Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

BSI NEWSLETTER

Bridge Software Institute

In this issue, we discuss recently developed features for FB-MultiPier v5.7.0

Engineers making use of software produced by the Bridge Software Institute (BSI) are encouraged to communicate suggestions for new features and program improvements to BSI. These suggestions may be general or very specific to project needs. We firmly believe that you are in the best position to know what those needs are!

NOTE: You can now renew any number of licenses for BSI products by navigating to bsi.ce.ufl.edu, hovering over the Order navigation bar item, and then clicking Renew.

In this release of FB-MultiPier, the enhancements listed below were influenced by YOUR suggestions:

a) Parallel solution of equilibrium equations;
b) Enhanced AASHTO LRFD wind load generation; and,
c) Automated determination of controlling pile cap internal forces.

The FB-MultiPier v5.7.0 enhancements are highlighted below.

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Parallel Solution of Equilibrium Equations

A new solver option has been implemented in FB-MultiPier that performs multiple computation operations in parallel (i.e., simultaneously) during analysis. Such operations are performed for the purpose of more rapidly factorizing and solving systems of equations until equilibrium is achieved to within a specified convergence tolerance.

When the parallel equation solver is utilized, multiple cores are allocated during analysis, and solution of the matrix equilibrium equation is divided among each core. Consequently, the parallel solver generally outperforms the (single-core) serial solver for analysis of relatively large models. Efficiencies (i.e., time savings) depend on the configuration of the bridge structure and may require test runs to determine if benefit is derived from use of parallelization.

Various options for invoking the parallel equation solver during analysis are housed within the Project Settings dialog (Fig. 1). These include specification of the number of cores to utilize during analysis and various advanced options (e.g., continuing with solution even if the requested number of cores is not available). Additional details regarding the advanced parallelization options are found in the FB-MultiPier Help Manual. For instances where the Analysis Core(s) Requested is input as 1 (the default input), then the serial solver is utilized during analysis. For instances where the number of requested cores is greater than 1, then the parallel equation solver is utilized.

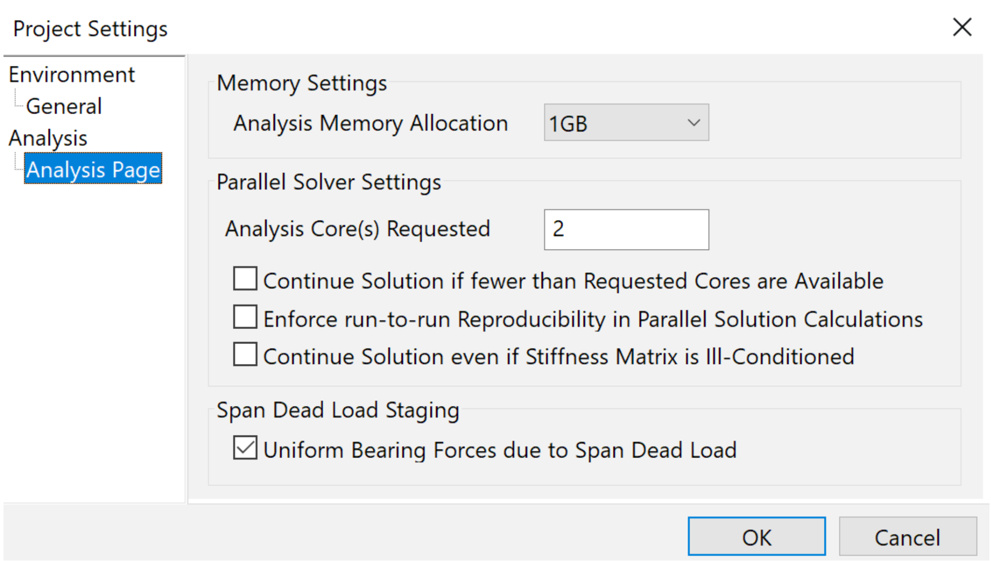


Figure 1. Project Settings dialog

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

As illustration of the potential benefits afforded from use of the parallel equation solver in FB-MultiPier, three demonstration cases are presented in the following. A range of model types is considered, including: pile group, pier, and bridge configurations. All cases considered derive from “real-world” models developed by practicing engineers. For each demonstration case, analyses are carried out using the longstanding FB-MultiPier serial solver, and also, using the newly implemented parallel equation solver with various core allocations. These cases are presented to showcase the potential for time-savings, and to facilitate the decision-making process regarding whether or not analysis of a given model may benefit from use of the parallel equation solver.

Provided below are brief descriptions of each structural configuration considered (and the corresponding finite element models). Subsequently, summaries of the times required to conduct various analyses (serial, parallel) are presented for the three configurations. Collectively, these demonstration cases make obvious the potential for utilization of parallel equation solution when working with relatively large numerical models.

Case 1: Pile Group

As the first of three demonstration cases, consider the pile group configuration shown in Fig. 2. The pile group is fitted with a relatively large (131 ft x 181 ft) pile cap of varying thickness. The cap thickness varies between 20 ft and 40 ft. Portions of the cap that are 40 ft thick are concentrated at the center of the pile group. The pile cap finite element model (Fig. 3) consists of 475 “thick” shell elements. In addition, 432 h-piles (spaced at 7.5-ft intervals) support the pile cap. Further, the top 10 ft of each pile is rigidized, and the group is embedded in a dense sand overlying stiff clay. Loading is concentrated at the center of pile cap (corresponding to the thickened cap portion), and only a single load case is analyzed.

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

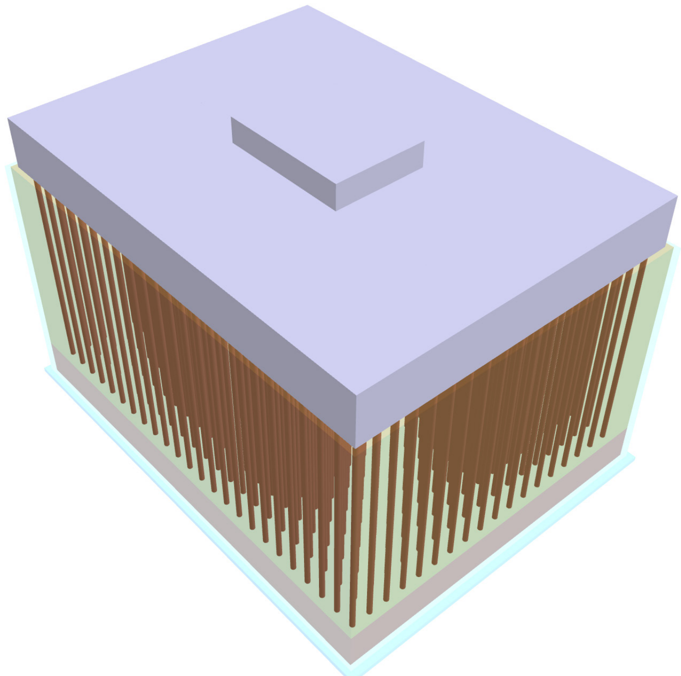


Figure 2. Pile group configuration with 432 piles and 475 shell elements (1 load case)

