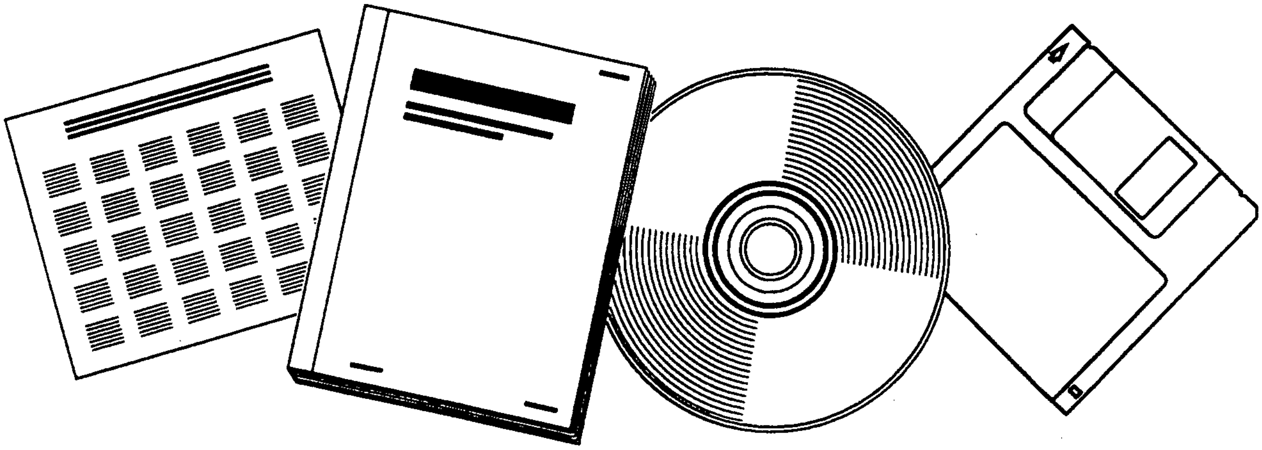
PB98-115033

DEVELOPMENT OF FINITE ELEMENT MODELS TO PREDICT DYNAMIC BRIDGE RESPONSE

OCT 97

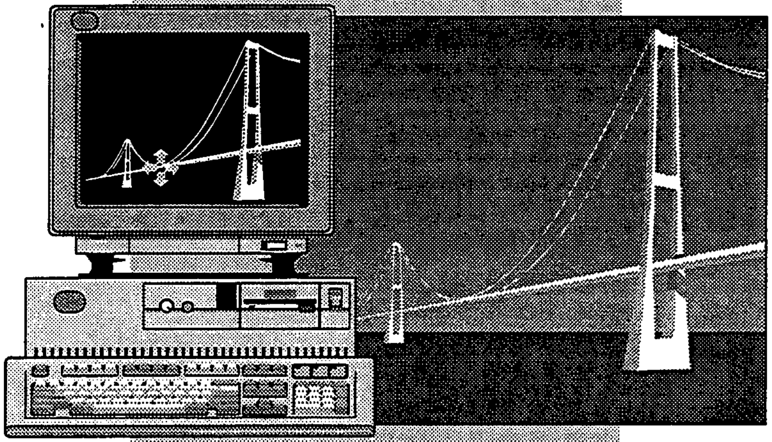


U.S. DEPARTMENT OF COMMERCE
National Technical Information Service

FINAL REPORT

PB98-115033

DEVELOPMENT OF FINITE ELEMENT MODELS TO PREDICT DYNAMIC BRIDGE RESPONSE



JOHN B. BAREFOOT
Graduate Research Assistant

FURMAN W. BARTON, Ph.D.
Faculty Research Scientist

THOMAS T. BABER, Ph.D.
Faculty Research Scientist

WALLACE T. McKEEL, Jr.
Research Manager

V·I·R·G·I·N·I·A
VTRC
TRANSPORTATION RESEARCH COUNCIL

VIRGINIA TRANSPORTATION RESEARCH COUNCIL

Standard Title Page - Report on Federally Funded Project

1. Report No.
FHWA/VTRC 98-R8

2. Government Accession No.

3. Recipient's Catalog No.

4. Title and Subtitle
Development of Finite Element Models to Predict Dynamic Bridge Response

5. Report Date
October 1997

6. Performing Organization Code

7. Author(s)
John B. Barefoot, Furman W. Barton, Thomas T. Baber, and
Wallace T. McKeel, Jr.

8. Performing Organization Report No.
VTRC 98-R8

9. Performing Organization and Address

Virginia Transportation Research Council
530 Edgemont Road
Charlottesville, Virginia 22903

10. Work Unit No. (TRAIS)

11. Contract or Grant No.
0556-020-940

12. Sponsoring Agencies' Name and Address

Virginia Department of Transportation	FHWA
1401 E. Broad Street	1504 Santa Rosa Road
Richmond, VA 23219	Richmond, VA 23239

13. Type of Report and Period Covered
Final

14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

Dynamic response has long been recognized as one of the significant factors affecting the service life and safety of bridge structures. Even though considerable research, both analytical and experimental, has been devoted to dynamic bridge behavior, the identification and extent of the controlling parameters that govern dynamic response have still not been clearly identified. One major requirement of any research program designed to address these issues is a convenient, accurate, and reliable analysis methodology that will permit any bridge engineer to easily construct a computer model of a bridge structure that will predict dynamic response.

The primary objective of this investigation was to develop a convenient and reliable analysis methodology, specifically, a procedure for developing finite element bridge models that can accurately predicting the static and dynamic response of bridges. Much of the previous research concerned with evaluating the dynamic response of bridges required the development of individual finite element models. In these studies, it was observed that the commands and procedures used to define these models were remarkably similar, even for different bridges. Thus, this study focused on developing an interactive framework, which consisted of a software package using ANSYS 5.0, that would permit bridge engineers to easily model any steel girder bridge regardless of skew, number of girders, or number of spans. This report describes the development of this finite element framework, provides validation through comparison with field test data, and illustrates its application to a typical bridge.

