Bridge Pier Analysis for Ship Impact

M. I. Hoit, Mike McVay, and Scott E. Breneman, *University of Florida, Gainesville*

A comprehensive computer program that analyzes bridge piers including the pier structure, nonlinear piles, and nonlinear soil interaction is discussed. The program, LPGSTAN (laterally loaded pile group and structural analysis), includes pile group effects, missing and battered piles, and bridge connection effects. The program is unique in that the analysis models are defined by using the designer specifications such as pile spacing, number of columns, and soil layer information. All model definition and result review are performed in a graphical environment. The program has not been tested by comparisons with data in the literature on pile group tests and an extensive series of centrifuge tests performed at the University of Florida. The program is in use at the Florida Department of Transportation and consulting firms throughout Florida. An overview of the program's assumptions, modeling, and capabilities is given.

Over the past decades the abilities of general purpose analysis software have increased dramatically. With the increased abilities of these packages has come an increase in the complexity of using them for specific problems. A general purpose structural analysis package requires huge amounts of time to define and iterate on a complex structural design. These packages generally do not take advantage of the similarities of structures being repeatedly modeled by a particular individual or institution. Engineers must deal with the specifics of software modeling when it would be much more efficient to work with the structural design parameters directly. To allow this, structural analysis packages must be designed to fit the needs of specific types of analysis. For this reason the Department of Civil Engineering at the University of Florida, with major funding from the Florida Department of Transportation, has developed a structural analysis package that can be used to analyze bridge foundations that include a pile group and pier structure with nonlinear soil interaction under ship impact loading.

The analysis program that has been developed, LPGSTAN (laterally loaded pile group and structural analysis), is a nonlinear finite-element analysis program designed for analyzing bridge pier structures composed of a pier cap and columns supported on a pile cap and nonlinear piles with nonlinear soil. This analysis program couples standard structural finite-element analysis with current nonlinear static soil models for axial and lateral loading to provide a robust system of analysis of the coupled bridge pier structure and foundation system. LPGSTAN performs the generation of the finite-element model internally given the geometric definition of the structure and foundation system. This allows the engineer the opportunity to work with the design parameters directly and lessens the bookkeeping necessary to create and interpret a model. Coupled with the analysis program is a graphical preprocessor, LPGGEN, and a postprocessor, LPGPLOT. These programs allow the user of LPGSTAN to view the structure while generating the model and view the resulting deflections and

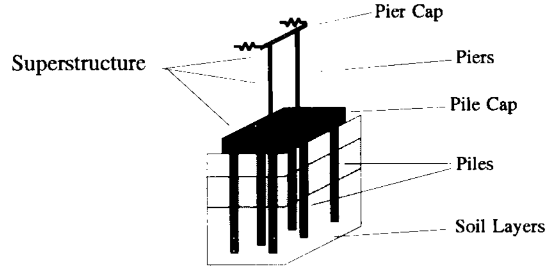

FIGURE 1 LPGSTAN model components.

internal forces in a graphical environment. LPGGEN provides an efficient method for defining the configuration of the structure to be analyzed. After analysis LPGPLOT can plot the undeflected shape, the deflected shape under the load conditions, and the internal stresses and forces in the members. The major components of the LPGSTAN model can be seen in Figure 1.

LPGSTAN is being used throughout the state of Florida for the design of bridge piers. Both the state department of transportation as well as consultants are using and verifying the program on active design projects. Five other states including Washington, Oregon, Oklahoma, Utah, and Louisiana are all testing the software for possible adoption. As the result of an FHWA workshop on the design of highway bridges for extreme events, LPGSTAN was selected as an acceptable analysis solution (*1*).

All of the programs in the LPGSTAN package have been developed to be portable. Working DOS and Unix versions are available. The analysis can be performed in a reasonable time (less than 10 min) on a personal computer with a minimum of 8 megabytes of memory and a 486-based central processing unit or better.

LPGSTAN Modeling

The structural components of the LPGSTAN model are all standard finite elements. The piles are modeled either linearly or nonlinearly by a discrete-element approach. The pile-soil interaction is modeled with lateral P-Y curves and axial T-Z curves. Also pile-soil-pile and group effects are modeled in two ways. The major structural components of the system are the piles, pile cap, pier columns, and pier cap. The piles, pier columns, and pier cap are all modeled by using three-dimensional beam elements. The pile cap is modeled by using three-dimensional flat shell elements. Additional beam elements are used in the modeling to connect the pier columns to the pile cap. These connector elements do not correspond to any true structural component of the bridge foundation structure but act to distribute the column load over an area and stiffen the cap. A view of a typical structure as show in LPGSTAN is given in Figure 2.