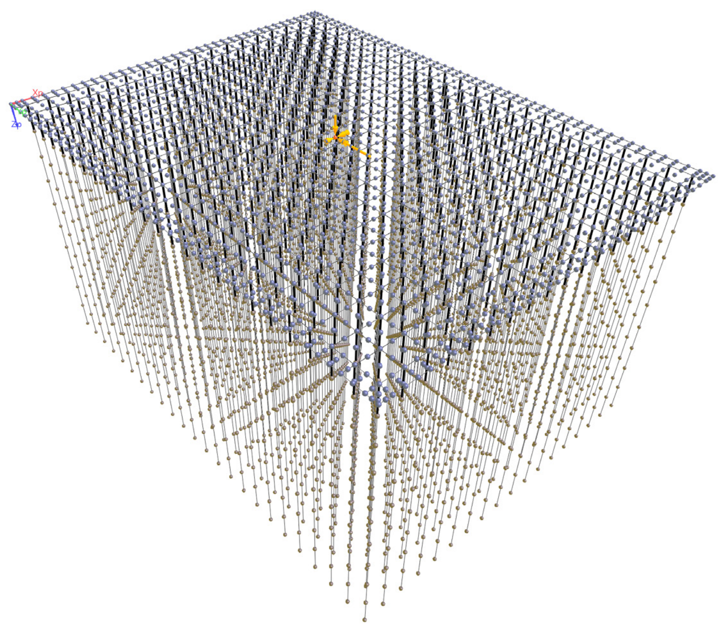


Figure 3. Pile group finite element model

Index Table

FB-MultiPier v5.7.0
FB-Deep v3.0.0
Atlas v7.1

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Case 2: Pier

The second of three demonstration cases consists of a pier configuration, as shown in Fig. 4. Making up the pier structural configuration is a relatively large pile cap (30 ft x 44 ft x 8.5 ft), which integrates a hammerhead pier and 28 drilled shafts (24-in. diameters). The shafts are embedded in a layered profile, consisting of medium dense sand, dense sand, and stiff clay layers. Although the 51-ft tall hammerhead pier column aligns with the rectangular pile cap, the pile layout is that of a skewed configuration. Consequently, a relatively large number of thick shell elements (900) are defined in the finite element model (Fig. 5, Fig. 6).

Loading of the pier comprises 66 AASHTO LRFD load combinations, including STRENGTH-I, III, V, and SERVICE-I limit states. Across the 66 load combinations, nodal load positions range from bearing pads down to the pile cap midplane. In addition, for this demonstration case, lateral stability analysis is carried out to investigate a range of 15 candidate embedment lengths (12 ft to 42 ft). Note that this feature (lateral stability analysis) is automated in FB-MultiPier. To further the illustration, the resulting, program generated plot obtained from the lateral stability analysis for load combination 10 (of 66) is shown in Fig. 7. Lateral displacements vary from 3 in. to 1 in., and the characteristic (asymptotic) curve is evident in the computed response.

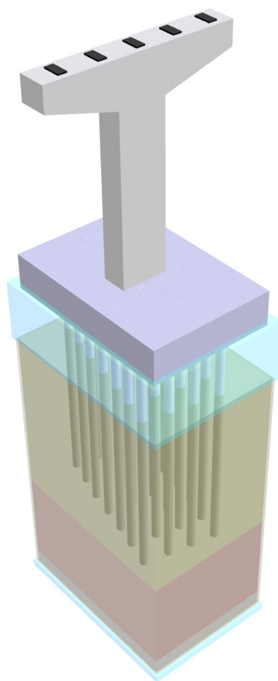


Figure 4. Pier configuration with 900 shell elements, 15 candidate embedment lengths, and 66 AASHTO LRFD load combinations