17 Key Words finite element models, bridge models, dynamic response, field testing

18. Distribution Statement
No restrictions. This document is available to the public through NTIS, Springfield, VA 22161.

19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 20	22. Price

Form DOT F 1700.7 (8-72) Reproduction of completed page authorized

FINAL REPORT

DEVELOPMENT OF FINITE ELEMENT MODELS TO PREDICT DYNAMIC BRIDGE RESPONSE

John B. Barefoot
Graduate Research Assistant

Furman W. Barton, Ph.D.
Faculty Research Scientist

Thomas T. Baber, Ph.D.
Faculty Research Scientist

Wallace T. McKeel, Jr.
Research Manager

Virginia Transportation Research Council
(A Cooperative Organization Sponsored Jointly by the
Virginia Department of Transportation and
the University of Virginia)

In Cooperation with the U.S. Department of Transportation
Federal Highway Administration

Charlottesville Virginia

October 1997
VTRC 98-R8

ABSTRACT

Dynamic response has long been recognized as one of the significant factors affecting the service life and safety of bridge structures. Even though considerable research, both analytical and experimental, has been devoted to dynamic bridge behavior, the identification and extent of the controlling parameters that govern dynamic response have still not been clearly identified. A major requirement of any research program designed to address these issues is a convenient, accurate, and reliable analysis methodology that will permit any bridge engineer to easily construct a computer model of a bridge structure that will predict dynamic response.

The primary objective of this investigation was to develop a convenient and reliable analysis methodology, specifically, a procedure for developing finite element bridge models that can accurately predict the static and dynamic response of bridges. Much of the previous research concerned with evaluating the dynamic response of bridges required the development of individual finite element models. In these studies, the commands and procedures used to define these models were remarkably similar, even for different bridges. Thus, this study focused on developing an interactive framework, consisting of a software package using ANSYS 5.0, that would permit bridge engineers to easily model any steel girder bridge regardless of skew, number of girders, or number of spans. This report describes the development of this finite element framework, provides validation through comparison with field test data, and illustrates its application to a typical bridge.