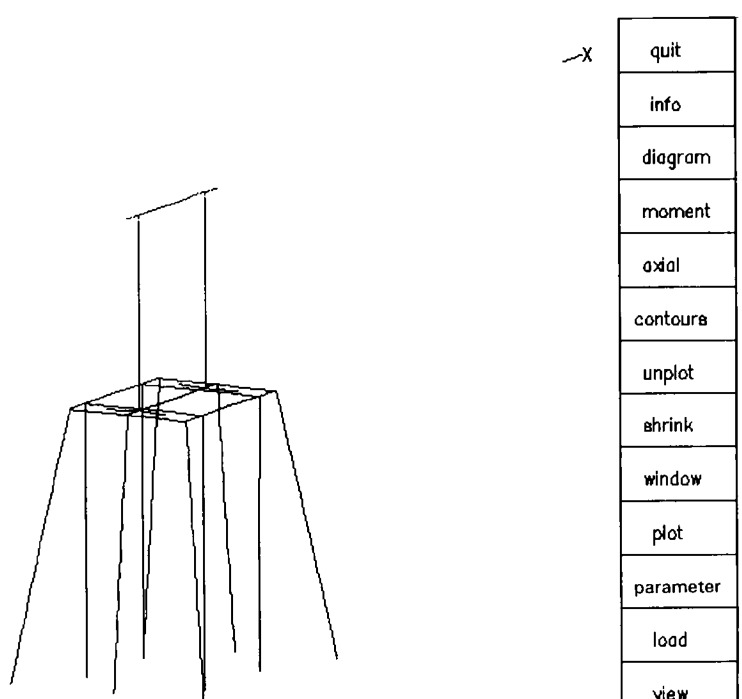

FIGURE 2 Example structure, three-dimensional view.

Pile Modeling

Each pile is modeled with 16 three-dimensional beam elements. These can be standard linear beams or nonlinear discrete-beam-element models (2). The nonlinear piles can model reinforced concrete for both square and round piles. Both elements model biaxial bending, torsion, and axial deformations. If the piles have a freestanding length above the soil layer, the first pile element connects from the pile cap to the top of the soil layer. For the nonlinear model the first element is composed of many subelements. This ensures the accuracy of the nonlinear behavior in the freestanding portion. The rest of the elements are of equal length down the remaining portion of the pile. If there is no freestanding length then all elements in the pile are the same length. For the linear elements the required properties are the moments of inertia, torsional constants (polar moment of inertia), material elasticities, and areas and diameters of the piles. The nonlinear elements require the concrete cross section, steel, and corresponding stress strain behaviors to be specified. The piles can be battered at angles from the plumb line in both the x and y directions. When the piles are battered the total length of the pile does not change but the total depth decreases. Additionally, the pile connection with the pile cap can be either a pinned or a fixed connection.

Pile Cap

The pile cap is modeled by using nine-node shell elements. The shell elements are based on Mindlin theory and include special reduced integration to account for shear deformations and to avoid zero-energy modes. Inplane torsion effects are also included. The pile positions make up the four corner nodes of each element. Additional nodes are placed at the midpoints of adjacent piles to give sufficient nodes for modeling the flexibility between piles. The shell element used can model both bending and shear in the pile cap. The shell elements require the thickness of the cap and the Poisson's ratio and Young's modulus of the cap material.

Pier Columns and Pier Cap

The pier columns and pier cap are modeled with the same linear beam elements as the linear piles. The pier cap connects adjacent pier columns and can have a cantilever length extending from the end pier columns. The pier columns and pier cap girders have independent sets of properties. The center of the girder that connects adjacent columns has its own independent set of properties. This center property can be set to zero to model unconnected pier columns. Each of these properties requires the moment of inertia, torsional inertia, cross-sectional area, material elasticity, and shear modulus.

LPGSTAN can also model the cross bracing between the pier columns, pile cap, or pier cap. Additional beam elements can be added between any nodes in the pile cap and pier columns. These additional elements have properties similar to those of the pier columns and pier cap.

Connectors

The top nodes of the pile elements are directly connected to the shell elements in the pile cap. Because of the general placement capabilities of the columns, the bottoms of the pier columns will not always match up with node locations in the pile cap. Because of this additional connector elements are placed between the nodes in the pile cap and the bottom node of the pier columns. These connectors eliminate the stress concentration in the pile cap and represent the pier column width. The connector elements are much stiffer than the pier columns to ensure the modeling of a stiff connection between the pier and the pile cap. LPGSTAN automatically sets the position and materials for these connector elements.

