bridge is primarily subjected to vertical loads, but there is some concern about a possible seismic event. The bridge model developed in Example MP-2 will be modified to study the effect of a time-dependent, seismic loading (El Centro acceleration record).

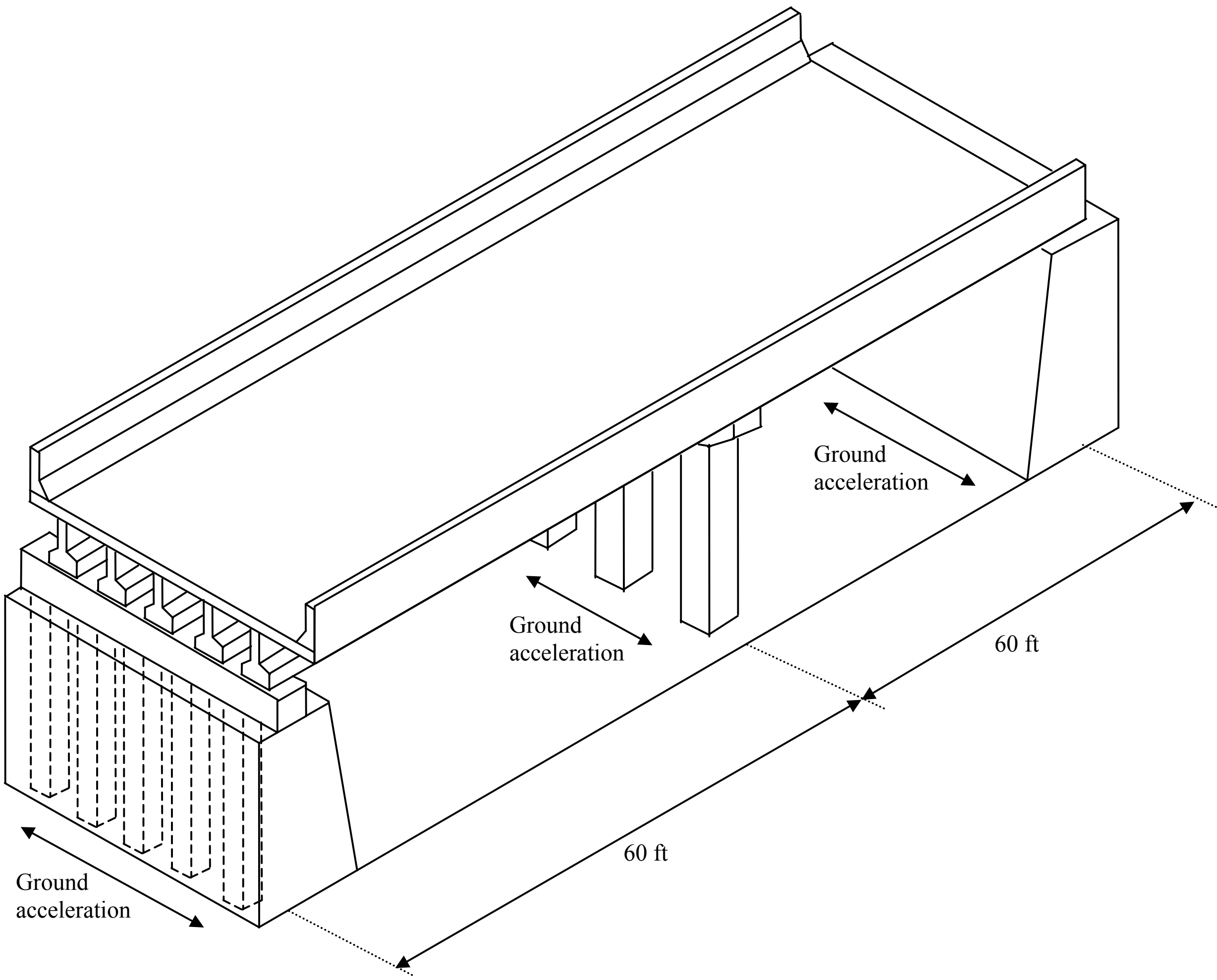


Figure 4.1 Example MP-2, Two-Span Highway Overpass Structure

Select Open from the File menu and choose Example MP-2.in from the program directory (**Figure 4.2**).