FINAL REPORT

DEVELOPMENT OF FINITE ELEMENT MODELS TO PREDICT DYNAMIC BRIDGE RESPONSE

John B. Barefoot
Graduate Research Assistant

Furman W. Barton, Ph.D.
Faculty Research Scientist

Thomas T. Baber, Ph.D.
Faculty Research Scientist

Wallace T. McKeel, Jr.
Research Manager

INTRODUCTION

The goal of bridge engineers has always been to design economical structures that are safe, serviceable, and durable. Accomplishing this task requires an understanding of the overall behavior of bridge structures and a knowledge of parameters affecting their response to loads. These structures have long been designed with the primary objective of avoiding failure under static loads. This aspect of safety is adequately addressed in the initial design by adherence to appropriate design specifications such as those of the American Association of State Highway and Transportation Officials (AASHTO). In most cases, the response of bridge structures to static loadings can be determined quite satisfactorily by any of a number of approximate analysis techniques, including relatively simple finite element models (Robson 1993). It is becoming increasingly obvious, however, that dynamic response is not as easy to predict and yet can play a major role in bridge response (Cantieni 1992). In fact, dynamic response may be the major factor that influences long-term behavior (Wang et al.1992, Wolek 1992).

Under current design practice, dynamic effects are taken into account by increasing the static load by an impact factor, *I*, which is a function only of span length. This results in attractive, sleek bridge designs that may satisfy safety and strength requirements but may have undesirable dynamic response characteristics and, consequently, may suffer distress as a result of unexpected response. Stresses developed with heavy vehicles moving at high speeds over a rough bridge deck are much greater than those obtained by incrementing static live loads by dynamic allowance factors prescribed in bridge codes (Inbanathan & Wieland 1987). Good design requires the ability to accurately predict the field response of the final structure to all types of loading, both static and dynamic. Existing analysis and design procedures do not always predict these unexpected and undesirable responses.

Although the importance of dynamic response with regard to the service behavior, life, and safety of bridge structures has long been recognized and has been the focus of considerable research, current analysis procedures are not always able to accurately predict dynamic response. Recent studies relating to the dynamic response of bridges have been performed to characterize the response of a structure based on the forcing function (vehicle-bridge interaction) and its basic parameters, such as length. Investigators at Purdue (Gaunt & Sutton 1981) and at the Road Research Laboratory (Leonard 1974) studied the vehicle-bridge system in the 1960s to determine which parameters of the vibration response have the most effect in decreasing user discomfort. Vehicle parameters such as wheel spacing, weight, and speed were considered along with construction parameters, such as roadway roughness. Recognized bridge parameters including span length and general girder stiffness were also studied. Numerous additional studies have been performed to validate the current impact factors through consideration of both dynamic and static field records. In one study (Bakht & Pinjarkar 1989), the different ways of calculating the experimental impact factor were evaluated and compared. In a separate study, Nowak (1990) also noted that dynamic properties of the vehicle, road roughness, and dynamic properties of the bridge influence the impact factor or dynamic load factor.

PROBLEM STATEMENT

In spite of the recognition of the importance and role of dynamic response in the deterioration and fatigue damage of bridges, there is clear evidence that current design practice can still result in bridges with unacceptable dynamic response in the form of large displacements. These unanticipated large displacements in several recently constructed long-span composite bridges may well affect their long-term performance because of increased stresses in the decks and parapets and fatigue in the girders. Resulting fatigue deterioration can override any cost-efficiency of these bridges because of the increased cost of maintenance and rehabilitation. Because these problems are not isolated, it is essential that bridge engineers focus their attention on these problems and determine the cause and solution to mitigate the negative effect to the maximum extent possible.