Deck Connectivity

In real structures the behavior of the foundation is not isolated from the behavior of the connected bridge deck and supports. The ability to include this interaction is provided by adding springs to the pier cap. In typical situations the springs would only be placed on the pier cap girders representing the bridge girders. It is acceptable to also place springs at any node in the pier columns and pile cap. Each spring can have a stiffness in each of the three translational and rotational directions. The values of these springs can be found through an analysis of the bridge deck by alternate static analysis methods or by considering the bridge deck as a large composite beam.

Soil Modeling

The soil modeling used by LPGSTAN includes the axial and lateral effects of pile displacements (3). Group effects are included through the use of P-Y multipliers for the pile rows or the use of pile-soil-pile springs.

When modeling the lateral soil-pile interaction the user has the choice of several P-Y curves, depending on the soil conditions at the site. For sand there is O'Neill's recommended P-Y curve and the P-Y model from Reese, Cox, and Koop. The latter model for sand is included in FHWA's COM624P, which is a single-pile lateral load analysis program (*4*). For clay there is O'Neill's model, Matlock's representation for soft clays below the water table, Reese's model for stiff clays below the water table, and Reese and Welch's model for stiff clays above the water table. All of these clays except for O'Neill's are also included in FHWA's COM624P (*4*). A user-defined P-Y curve is also available.

The axial effects include soil properties along the length of the pile as well as a soil tip model. The axial soil along the length is given by the T-Z curves of McVay et al. (*3*). The soil tip model is based on similar T-Z curves and an ultimate tip resistance. If no soil is being used or the tip properties are not known, the tip can be either vertically constrained with a linear tip spring or constrained in all directions of motion.

One-way group effects are included in the analysis by using P-Y multipliers applied to the P-Y curves of individual piles. These multipliers reduce the resistance (P) of the soil to which they are applied. Experimental values for P-Y multipliers have been determined for only a few group configurations, and these have been for loading in the principal directions of the pile group (*5*). The choice of the P-Y multipliers is left to the users of LPGSTAN. When the net horizontal loading is not in one of the principal directions (x or y), the P-Y curves are superimposed for both directions.

As part of the research in developing the programs, centrifuge studies were performed on different pile group configurations. These studies had the dual purpose of (a) developing additional P-Y multipliers for different pile configurations and (b) verifying the program. The results of these tests are given by McVay et al. (*3*).

Figure 3 provides a comparison of the results of the LPGSTAN program with those of the centrifuge study for an 18-in. round steel pipe pile that was 45 ft long in a medium-dense sand simulation. These results are typical for comparisons between the program and the centrifuge tests.

An alternate method of including group effects is to use pile-soil-pile springs. These are derived from considering the soil between the piles as a linearly elastic medium interconnecting the piles. This method is computationally expensive and is awaiting further advances. It is not recommended for use on microcomputers.

Loading

Structural loading consists of point loads applied to the superstructure. Multiple load cases can be defined in the program. Multiple loadings are treated separately as independent analyses. The multiple load specification is for user convenience. LPGSTAN was originally developed to handle ship impact loading. This type of loading is calculated following AASHTO, which results in a concentrated static load. Any type of static loading such as wind or traffic can be represented by an equivalent set of concentrated loads. However, these loads are currently not automatically defined.

Nonlinear Solution Method

Because the soil and pile models are nonlinear, LPGSTAN performs an iterative solution process. To minimize the work at each iteration the linear portions of the structure that include the pile cap and pier struc-

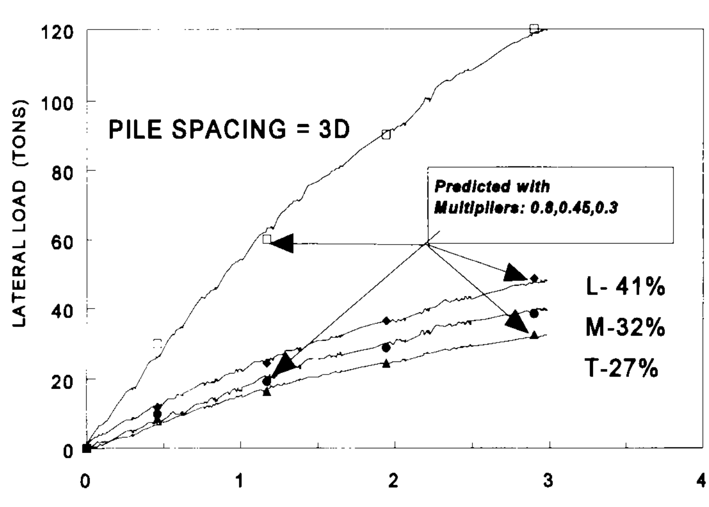

FIGURE 3 Typical comparison of centrifuge data and LPGSTAN.