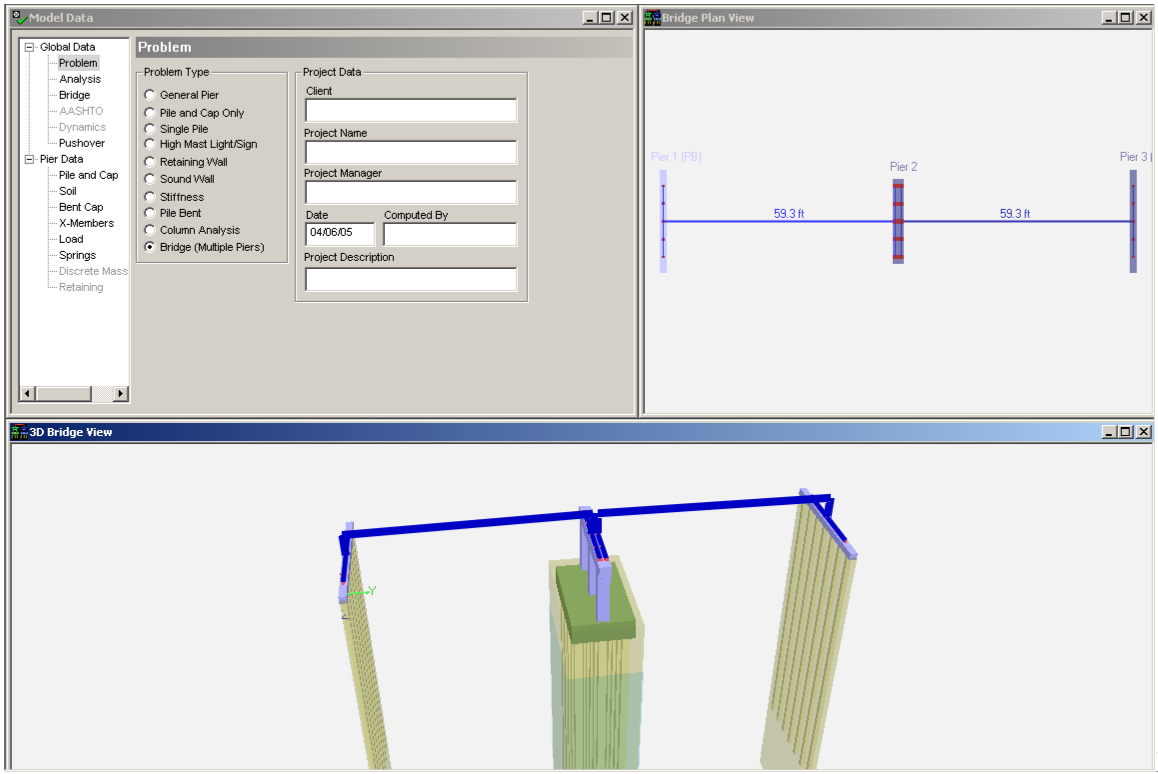


Figure 4.2 Example MP-2, Bridge Model

Example MP-2 utilized a concentrated load to simulate a vehicle collision event. This load is not applicable in this example and should be removed. To do so, select “Pier #2” from the pier selection combo box in the toolbar. Next, select the *Load* data page in the model data window and then click the “Del” button to the right of the Node Applied list to remove the concentrated nodal load at Node 151. The model is now ready for the seismic load application.

The current example considers only static loads. To switch to a dynamic analysis, select the *Analysis* data page and choose “Dynamic” for the Analysis Type. Click “Yes” to remove all static load cases beyond load case 1 (a dynamic analysis can only contain a single load case). In this example, the El Centro ground acceleration record will be used to excite the bridge model. This record (provided with the program) contains points at an equal time step of 0.02 seconds. In the *Dynamics* page, enter “0.02” seconds for the time step, “1000” for the number of steps, and select “Ground Accel.” and click “Yes” to proceed. Finally, enter "386.4" in/sec^2 for the gravity factor for the acceleration record. These dynamic parameters are shown in **Figure 4.3**.

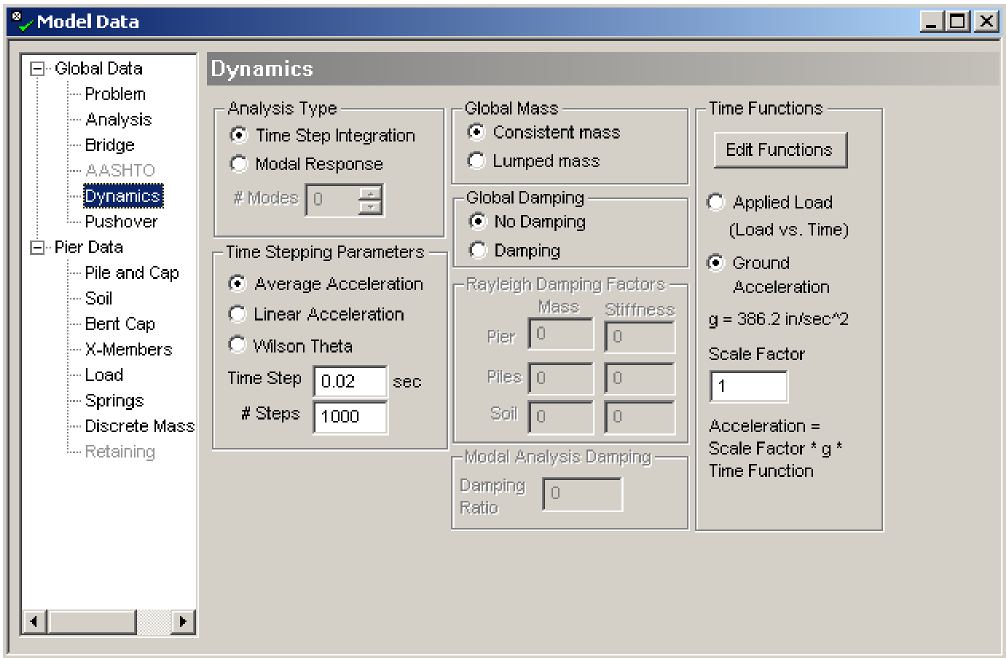


Figure 4.3 Dynamics Data Page With Ground Acceleration Parameters

The next step is to provide a ground acceleration record for the seismic analysis. To do so, click the "Edit Functions" button in the *Dynamics* data page. Next, click the "Read From File" button to load an existing acceleration record. Click "Yes" to add a new (blank) dynamic load function. Select "1940 El Centro.acc" from the program directory and click "Ok." The selected load function is graphed in the load function plot as shown in **Figure 4.4**. Note that load function values can be entered and modified using a grid table by clicking the "Edit Function Values" button.

The final step in modeling the ground acceleration involves the application of the dynamic load function. The currently defined ground acceleration must be applied in the *Load* data page for each pier. Begin with the first pier (abutment) by selecting "Pier #1" from the pier selection combo box in the toolbar. After selecting the *Load* data page, notice that a placeholder, labeled "Acc. (All Nodes)," is provided for the ground acceleration. This placeholder indicates that the ground acceleration will be applied to all nodes in the pier. Notice that the available load parameters change after selecting the placeholder. To apply the ground acceleration in the transverse direction, enter “1” for the X Dir factor. Leave all other direction factors as “0.” Finally, make sure the “1940 El Centro” record is selected to apply to the pier. When done, confirm that the same acceleration record and direction have been applied to the remaining piers by selecting “Pier #2” and “Pier #3” from the pier selection combo box.