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

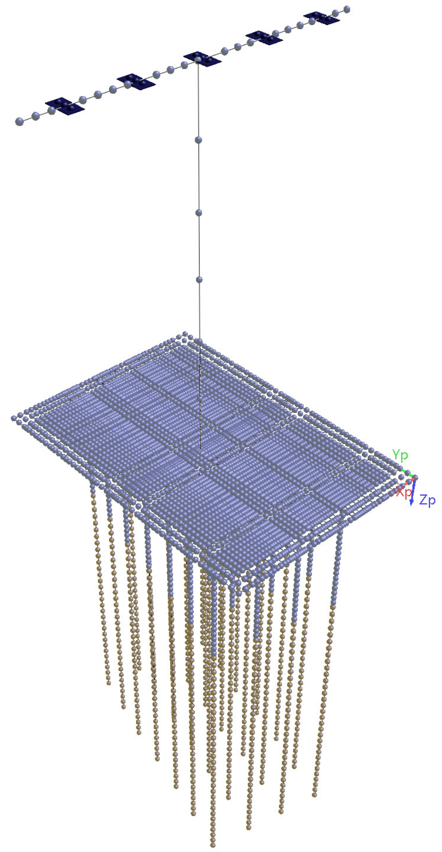


Figure 5. Pier finite element model

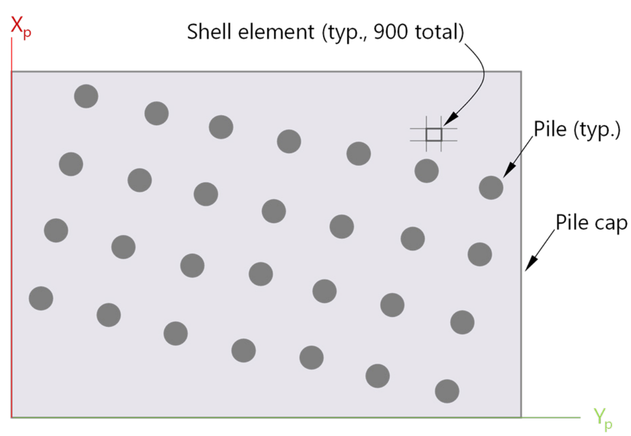


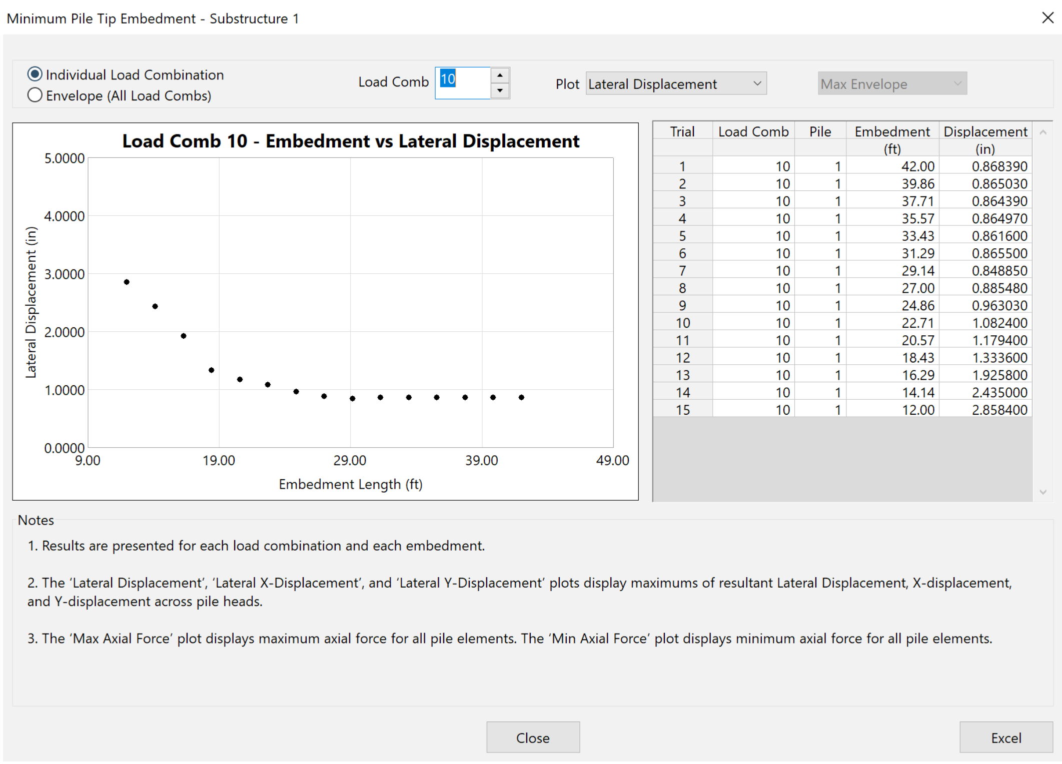
Figure 6. Plan view of pile cap and skewed pile layout

Index Table

FB-MultiPier v5.7.0
FB-Deep v3.0.0
Atlas v7.1

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu



Trial	Load Comb	Pile	Embedment (ft)	Displacement (in)
1	10	1	42.00	0.868390
2	10	1	39.86	0.865030
3	10	1	37.71	0.864390
4	10	1	35.57	0.864970
5	10	1	33.43	0.861600
6	10	1	31.29	0.865500
7	10	1	29.14	0.848850
8	10	1	27.00	0.885480
9	10	1	24.86	0.963030
10	10	1	22.71	1.082400
11	10	1	20.57	1.179400
12	10	1	18.43	1.333600
13	10	1	16.29	1.925800
14	10	1	14.14	2.435000
15	10	1	12.00	2.858400

Figure 7. Program-generated lateral stability analysis results for pier model (load combination 10 of 66)

Case 3: Bridge

The third of three demonstration cases pertains to a 12-span bridge configuration (Fig. 8, Fig. 9), which spans over a navigable waterway. Pier spacings range from 137 ft up to 255 ft at the main span (between the third and fourth piers, from left, in the bridge model). The bridge model is analyzed when subjected to 88 AASHTO LRFD load combinations, with consideration of the STRENGTH-I, III, V, and EXTREME-II limit states.

All 13 piers include rectangular waterline pile caps and varying layouts of plumb/battered prestressed cylinder piles. For example, consider the pier located fourth from left in the overall bridge model, which is a channel pier (Fig. 10, Fig. 11). The pier contains two circular pier columns (of 58-ft height) and a shear strut midway (vertically) between the pile cap and pier cap. Fifteen 54-in. prestressed cylinder piles support the pier, where the columns and piles are joined via the 28-ft x 55-ft x 6.5-ft waterline pile cap. Pile orientations include plumb, single-batter, and double-batter, all terminating 42 ft below the ground surface. The piles are embedded in dense sand and terminate just prior to reaching a layer of limestone.

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Throughout the bridge model, the number of elements per pier is not substantial relative to the configurations of the other demonstration cases considered. For example, for the pier shown in Fig. 10, the pile cap is discretized into 24 thick shell elements, and each pile is divided into 17 nonlinear frame elements. However, the cumulative number of elements in the model across 13 piers exceeds 4,000 elements (approximately 350 shell elements and 3,650 frame elements).

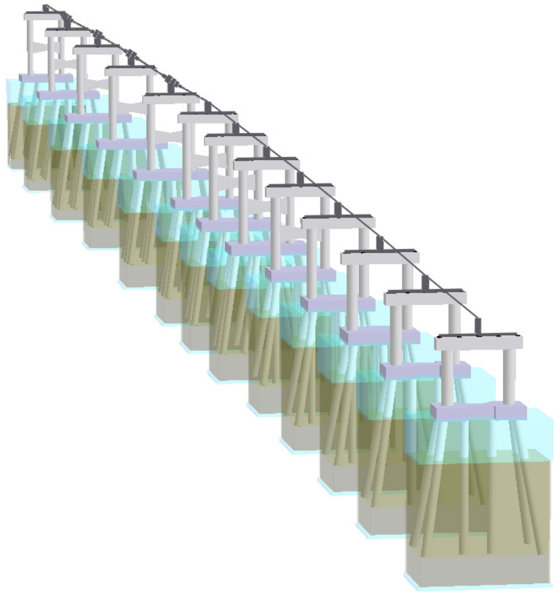


Figure 8. 12-span bridge configuration analyzed using 88 AASHTO LRFD load combinations