ture are statically condensed before the iterative solution begins. The iterative method uses a secant method approach for the solution of the nonlinear equations. At each iteration, LPGSTAN finds the stiffness of the soil and piles (if nonlinear) for the current displacements, assembles the stiffness matrix, and solves for new displacement values. Convergence occurs when all nodes in the piles are in static equilibrium. Before this occurs the nodes will have out-of-balance forces. LPGSTAN uses the largest value of the out-of-balance forces as the measure of convergence. This maximum out-of-balance force should be a small percentage of the total load applied to the structure. The final converged displacements are then used to find the internal loads in the pile elements, and the forces in the superstructure are recovered. Users can check if the convergence tolerance is sufficiently small by looking at the printed out-of-balance forces at the pile nodes.

Analysis Procedures

To perform an analysis with LPGSTAN an engineer can follow this simple procedure. After having determined the desired configuration of piles, spacing, and soil, the engineer creates an input file that describes the structural configuration. This can be done graphically with the LPGGEN pre-processor. By using LPGGEN the structural properties and configuration as well as the soil and loading of the structure are defined.

Once the structure has been fully defined LPGSTAN is run. LPGSTAN analyzes the structure and writes the generation and solution information to the specified output file. By next running LPGPLOT the user can view the deflected structure and the resultant forces. Specific information about the internal forces and displacements at particular positions in the structure are contained in the output file. This information can be useful for finding the forces and displacements at particular locations of interest that have been located with LPGPLOT.

Both the LPGGEN preprocessor and the LPGPLOT postprocessor are graphical menu-driven programs that require the use of a pointing device. These programs show the current structure from a three-dimensional view that can be rotated and scaled to allow the user to visually inspect all of the components of the structural model.

Using LPGGEN

The graphical LPGGEN preprocessor is used to define the desired structure and to create the input file for LPGSTAN. While editing, LPGGEN displays the structure in a three-dimensional view. It also allows the user to zoom in on parts of interest. Most of the operations require the user to select an option from a menu and then type any required values. Many properties of the structure can be edited by clicking on the point of interest. A commonly used interface method of displaying a table of properties and their corresponding values is shown in Figure 4.

LPGGEN includes all of the menu commands required to make design modifications to the structure, including pile cap, pile configuration, pier columns and cap, soil, and loading. Figure 5 shows the starting screen of LPGGEN. This menu controls the flow of the problem definition.

Many of the structural properties can be edited while viewing the structure in a three-dimensional view, as seen in Figure 5. The notable exceptions to this are when editing the pile configuration and when defining the soil layers. For the configuration of the pile group, the piles and dimensions are displayed in the standard plan view of the group. An example of the plan view is provided in Figure 6. The batter, relative pile size, and cap overhang are shown.

The soil layers are defined while showing the layers graphically with a single representative plumb pile. An example of this can be seen in Figure 7. Here the user can graphically see the defined layers and can easily access the properties of each layer individually. The free length of the pile is shown proportionally to the true length of the pile.

Using LPGPLOT

The LPGPLOT postprocessor uses a three-dimensional view and interface similar to that used by LPGGEN. It allows the user to visually display a model that has been analyzed by LPGSTAN. Geometry, element and node numbering, and element connectivity can be checked visually by using LPGPLOT. In addition, frame element local coordinate systems, stress contours, displacement contours, deflected shapes, and pile moments of the model can be displayed. Stress and displacement contours can be plotted only for the pile cap. A screen similar to that in Figure 8 shows the main menu items when LPGPLOT is run.

LPGPLOT also plots the deflected shape for each of the load cases performed in the analysis. The magnitudes of the deflections are exaggerated to make them easily visible. Deflections of the individual nodes can be inspected to obtain numeric values. The same structure shown in Figure 8 but shown from a different point of view, including deflections, is given in Figure 9.