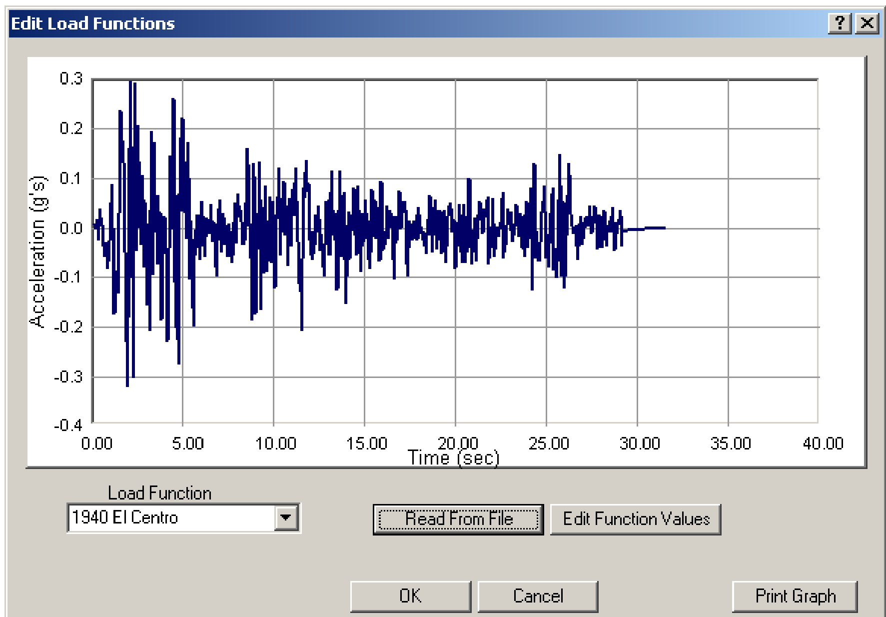
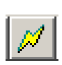
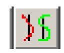


Figure 4.4 El Centro Acceleration Record

Before analyzing the bridge model it is important to note that only 1000 time steps are being considered even though the El Centro record contains 1576 points. This decision was made solely to reduce the analysis time for this example. A complete analysis should consider the bridge behavior during the entire seismic event.

This completes the modeling phase of this bridge model. To analyze the model under the applied loads, click the  (Analyze) button in the toolbar. The analysis will take several minutes depending on the speed of the computer. Results can be viewed after reaching a converged solution to the problem. Results are presented per time step per pier and the pier selection combo box in the toolbar is used to switch between piers.

The amount of numerical results data is significant for a dynamic analysis. Keep in mind that processing the data requires a significant amount of computer resources and the printed output file is significantly larger than an output file from a static analysis. In fact, it is not uncommon to have more than 100MB of results data for a dynamic analysis of a full bridge. Results data summaries are available in order to assist the user in identifying maximum load conditions. For example, a summary of the maximum internal forces in the piers (and the corresponding time step) is given at the end of the printed output file. Additionally, the maximum internal force results and corresponding time step number can be viewed using the graphical interface.

To view the maximum results for the piles, click the  (Pile Results) button. To demonstrate the plotting capabilities, select “Pier #2” from the pier selection combo box in the toolbar to view the results for the interior pier. FB-MultiPier allows the user to view the maximum pile member force results and the corresponding time step. To view the maximum pile shear force in the transverse direction, select “Shear2” from the Member Force combo box as shown in **Figure 4.5**. Notice that the corresponding time step in which the maximum Shear2 occurred is shown next to the pier selection combo box in the toolbar. In this example, the maximum occurred in Pile #5 at Time Step 592 (592 x 0.02 sec = 11.84 seconds into the analysis).

A displacement history for any node in the model can also be plotted, as shown in **Figure 4.6**. Finally the displaced shape can be animated, using the “Animate!” button, to view the deformation behavior over time. An animation control dialog is also available to play, pause, and quickly advance to a particular time step during the animation. The “Animation Control” option is accessible from the toolbar.

This concludes the time history analysis portion of this example.

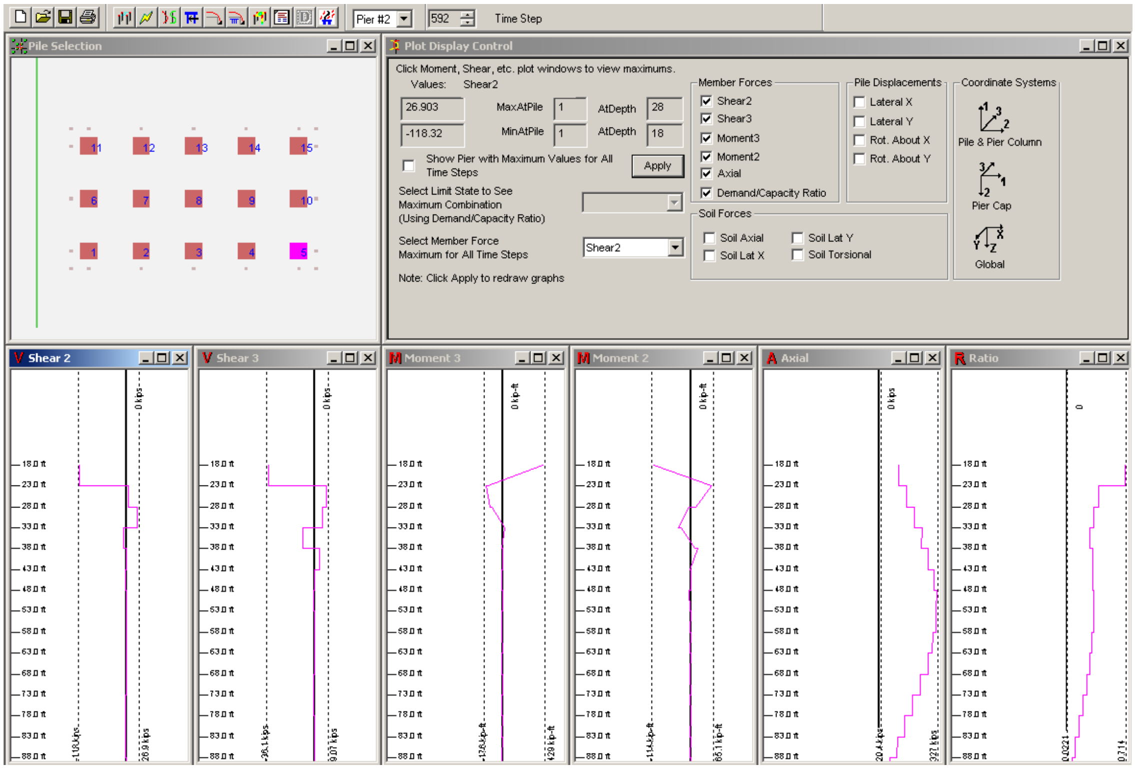


Figure 4.5 Maximum Pile Shear Force (Pile 5)