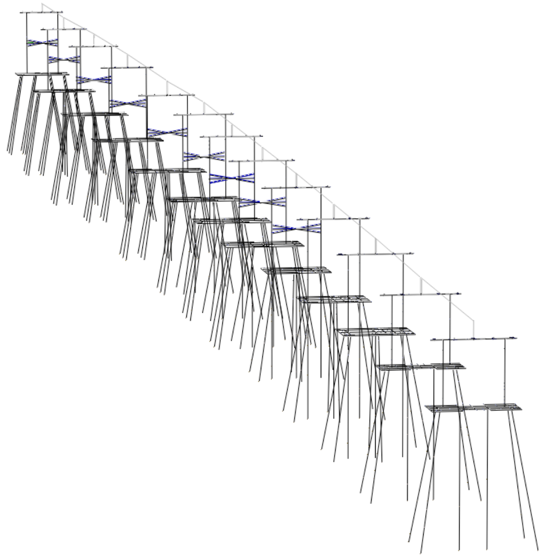


Figure 9. Bridge finite element model

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

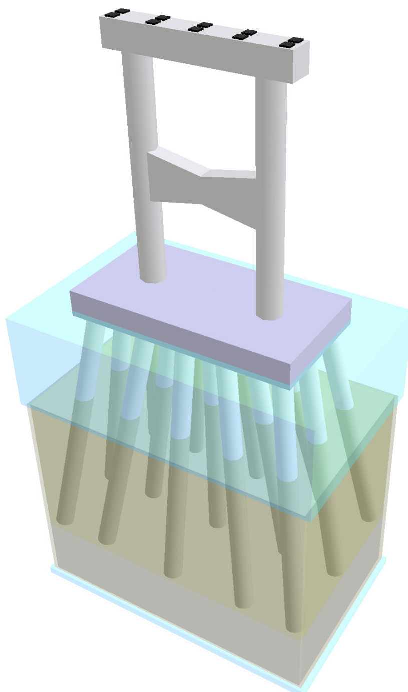


Figure 10. Channel pier (4th from left in bridge model) configuration

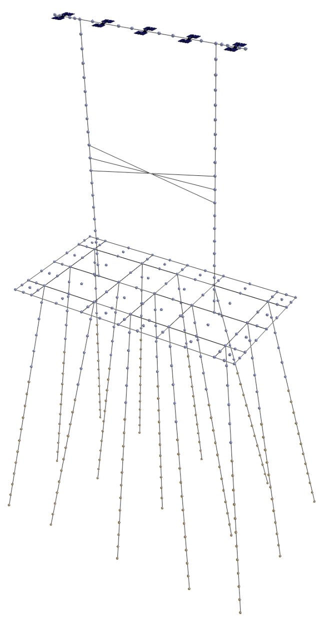


Figure 11. Channel pier (4th from left in bridge model) finite element model

Index Table

FB-MultiPier v5.7.0
FB-Deep v3.0.0
Atlas v7.1

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Comparison of Serial and Parallel Run-times

Plotted in Fig. 12 are comparisons of the analysis durations required for each of the three demonstration cases. For each case, analyses are carried out with allocation of 1 (serial), 2 (parallel), and 4 (parallel) cores. Note that, in Fig. 12, the analysis durations reported for the pier case reflect those of a single candidate embedment length. Comparisons of the run times across all candidate embedment lengths, for the pier model, are presented in Fig. 13. For the models and analyses investigated, the required times to complete the analyses are consistently reduced by significant amounts.

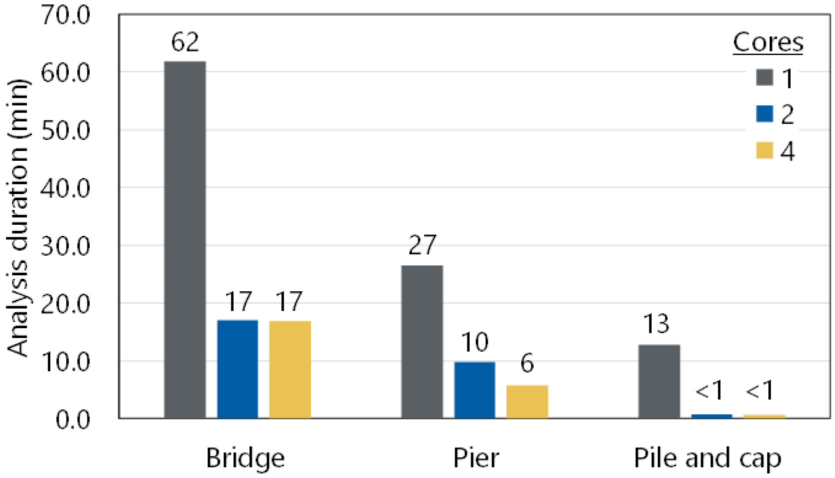


Figure 12. Analysis durations for the pile group, pier (1 candidate embedment), and bridge models

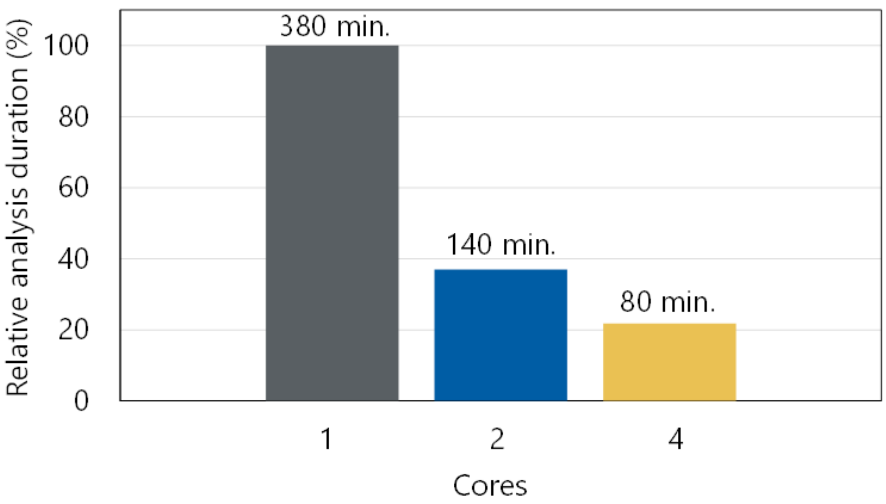


Figure 13. Analysis durations for lateral stability analysis of the pier model (15 candidate embedment lengths)

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Based on investigation of the above (and other) cases, the following approximate criteria may aid in the decision-making process for conducting serial or parallel analysis. Namely, if the model contains one or more of the following characteristics, then decreases in analysis time may be realized through use of parallelization:

- Substructures ~4 or more;
- Piles ~50 or more;
- Nodes ~3,500 or more;
- Pile cap shell elements ~200 or more; or,
- If serial analysis requires ~20 minutes or longer.