LPGPLOT can also show the relative axial forces and moments in each of the structural beam elements (piles,

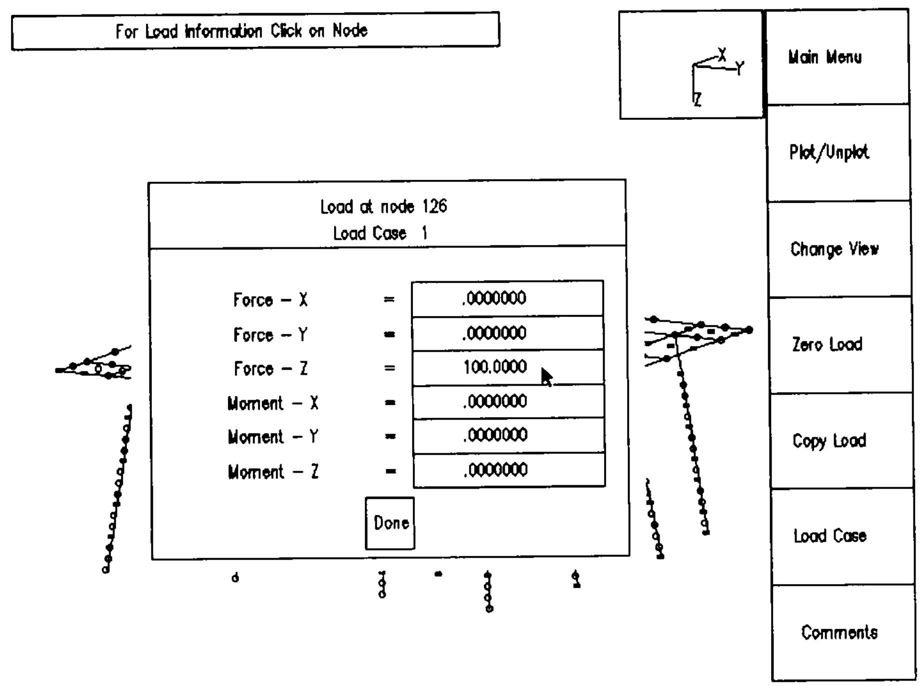

FIGURE 4 Example of table of values of LPGGEN.

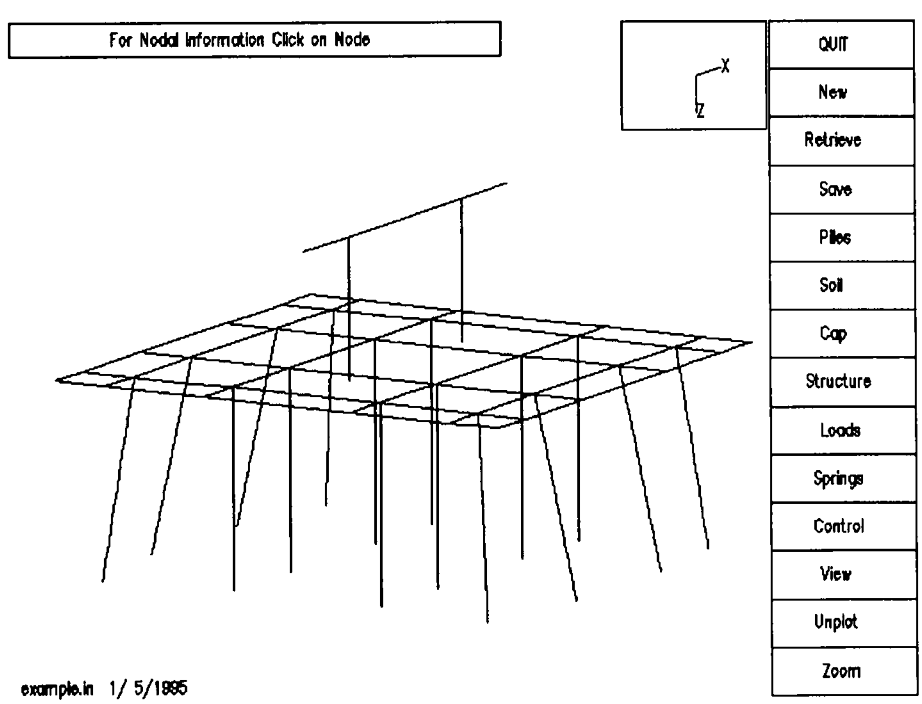

FIGURE 5 Main menu of LPGGEN.