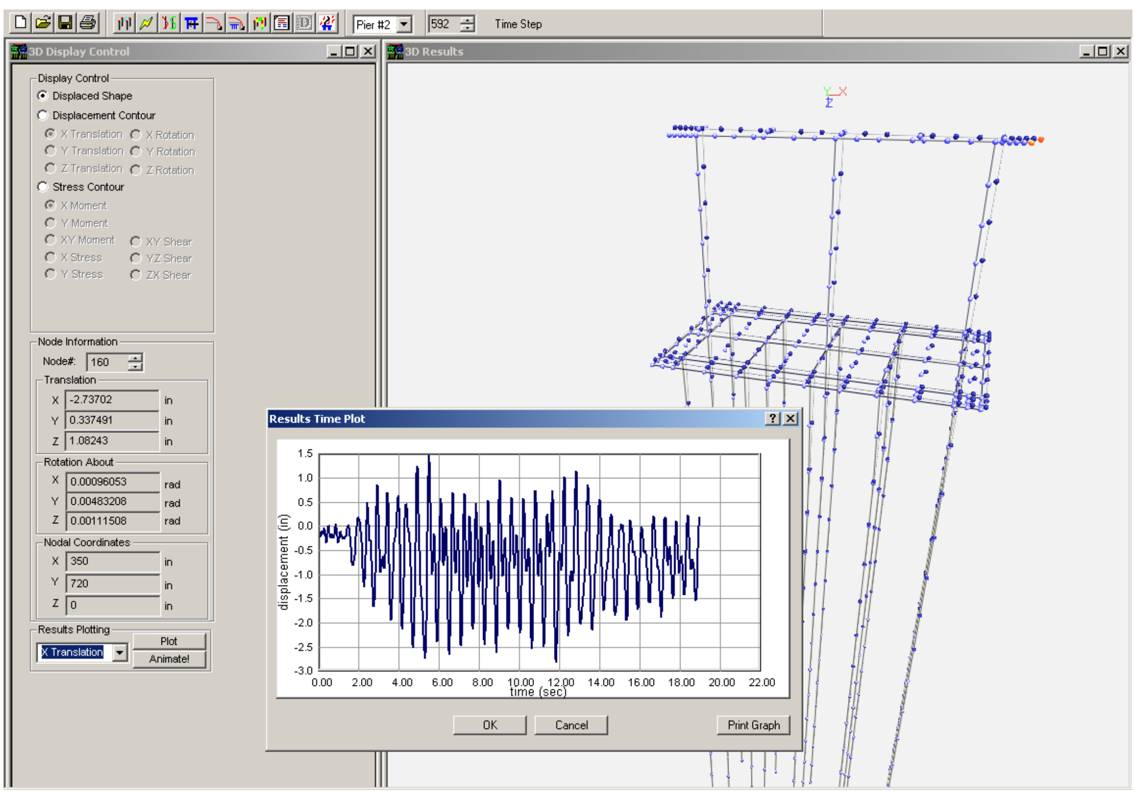


Figure 4.6 Displacement Response Record (Node 160 – Pier Cap End)

RESPONSE SPECTRUM ANALYSIS COMPARISON

FB-MultiPier provides a modal response spectrum analysis capability as an alternative to a time step integration analysis. One key benefit of the response spectrum analysis is the rapid analysis time due to the reliance on pre-computed maximum response quantities. A response spectrum analysis does have several drawbacks, however. Most notably, the user must include enough modes in the analysis to capture at least 90 percent of the structural response. The number of modes is generally obtained by trial and error and can require several analysis attempts. Also, the modal analysis requires a linear material behavior in order to correctly combine the modal contribution results. Despite these drawbacks, a comparison of both dynamic analysis methods is presented here to provide some insight into the differences between the methods.

The implementation of a response spectrum analysis in the FB-MultiPier program is somewhat unique due to the fact that soils exhibit a nonlinear behavior under most types of loading. By definition, modal analysis requires the linear combination of modal response quantities. FB-MultiPier, therefore, provides an approximate response spectrum analysis by combining the modal response of a structure with nonlinear properties (soil and possibly nonlinear pier and pile behavior). The modal analysis begins by applying all static loads to the model, including self weight. The program then performs an eigen analysis to generate the requested number of mode shapes and vibration frequencies while the model is *in the equilibrium (deformed) position*. The response of each mode is then combined (CQC) using the direction of excitation and the values for the response spectrum. Modal contribution factors are provided to indicate the contribution of each mode. Final combined results are available for pier internal forces and displacements. **Note that the reported results only reflect the external loads applied to the structure while in the equilibrium position**. That is, the internal forces from static loads, such as self weight, are not included in the results (because these loads were used to obtain the equilibrium position). Also, the results are based off the initial soil stiffness. The soil stiffness is not adjusted using an iteration process. Despite these factors, it is still possible to obtain results that are similar to those from a time step analysis.

The current example will now be analyzed using a response spectrum analysis. To begin, select the *Dynamics* data page to change the dynamic analysis type. Select "Modal Response" and click "Yes" to remove any previous load functions. Notice that most of the dynamic options are disabled for a response spectrum analysis. Next, enter "10" for the number of Modes. Since the number of modes needed to full represent the dynamic response is not known in advance, "10" is a good initial guess. The Dynamics data page is shown in **Figure 4.6**.