Enhanced AASHTO LRFD Wind Load Generation

The FB-MultiPier Wind Load Generator is enhanced in v5.7.0 to reflect codified provisions given in the AASHTO LRFD Bridge Design Specifications, 8th Ed. As context, the AASHTO provisions pertaining to wind loading derive from the model code, ASCE Minimum Design Loads and Associated Criteria for Buildings and Other Structures. The provisions given in the AASHTO LRFD (8th Ed.) specifications, relative to prior editions, utilize wind gust speeds averaged over 3 seconds, whereas prior editions utilized the fastest-mile wind speed.

Technical Considerations

Enhancement of the wind load generation feature in FB-MultiPier v5.7.0 includes the following technical considerations:

1. A design pressure is derived based on a velocity, which in turn, is obtained from contour maps provided in the AASHTO specifications. Also, a pressure exposure coefficient is included to account for elevation and the relative exposure of the structure to wind. Further, a gust factor (typically taken as 1 for bridges) is included, along with a drag coefficient. The latter term is dependent on the bridge component being considered.

2. There are four limit states relative to wind on structure loads (WS): STRENGTH-III, STRENGTH-IV, SERVICE-I, and SERVICE-IV. Load factors for each limit state are defined as 1.0.

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

3. Design wind velocities for STRENGTH-V and SERVICE-I are set to constant values (80 mph and 70 mph, respectively). The design wind velocity for SERVICE-IV is taken as 0.75 of the wind speed derived for STRENGTH-III.

4. The wind pressures applied to bridge superstructures and substructures are calculated for five wind directions. Tables of skew coefficients for the five wind approach angles are taken directly from the AASHTO provisions. The skew coefficients are multiplied by the base design pressures computed from the user supplied wind velocities, height correction, gust factor, and drag coefficient.

5. Wind pressures applied to the substructure are derived from the table of skew coefficients pertaining to columns, as given in the AASHTO provisions.

6. Regarding vertical wind applied to the superstructure: provisions concerning STRENGTH-III and SERVICE-IV are generally unchanged relative to prior editions of the AASHTO LRFD specifications. As an exception, pressures for SERVICE-IV are calculated from base pressures of 0.01 ksf rather than 0.02 ksf.

7. The vertical over turning wind force (as applicable to the zero-degree wind direction only) is not included in the FB-MultiPier generator, and corresponding nodal loads must be manually applied models.

New User Interface (UI) Features

Given the above technical considerations, dialogs, and controls pertaining to the AASHTO LRFD Wind Load Generator have undergone substantial enhancements. Presented below are brief introductions to the newly developed UI features that comprise the AASHTO LRFD Wind Load Generator in FB-MultiPier v5.7.0. The main dialog for generating wind loads is shown in Fig. 14.