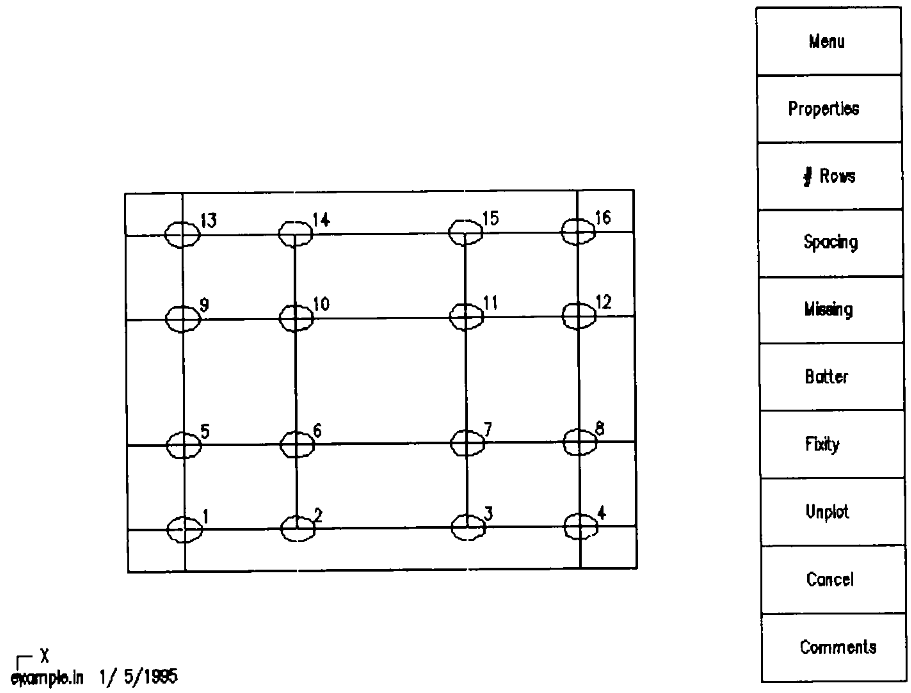

FIGURE 6 View of piles with batter.

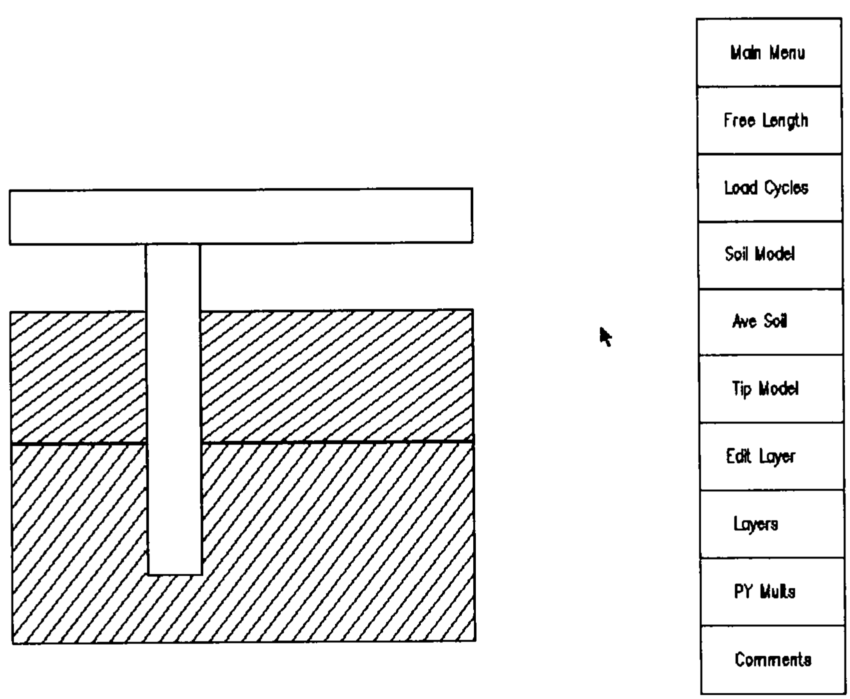

FIGURE 7 Example of soil menu view.