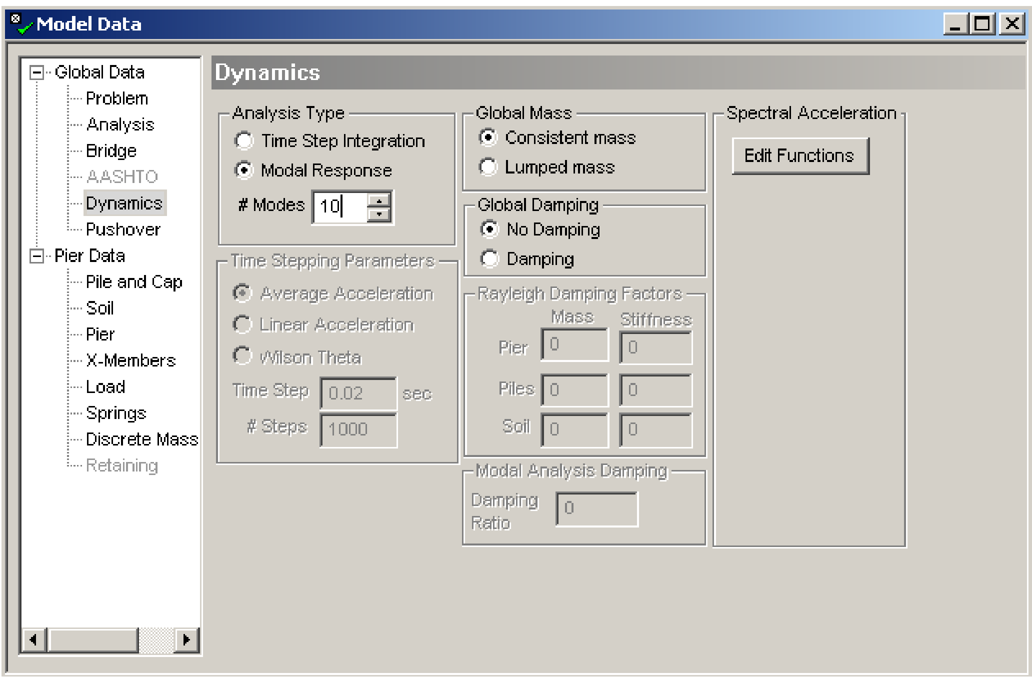


Figure 4.6 Dynamics Data Page – Modal Response Spectrum

Click the "Edit Functions" button to specify a response spectrum. In the Edit Load Functions dialog, click the "Read From File" button to load an existing spectrum for the 1940 El Centro earthquake. Click "Yes" to add a new load function. In the Open File dialog, select "elcen_00d.spt" from the list of response spectrums. The Edit Load Functions dialog with the plotted response spectrum is shown in **Figure 4.7**. Note that response spectrum values can be entered and modified using a grid table by clicking the "Edit Function Values" button. Also note that the response spectrum acceleration values represent the total acceleration, not the relative acceleration between the ground and structure. Acceleration values should be provided in length/time^2 units and frequency values should be provided in rad/sec. Click "Ok" to save the spectrum values.

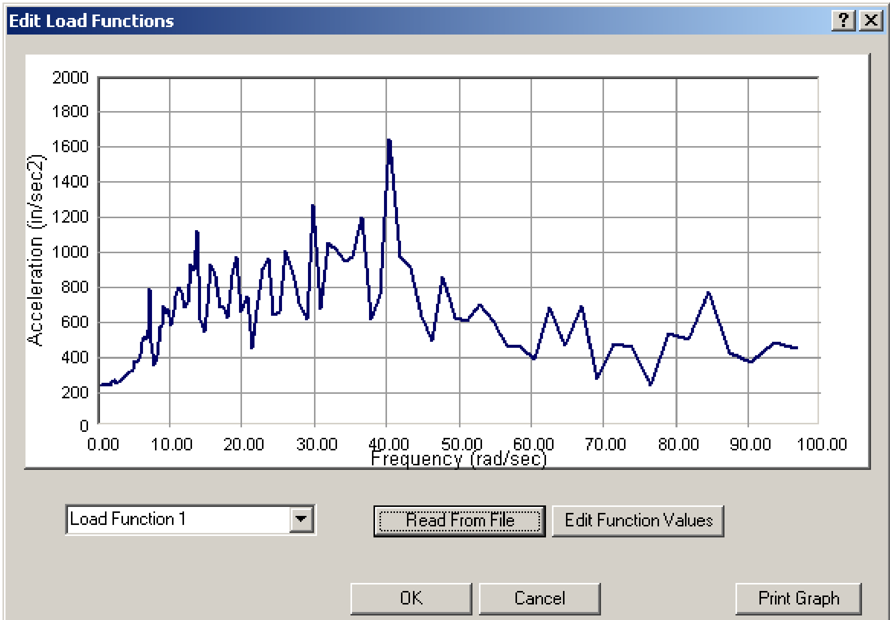


Figure 4.7 El Centro Response Spectrum

The final step in modeling the ground excitation involves the application of the response spectrum to the model. The currently defined spectrum must be applied in the *Load* data page for each pier. Begin with the first pier (abutment) by selecting "Pier #1" from the pier selection combo box in the toolbar. After selecting the *Load* data page, notice that a placeholder, labeled "Acc. (All Nodes)," is provided for the ground acceleration. This placeholder indicates that the ground acceleration will be applied to all nodes in the pier. Notice that the available load parameters change after selecting the placeholder. To apply the ground acceleration in the transverse direction, enter "1" for the X Dir factor. Leave all other direction factors as "0." Finally, select the "Load Function 1" record to apply to the pier. The *Load* data page is shown in **Figure 4.8**. When done applying the load to the first abutment, confirm that the same acceleration record and direction have been applied to the remaining piers by selecting "Pier #2" and "Pier #3" from the pier selection combo box.