Index Table

FB-MultiPier v5.7.0
FB-Deep v3.0.0
Atlas v7.1

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

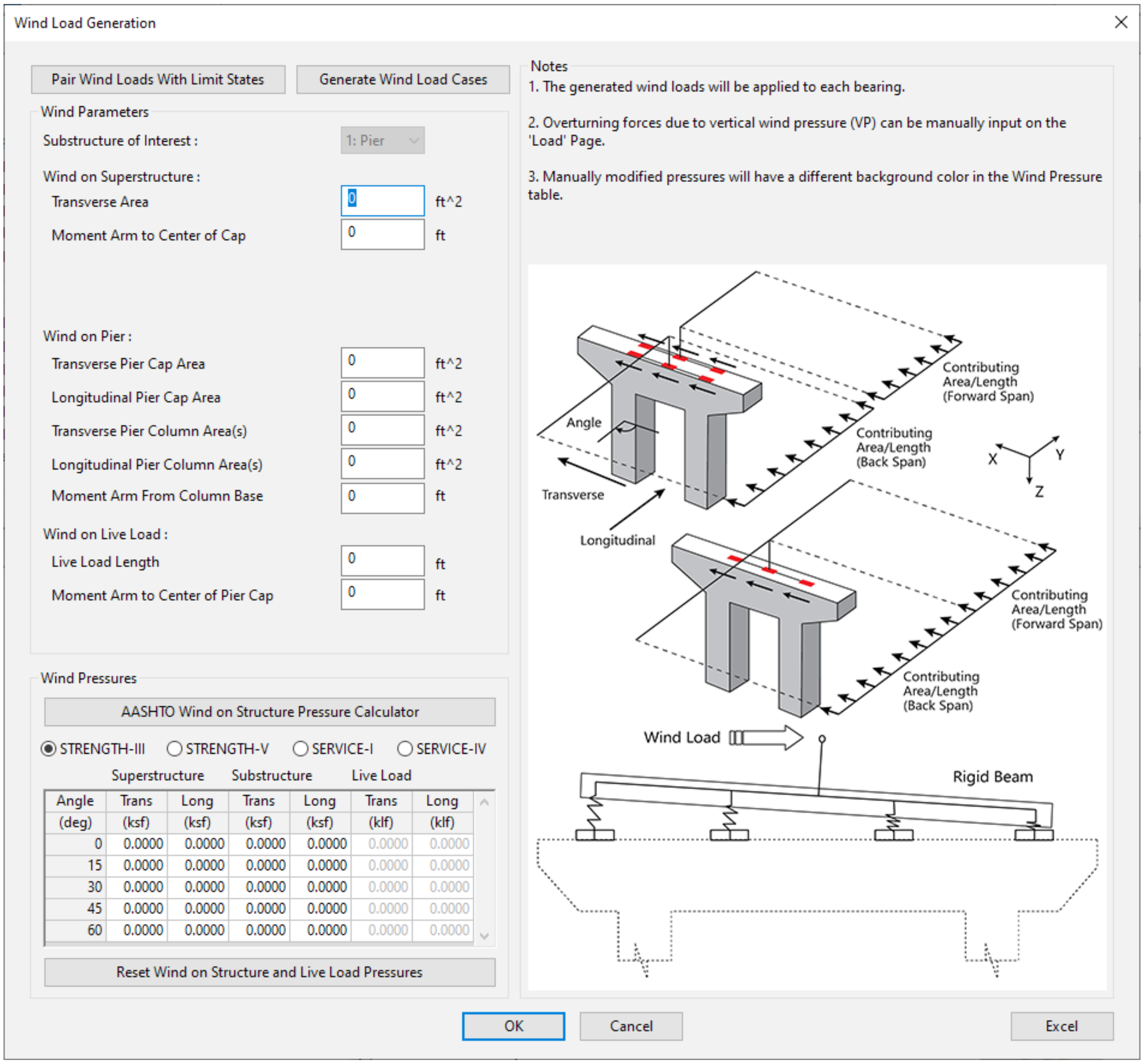


Figure 14. Wind Load Generation dialog

Wind tributary areas and associated moment arms are input from within the top portion of the main dialog for wind load generation (Fig. 14, top-left). These parameters are utilized in FB-MultiPier for automated calculation of the necessary wind forces, which are resolved at the bearing locations of each substructure. Parameters such as: wind speed, gust effect factor, pressure exposure coefficient, and drag coefficient are input from within the AASHTO Wind Pressure Calculator (Fig. 15), which is accessible from within the main dialog (Fig. 14, middle-left). The wind pressures are calculated and displayed for the four applicable limit states (STRENGTH-III, STRENGTH-IV, SERVICE-I, and SERVICE-IV), and can also be manually edited.

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

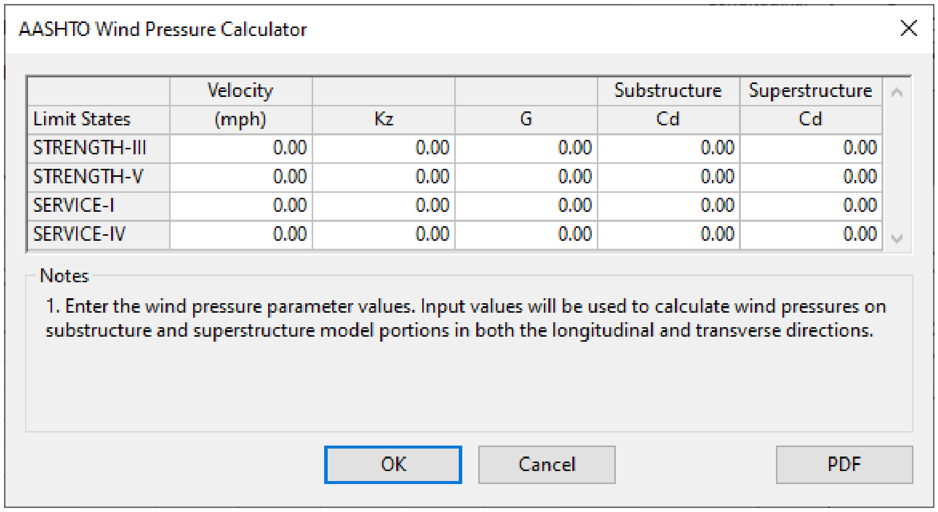


Figure 15. AASHTO Wind Pressure Calculator dialog

The “Pair Wind Loads with Limit States” dialog (Fig. 16) allows the engineer to specify the number of wind load cases generated—and to be applied—for any or all AASHTO Limit States that include wind loading. This data is then utilized by FB-MultiPier to automatically create the pertinent AASHTO Load Combinations (in accordance with selections made by the engineer).

The Load Case Manager dialog (Fig. 17) is accessible directly from the Design Specs. page of the Model Data window, and is used to control which limit states are to be considered. This dialog provides an additional means for engineers to specify the number of wind on structure (WS) and wind on live load (WL) instances to be created. For example, wind cases in addition to those automatically generated by FB-MultiPier (via the Wind Load Generator) may be manually created, and then, respective sets of nodal forces can be manually defined from within dialogs associated with the Load page of the Model Data window.