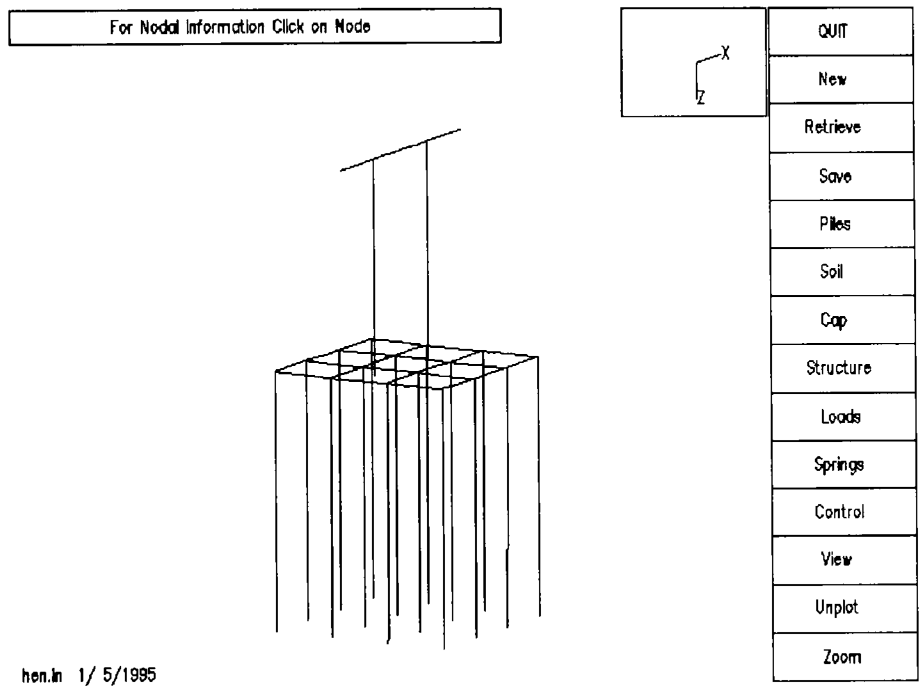

FIGURE 8 LPGPLOT initial view.

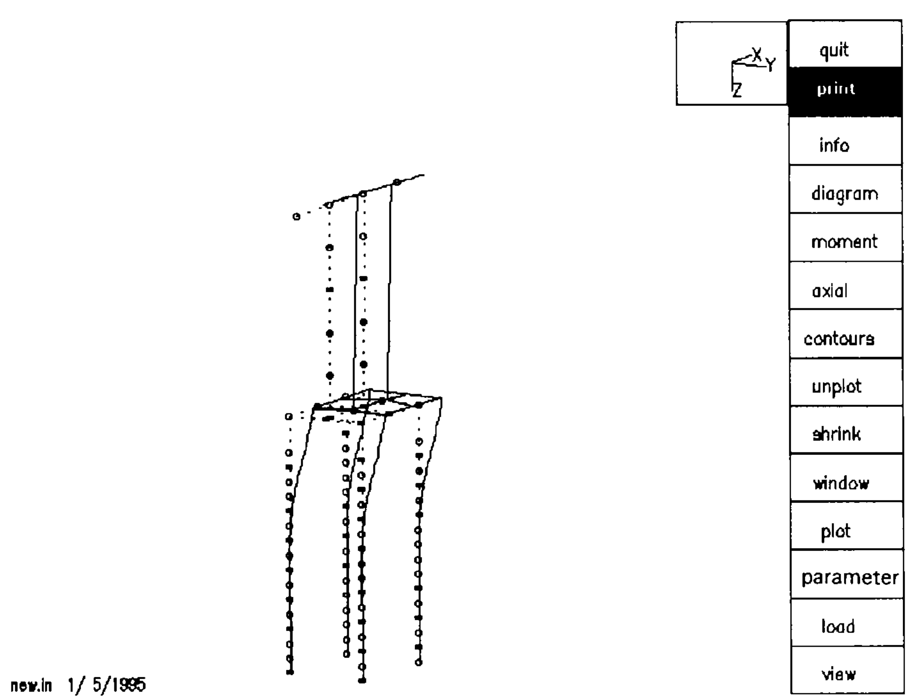

FIGURE 9 Deflected shape in LPGPLOT.

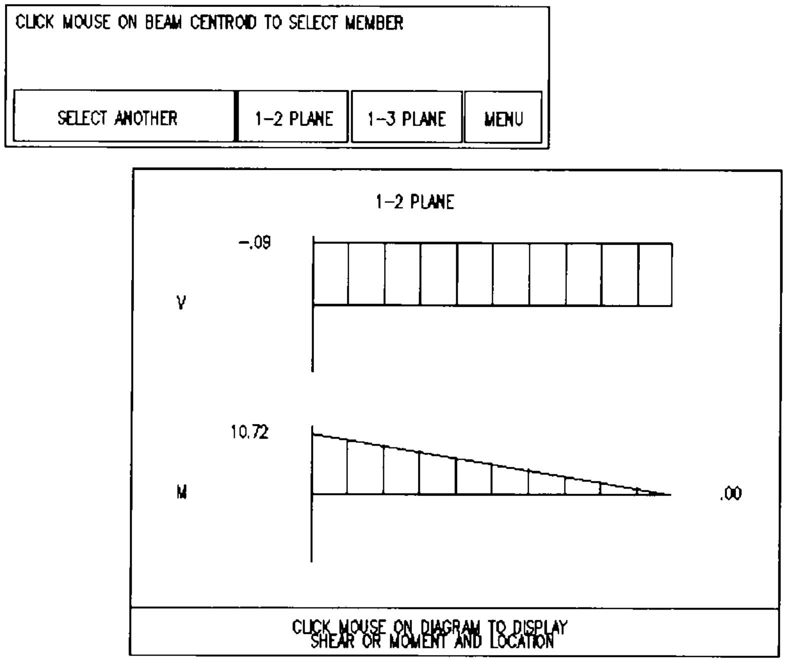

FIGURE 10 Moment diagram in LPGPLOT.