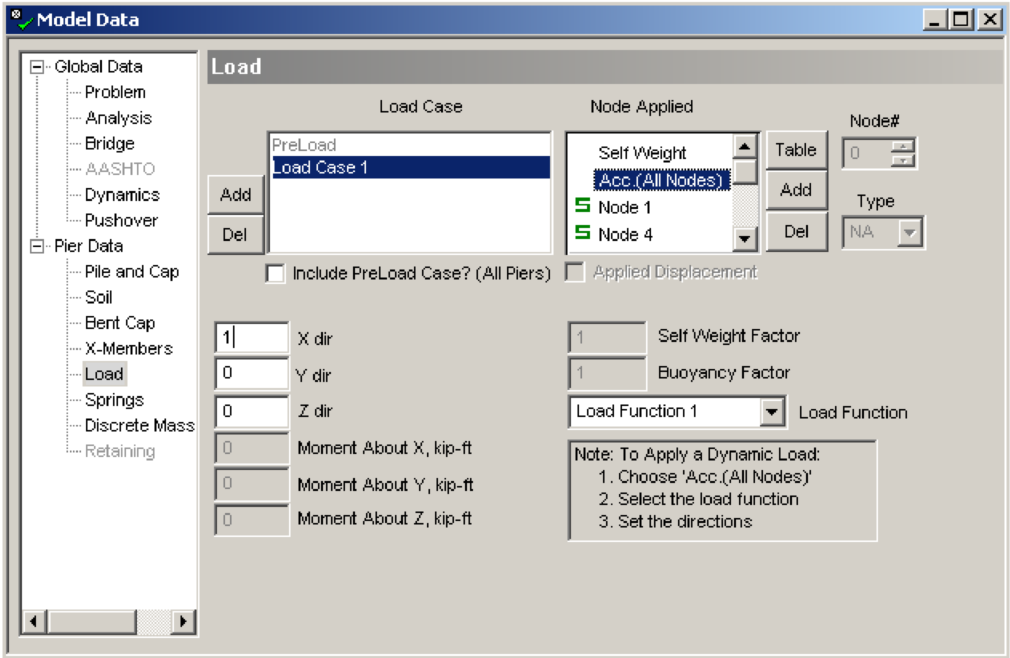


Figure 4.8 Applying the Response Spectrum

This completes the modeling phase for a response spectrum analysis. To analyze the model, click the  (Analyze) button in the toolbar. The analysis will take some time depending on the speed of the computer. Results can be viewed after reaching a converged solution to the problem. Results are presented per pier and the pier selection combo box in the toolbar is used to switch between piers. Both mode shape and combined displacement results can be viewed in the 3D Results mode for a single pier or the entire bridge. **Figure 4.9** shows the third mode response (5.64 Hz) for the entire bridge. This view was obtained first selecting “Mode Shape” in the Display Control for Pier 2 and then right-clicking the mouse in the 3D Results window and selecting “Bridge View.” **Figure 4.10** shows the combined displacement result for the interior pier support. This view was obtained by selecting “Displaced Shape” in the Display Control and then unselecting “Bridge View” and selecting “Pier #2” from the pier selection combo box in the toolbar. **Figure 4.11** shows the combined displacement result for the entire bridge.