Index Table

FB-MultiPier v5.7.0
FB-Deep v3.0.0
Atlas v7.1

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

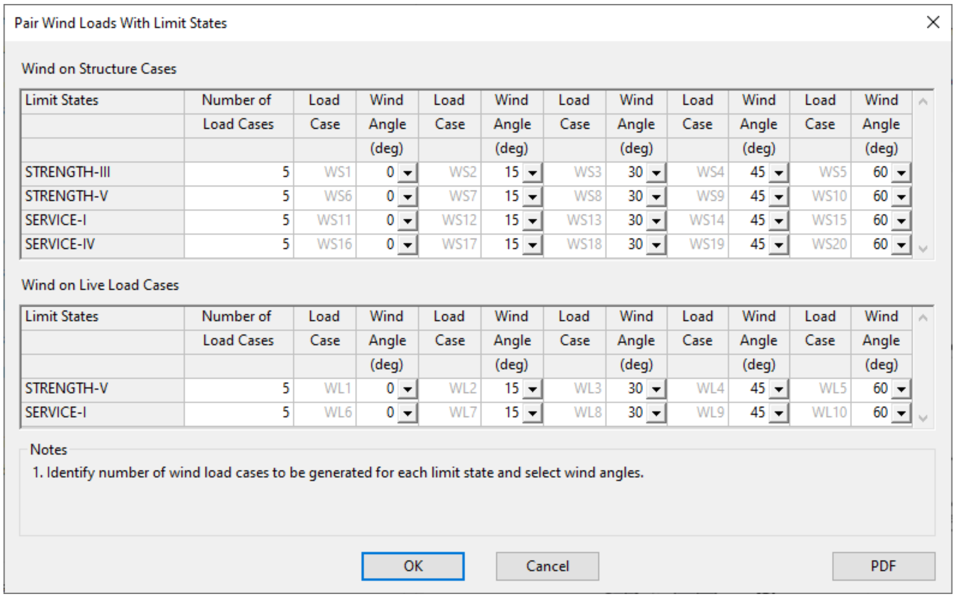


Figure 16. Pair Wind Loads with Limit States dialog

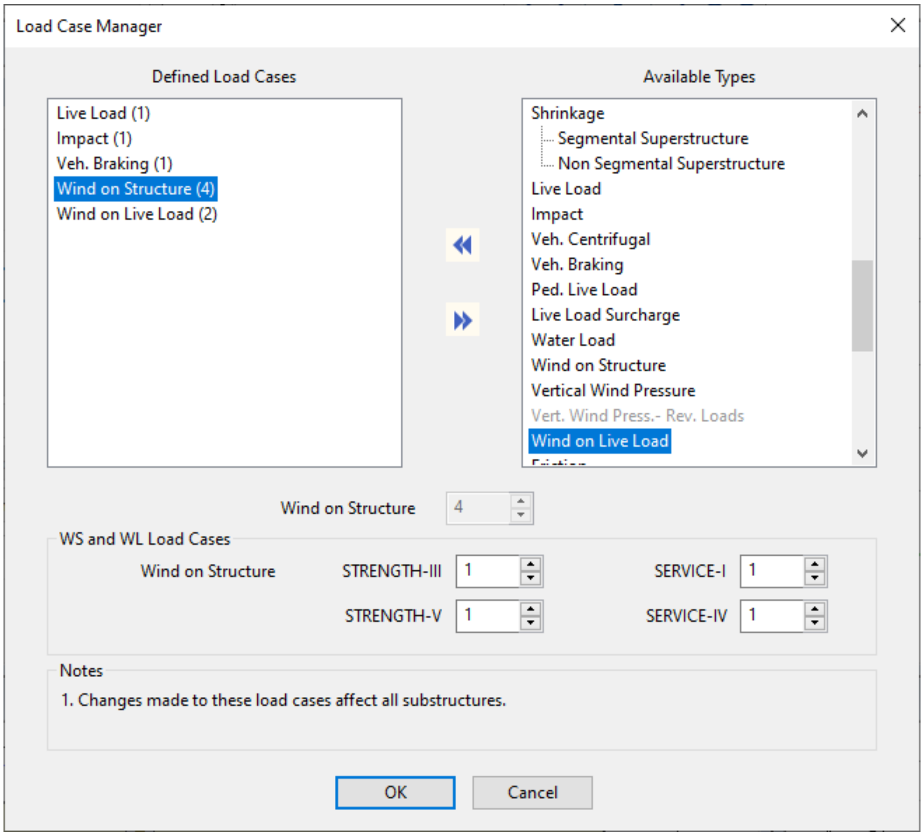


Figure 17. Load Case Manager dialog

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Automated Determination of Controlling Pile Cap Internal Forces

The Pile Cap Forces dialog (Fig. 18) aids engineers in quantifying the controlling shear forces and moments that develop within pile caps. In turn, the controlling forces can be used for deciding upon pile cap reinforcement layouts. The Pile Cap Forces dialog generates the maximum and minimum moment (about Xp or Yp) and maximum magnitude shear (parallel to Zp) along pile cap nodes.

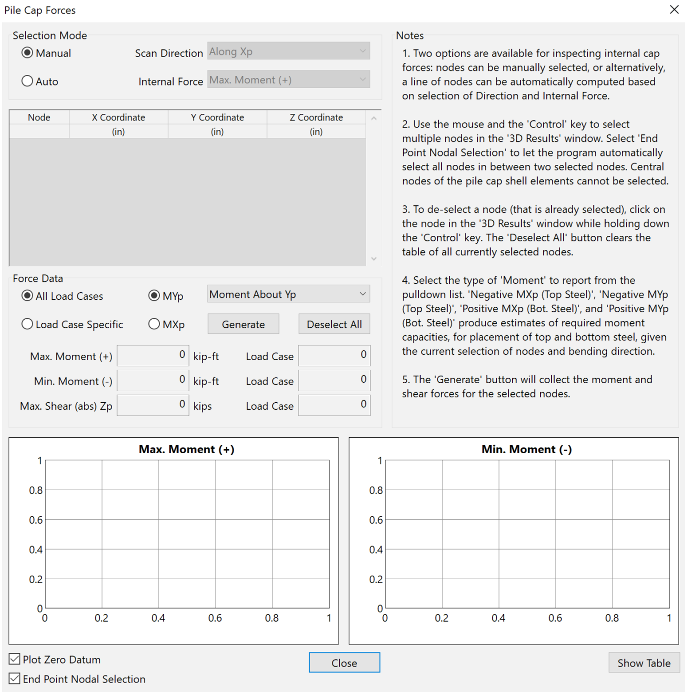


Figure 18. Pile Cap Forces dialog

Pile cap internal forces can be quantified for a specific load case (or load combination) of interest, or across all load cases (combinations). The dialog also provides, per unit length, maximum and minimum moment plots along the selected nodes. In addition, the Auto selection mode identifies the line of nodes along a single row in the Xp or Yp direction to be automatically identified, based on the selection of an internal force of interest.

For FB-MultiPier v5.7.0, two modes of pile cap force calculations are available: Manual selection of nodes; or, Auto selection of nodes. The Auto selection of pile cap nodes feature is newly implemented in the software, and identifies the location (line of nodes) along the pile cap that produces the controlling internal force (as selected from maximum moment, minimum moment, or maximum absolute vertical shear).

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Use of the automatic selection feature proceeds as follows:

1. Open and run an analysis with a model that contains a pile grid size of 2x2 or greater.

2. Click the 3D Results button on the top toolbar, and right-click within the 3D Results window. Then, select the Pile Cap Forces option from the context menu. Choose Auto selection in the top left corner of the Pile Cap Forces dialog.

3. Select the desired "scan" direction for the slices of pile cap nodes to be analyzed by the Auto selection feature (Along Xp or Along Yp). Note that selection of Along Xp will result in the Auto selection feature only considering subsets of nodes that are parallel to the Xp axis. Likewise, selection of Along Yp will result in the Auto selection feature only considering slices of nodes that are parallel to the Yp axis.

4. Select the Internal Force (Max. Moment, Min. Moment, or Max. Shear (abs) Zp).

5. Select the desired option from the Moment pulldown list. Note that the signs of moments generated using the Pile Cap Forces feature are such that positive moment values correspond to tension in the bottoms of shell elements, while negative moment values correspond to tension in the tops of shell elements.

6. Click the "Generate" button and the program will identify the line of nodes (in the desired direction) with the greatest internal force, and display the maximum and minimum moment plots.

As illustration, consider the highly simplified, pile group configuration shown in Fig. 19. A series of downward nodal loads (for a total load, P, of 900 kip) are applied midway across the span (L = 70 ft) between the two rows of relatively rigid, pinned-head piles (Fig. 19a). No other loading is present in the model. The resulting deflected shape is presented in Fig. 19b. Given the highly simplified configuration, the maximum moment about the Xp axis is predicted to be approximately P*L/4. Supplying the respective model parameters results in: (900 kip * 70 ft) / 4 = 15,750 kip-ft. Further, the maximum moment is predicted to occur at mid-span.