pier columns, and cap) or can give the shear and moment diagrams for individual elements, as shown in Figure 10.

Conclusions

The LPGSTAN bridge pier analysis package is an easy-to-use yet powerful analysis tool for engineers designing bridge pier foundation structures. Design parameters including the pile configuration and properties are changed directly, yielding fast iterations through the design process. The analysis is a highly accurate model of the structural foundation system and the soil interaction. The program has been verified by centrifuge studies and full-scale tests. The entire package runs efficiently on personal computers, making it accessible to all engineers working in bridge design.

Current extensions to the package being worked on are the ability to have automated or assisted pier column design and the automatic inclusion of AASHTO load conditions in the analysis. A grant from the Florida Department of Transportation involves the addition of concrete design check options for the pier columns and cap. The authors are also evaluating the proper design procedure for using AASHTO load combinations in preliminary design when using linear piles. Also being investigated is the use of nonlinear elements in the superstructure and when and how to switch to a complete design with nonlinear piles and superstructure.

Another needed extension is the complete coupling of the substructure to the superstructure (complete bridge). Although LPGSTAN handles the effects of bridge stiffness through the use of springs, this does not include the interaction between different piers. This coupling effect becomes even more important in dynamic analysis. Future extensions may also include multiple piers connected by a bridge span. The modeling of the bridge span can range from simple (a single beam) to a complete bridge model. In the long term use of a complete nonlinear finite-element bridge model connected to multiple piers with a time domain solution would give the most accurate dynamic analysis. However, for use as a design program this is not feasible because of the extremely large computational demands. A critical software need is in the area of bridge modeling.

LPGGEN provides an excellent designer interface for rapid design iterations in combination with the quick analysis capabilities of LPGSTAN. No such software exists for bridge modeling. The closest software for complete bridge modeling is the BRUFEM (Bridge Rating Using the Finite Element Method) system (6). BRUFEM is an interactive three-dimensional modeling system for bridges. Although BRUFEM is comprehensive and user friendly, it is not graphics based like LPGGEN. In addition, the analysis is only linear and static.

LPGSTAN is an excellent starting point for the pier portion of any software to be used as complete integrated analysis solution. Extending LPGSTAN to include multiple piers and a dynamic solution is a first step in developing a complete dynamic solution bridge system. Interim solutions for better bridge modeling range from simple beam models to the use of a complete BRUFEM-type model. Future bridge modeling requires a graphics interface (in the nature of LPGGEN) and inclusion of nonlinear modeling.

REFERENCES

1. Design of Highway Bridges for Extreme Events. FHWA Workshop, San Francisco, Sept. 1994.
2. Hoit, M. I., M. C. McVay, C. O. Hays, and P. W. Andrade. Nonlinear Bridge Pier Analysis Using LPGSTAN. *ASCE Journal of Structural Engineering*, submitted for publication.
3. McVay, M. C., M. O'Brien, F. C. Townsend, D. G. Bloomquist, and J. A. Caliendo. *Numerical Analysis of Vertically Loaded Pile Groups*. ASCE Foundation Engineering Congress, Northwestern University, Chicago, July 1989, pp. 675–690.
4. Wang, S.-T., and R. C. Reese, *COM624P—Laterally Loaded Pile Analysis Program for the Microcomputer*. FHWA Report FHWA-SA-94-048. FHWA, U.S. Department of Transportation, 1993.
5. McVay, M. C., R. Casper, and T. Shang. Lateral Response of Three Row Groups in Loose to Dense Sands at 3D and 5D Pile Spacing. *ASCE Journal of Geotechnical Engineering*, submitted for publication.
6. Hays, C. O., M. I. Hoit, M. Selvappalam, M. Vinayagamoorthy, and G. R. Consolazio. *Bridge Rating of Girder-Slab Bridges with Automated Finite Element Modeling Techniques*. Structures and Materials, Research Report 90-3. Engineering and Industrial Experiment Station, University of Florida, Gainesville, Oct. 1990.