Experimental studies (Wolek 1992) have indicated that present methods of analysis underestimate impact and, thus, the dynamic effect for many structures. Since so many variables play a significant role in the generation and control of the total response, it is essential that bridge designers develop a better understanding of the dynamic behavior of bridge structures and the physical characteristics and parameters that affect response. A well-designed and carefully planned research program that focuses on identifying those design and geometric parameters that are most significant in affecting dynamic response should offer significant potential for improved dynamic behavior of new designs and enhanced behavior of rehabilitated structures through reduction of damaging motion. A major component of any such research program must be the availability of a reliable and convenient analysis methodology that will permit the engineer to determine the static or dynamic response of this structure to any type of loading. A convenient method for constructing an accurate and reliable finite element model of a variety of bridge structures would be of significant benefit for bridge designers.

PURPOSE AND SCOPE

This investigation was one phase of a broad, long-term study of the dynamic response of highway bridges conducted at the Virginia Transportation Research Council (VTRC) in cooperation with the University of Virginia. The long-term objective of the study was to develop a better understanding of the dynamic response of highway bridges and those factors that contribute to unacceptable dynamic response that, in turn, may contribute to a shortened bridge life. The objective of this phase of the study was to develop a convenient and reliable analysis methodology for predicting the static and dynamic response of bridges. In previous studies, it was observed that the input commands required to define finite element bridge models are remarkably similar from one structure to another. Consequently, a major goal of this study was to develop a methodology for interactively constructing a finite element model that was reliable yet remained general in nature.

Steel girder bridges built within the last 10 years tend to be more flexible than concrete bridges of similar age and, thus, were chosen as a focal point of this study in terms of both modeling and field tests. In the past few years, certain steel girder bridges in Virginia have had an unexpected response to normal vehicular traffic, particularly to truck loading. The Route 265 bridge over the Dan River and the Route 58 bridge over the Meherrin River were studied at the University of Virginia because of their unfavorable dynamic response and were selected for inclusion in this study.

METHODOLOGY

Developing an interactive methodology for constructing a reliable finite element model of any bridge with slab-girder construction involved a number of phases.

1. *To accommodate the requirement for dynamic analysis and, thus, sequential analysis steps, identify a finite element code that was capable of at least a modest level of programming within the code to permit iterative analysis steps.* After examining a number of commercial codes, we decided to adopt ANSYS 5.0 as the basic code to be used in the development of the bridge analysis program. This software package permits the input of parameters to define bridge geometry and loading while retaining the vast capability of a commercial code for both static and dynamic analysis. In addition, the University of Virginia has a site license for the code that facilitated its use.

2. *Identify those bridge parameters that were the key features of a slab girder bridge, and determine how to best represent these features in a finite element model.*

3. *Validate the model.* This was accomplished by comparing response data predicted by the finite element model with corresponding response information calculated using other procedures and with experimental response data obtained in field tests.

MODELING ISSUES

The development of the finite element model, which will be used in a large-scale computer code to predict the response of structures, is one of the most critical components in any analysis process. For the model to accurately predict the response of the structure, it must represent the physical nature of the actual structure. Issues such as types of elements, size of the model, degrees of freedom, mesh refinement, specification of boundary conditions, and contributions of secondary components (drainage, curbs, voids, etc.) must be given careful consideration. Once a method of representing the different components of the structure has been selected, it is still necessary to define the model within the constraints of the finite element code, which may not be a trivial task.

Several questions need to be considered when initiating the modeling of a particular bridge structure. Should the girders be modeled using plate elements or simple beam elements? If beam elements will not be adequate, what types of plate elements should be used to represent the plate girders? How should the bridge deck be connected to the girders to best represent composite action; i.e., should rigid links be used to couple the slab and girder nodes or should some type of coordinate offset be employed? How should the boundary conditions be defined for end conditions such as simple supports or built-in ends? How are the supports and connections between slab and girder best represented for an integral backwall bridge? What elements of the model are the most sensitive in predicting response? Should parapets, diaphragms, or curbs be included in the model? Only proper consideration of these factors and their proper inclusion in the model will permit the bridge model to reliably predict response.