A set of corresponding results obtained from use of the Auto selection feature in the Pile Cap Forces dialog is presented in Fig. 20. The results indicate (consistent with the predictions above) that the line of nodes associated with the maximum moment is located midway between the two pile rows (i.e., the Yp coordinates are 480 in.). In addition, the computed moment value (about the Xp axis) is 15,743 kip-ft, which is practically identical to the predicted value of 15,750 kip-ft.

Index Table

FB-MultiPier v5.7.0
FB-Deep v3.0.0
Atlas v7.1

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

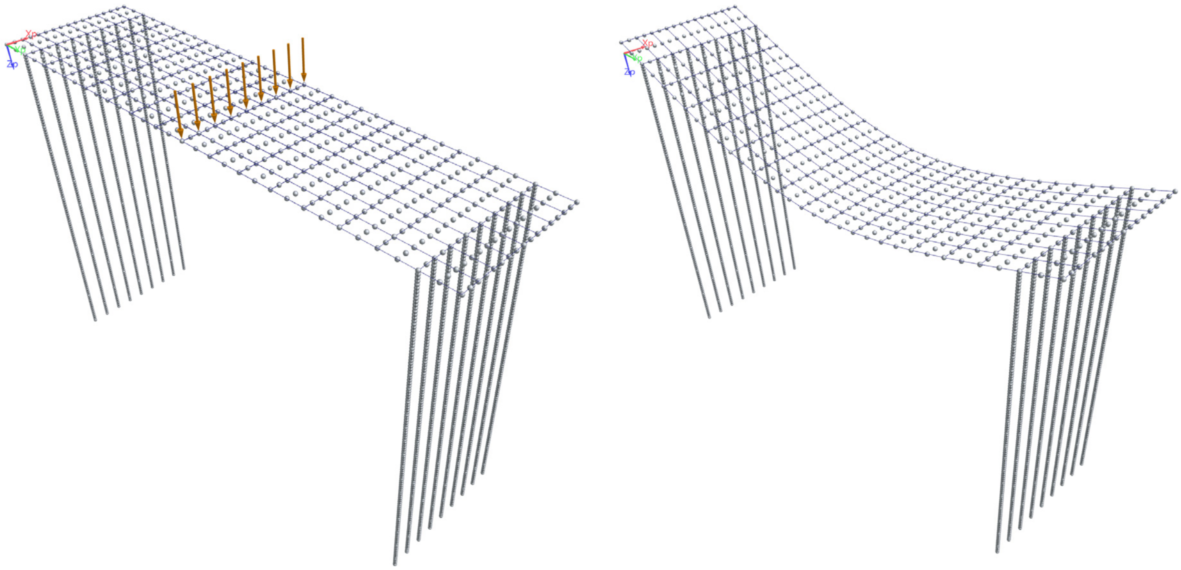


Figure 19. Illustrative pile group model: a) Finite element model with mid-span loading; b) Deflected shape

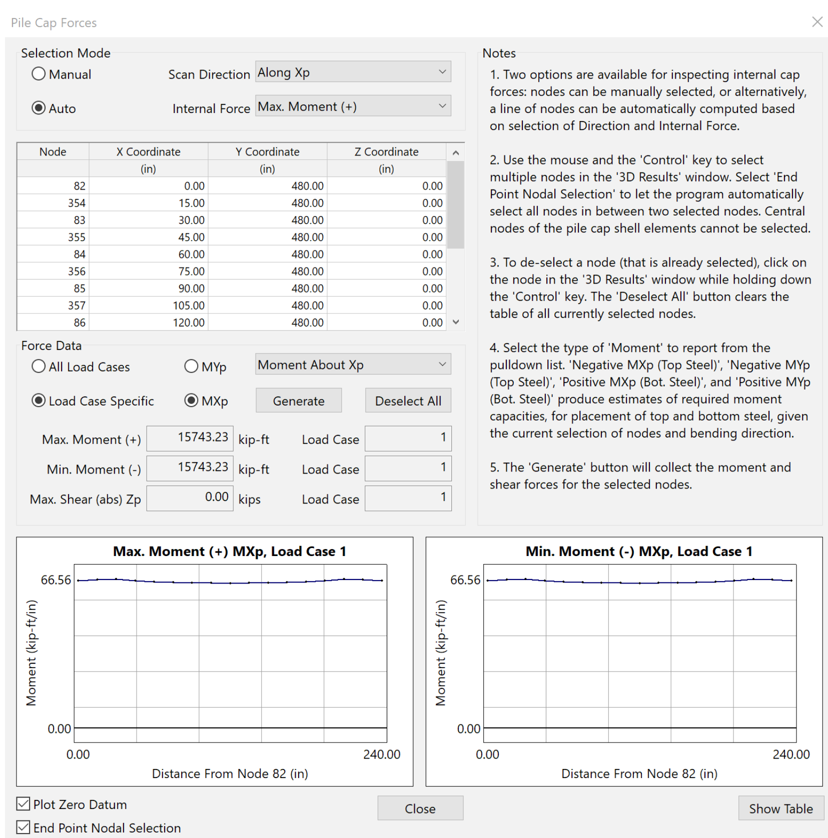


Figure 20. Pile Cap Forces dialog with illustrative use of Auto selection mode

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

BSI Program Status

FB-MultiPier v5.7.0 Download a FREE demo today!

Released August 2020 - Continuing Development - Technical Support Available

FB-MultiPier allows for the modeling of bridges, bridge piers, pile bents, and other foundation structures. In addition to allowing for multiple load cases and AASHTO load combinations, FB-MultiPier is also capable of performing dynamic analysis (time-history and RSA). For more information about FB-MultiPier, click **here**.

FB-Deep v3.0.0 Download a FREE demo today!

Released Feb 2020 - Continuing Development - Technical Support Available

FB-Deep is used to estimate the static axial capacity of drilled shafts and driven piles. The methodology is based upon Federal Highway Administration (FHWA) reports. FB-Deep guides the user through pile and shaft materials data, shape and dimensional inputs, soil properties, and boring log info. For more information about FB-Deep, click **here**.

Atlas v7.1

Released June 2019 - Limited Web Support Available

Atlas is a finite element analysis program that is used for the design/analysis of cable supported traffic signal systems. The Atlas program models dual cable supported systems including single-point, and two-point attachments systems. For more information about Atlas, click **here**.

Contact BSI

If you need to contact BSI for any reason you can use any of the methods below:
Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu

Mailing Address:

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Index Table

Bridge Software Institute
University of Florida
PO Box 116580
Gainesville, FL 32611

Online: bsi.ce.ufl.edu
Email: bsi@ce.ufl.edu