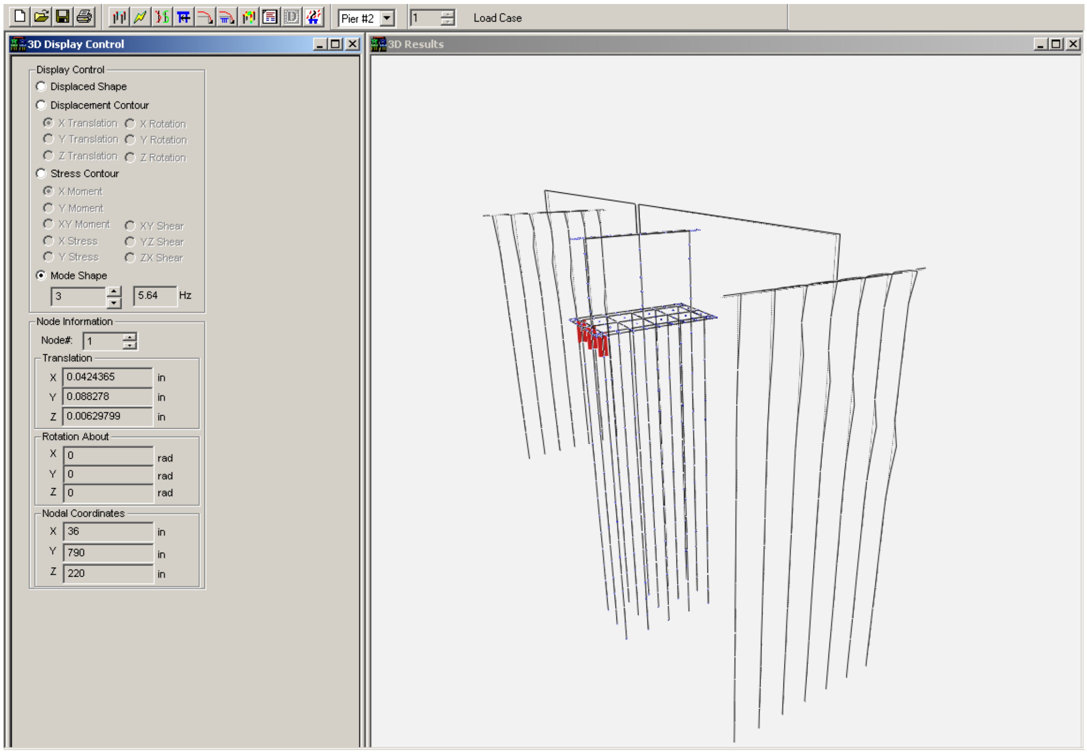


Figure 4.9 Third Mode Response – Entire Bridge

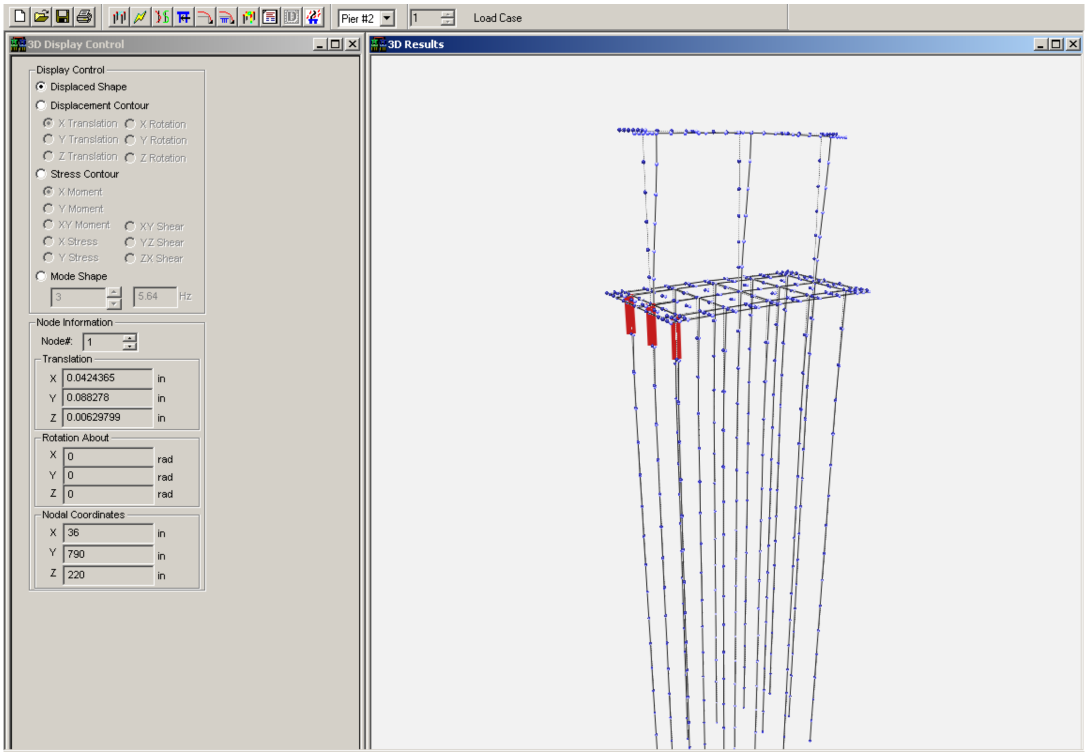


Figure 4.10 Combined Displacement Result – Interior Pier

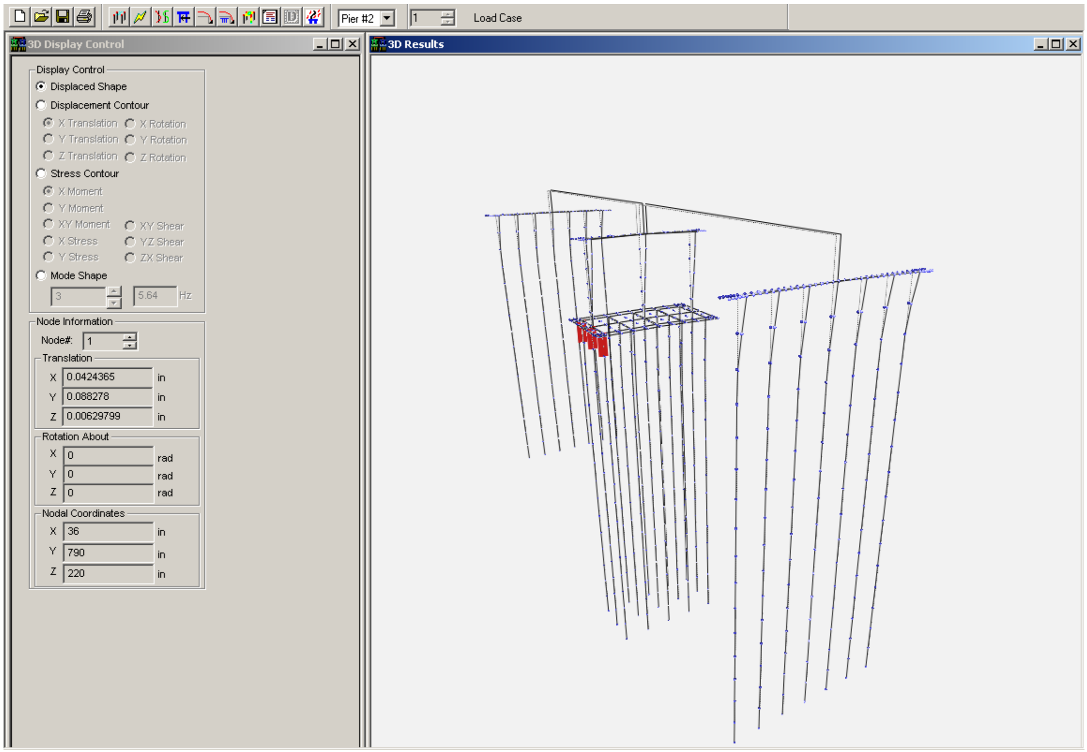


Figure 4.11 Combined Displacement Result – Entire Bridge

The modal contribution factors are printed in the output file to provide an indication of the modal participation in the response. The contribution factors for the first 10 modes are shown in **Figure 4.12**. Notice that the total modal contribution to the response is only 59.62%. This means that 40% of the dynamic response is unaccounted for. This level of contribution is well below the 90% contribution required in most design codes, so the results should not be used for design. The current contribution can only be increased by considering more modes in the analysis. For this example, at least 150 modes need to be considered in order to achieve at least 75% modal contribution and an undetermined amount of modes would be required to achieve at least a 90% contribution. This finding suggests that the modal response spectrum analysis may not always be a suitable replacement for a time step integration analysis. It is possible to represent the bridge response using a small number of modes if the pier foundation is simplified by replacing the soil with fixed supports (rigid springs) at the ground surface. Many more modes are typically needed when modeling soil-structure interaction.

Re-analyzing the model with 150 modes produces the contribution factors shown in **Figure 4.13**. Clearly, the choice of modes to use in the analysis is critical and should be evaluated before proceeding to the design of the bridge. As demonstrated in the example, for a full bridge analysis, it is possible to have many vibration modes that do not contribute to the dynamic response for the given direction of excitation.

```
- Percent Contribution Factors for X Direction
   ____________________________________________

 MODE #       1           2           3           4           5
 FACTOR      0.00       42.93        0.00        0.00        0.00

 MODE #       6           7           8           9          10
 FACTOR      0.42        0.00        0.01       16.25        0.00

 - Total Modal Contribution for X Direction =  59.62%
```

Figure 4.12 Modal Contribution Factors (10 Modes)

```
- Percent Contribution Factors for X Direction
   ____________________________________________

 MODE #       1           2           3           4           5
 FACTOR      0.00       42.93        0.00        0.00        0.00

 MODE #       6           7           8           9          10
 FACTOR      0.42        0.00        0.00       16.25        0.00

 MODE #      11          12          13          14          15
 FACTOR      0.00        0.39        0.00        0.00        0.00

 MODE #      16          17          18          19          20
 FACTOR      0.16        0.00        0.00        0.00        0.00

 MODE #      21          22          23          24          25
 FACTOR      0.00        0.00        0.08        0.00        0.00

 MODE #      26          27          28          29          30
 FACTOR      0.02        0.00        0.02        0.00        0.00

 MODE #      31          32          33          34          35
 FACTOR      0.00        0.13        0.01        0.01        0.09

 MODE #      36          37          38          39          40
 FACTOR      1.48        0.15        0.56        0.80        0.05

 MODE #      41          42          43          44          45
 FACTOR      0.49        0.01        0.11        0.01        0.01

 MODE #      46          47          48          49          50
 FACTOR      0.00        0.00        0.00        0.00        0.00

 MODE #      51          52          53          54          55
 FACTOR      0.00        0.00        0.00        0.00        0.00

 MODE #      56          57          58          59          60
 FACTOR      0.00        0.00        0.00        0.01        0.01

 MODE #      61          62          63          64          65
 FACTOR      0.00        0.00        0.06        0.01        1.93
```

```
MODE #      61          62          63          64          65
FACTOR      0.00        0.00        0.06        0.01        1.93

MODE #      66          67          68          69          70
FACTOR      0.04        0.00        0.00        0.00        0.00

MODE #      71          72          73          74          75
FACTOR      0.00        0.00        0.03        0.00        0.00

MODE #      76          77          78          79          80
FACTOR      0.23        0.00        0.03        0.00        0.00

MODE #      81          82          83          84          85
FACTOR      0.00        0.07        0.00        0.02        2.08

MODE #      86          87          88          89          90
FACTOR      1.02        0.00        0.01        0.01        0.00

MODE #      91          92          93          94          95
FACTOR      0.00        0.00        0.31        0.02        0.04

MODE #      96          97          98          99         100
FACTOR      0.00        0.00        0.00        0.00        0.03

MODE #     101         102         103         104         105
FACTOR      0.00        0.00        0.00        0.00        0.08

MODE #     106         107         108         109         110
FACTOR      0.00        1.22        0.03        0.00        0.00

MODE #     111         112         113         114         115
FACTOR      0.00        0.00        0.00        0.00        0.00

MODE #     116         117         118         119         120
FACTOR      0.00        0.00        0.00        0.00        0.46

MODE #     121         122         123         124         125
FACTOR      0.01        0.25        0.65        0.03        1.44

MODE #     126         127         128         129         130
FACTOR      0.00        0.00        0.03        0.00        0.00

MODE #     131         132         133         134         135
FACTOR      0.00        0.00        0.03        0.01        0.00

MODE #     136         137         138         139         140
FACTOR      0.01        2.12        0.00        0.00        0.00

MODE #     141         142         143         144         145
FACTOR      0.00        0.00        0.00        0.00        0.01

MODE #     146         147         148         149         150
FACTOR      0.06        0.00        0.02        0.01        0.36

- Total Modal Contribution for X Direction =  77.06%
```

Figure 4.13 Modal Contribution Factors (150 Modes)

The pile forces in the interior pier will be used for comparison with the time step integration analysis. After viewing the pile results, it was determined that Pier #2, Pile 2 has the maximum Shear2 force. The force results for Pile 1 are shown in **Figure 4.14**.

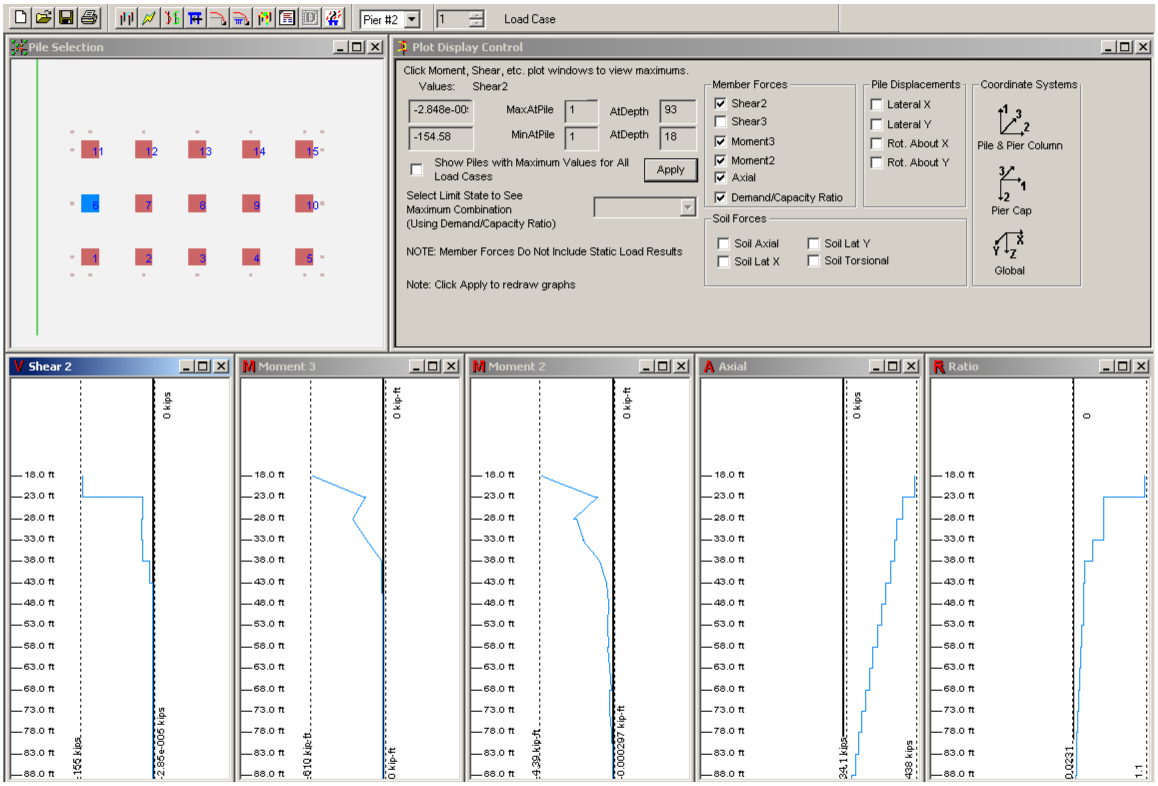


Figure 4.14 Maximum Pile Shear2 Result

Table 4.1 shows a comparison of time step integration and response spectrum analysis results for several important response quantities for the interior pier. The comparison shows a significant difference in some the reported results. The difference can be attributed to several factors:

1) The response spectrum analysis accounted for less than 90% dynamic participation.

2) The response spectrum analysis use the initial soil state, while the time step integration analysis updates the soil state with each time step. In the time step integration analysis, any nonlinearity from the soil behavior is cumulative. The model used in the response spectrum analysis is therefore not as flexible as the one used in the time step analysis (because the soil has not be allowed to soften).

3) The response spectrum results reflect the loads applied from the equilibrium position. Internal forces from static loads, such as self weight, are not included in the force results. Ideally the internal static forces should be added to the internal forces obtained in the response spectrum analysis; however, the response spectrum results represent an envelope of maximum results so it is considered overly conservative to simply add the results.

This example clearly demonstrates that some caution should be used in the assessing dynamic results for a model.

This concludes Example MP-4.