Representation of loading, which can take a number of forms and which includes dead load and live load, is another important consideration in model development. With ANSYS, the incorporation of dead load, represented as a weight per unit volume, is relatively easy to achieve by the use of a few simple commands. In the case of live loads, there are several similar methods of introducing loading in a finite element model. For example, a wheel load could be represented as a pressure, i.e., a distributed load over a region of the deck, or by a series of equivalent concentrated loads at node points. Although both might yield apparently similar results, one is usually a better representation of the actual loading condition than the other. Dynamic loading, whether in the form of inertia loads in free vibration or from moving vehicles, is a much more difficult problem. Even after carefully choosing parameters and modeling techniques, it is commonly necessary to calibrate the model to obtain results that correlate to those obtained from field testing.

Based on previous computer modeling of bridge structures, we knew that the basic commands required to define different bridge models in a particular code are remarkably similar. For this reason, we decided to develop this interactive modeling program within the commercial finite element code ANSYS 5.0. This interactive program can model any contemporary steel girder bridge regardless of skew, number of spans, or number of girder lines. Using modeling techniques valid for all bridges of this classification, this program allows the creation of quick and accurate models that require a minimum amount of "tweaking" and few adjustments. The interactive modeling program is simple to use, and the actual model definition requires the user only to answer a series of questions. The answers can be found quickly in any set of bridge plans, and a detailed bridge model can be assembled and placed into code in less than half an hour.

MODELING TECHNIQUES

Structure Representation

There are several basic methods of developing a finite element model of a bridge superstructure that consists of a concrete slab on a series of steel girders. One method involves using two layers of nodes; the first layer defines plate elements to represent the deck, and the second layer, below the first, defines beam elements to represent the girders. A second method represents the entire bridge in one layer of nodes by treating the girders and deck as a composite beam and using only beam elements to model the bridge. A third method, known as the grillage method, treats the beams and deck as separate elements (beam elements and shell elements) but in the same nodal layer. A combination of the second and third method was developed by the University of Colorado and used in a recent Federal Highway Administration (FHWA) study (Sawyer 1994).

In general, all of these models give reasonable answers for the first mode of vibration but differ significantly in their ability to predict subsequent modes. Also, the ability to predict stresses in various structural elements is affected by the choice of model. In this study, we selected the model consisting of two layers of nodes in which the top layer defined the plate elements representing the deck and the second lower layer defined beam elements representing the girders. Two additional rows of nodes were added above the edges of the deck to represent the parapets. This required a means of connecting the different sets of nodes to satisfy requirements for the global stiffness and mass matrices and interelement compatibility.

Several other attributes were added in an attempt to increase the reliability and adaptability of the model. The program can model several continuous spans with the assumption that the middle supports are represented as pins or rollers. Supports at the ends of the continuous span can be fixed, pinned, or partially pinned (roller). The definition of the model can be changed and noncomposite girder action may be modeled, simplifying the analysis of a bridge under dead or construction loading.

The representation of composite action, critical in bridge modeling (Casas 1995), presented the first challenge in the construction of the finite element model. To determine the best alternative to connect the concrete deck (represented by plate or shell elements) to the girders and parapets (represented as rows of beam elements), several modeling techniques were studied. After considering these techniques and evaluating response results, we determined that the use of rigid beam elements to connect adjacent nodes in the deck and beam representation were best able to model a fully composite slab and girder configuration. An option to reduce the rigid beam stiffness to nearly zero is available in the model if it is desired to represent noncomposite action. This option, together with the reduction of the concrete modulus, is useful in calculating bridge deflections under construction loads.

Girder properties often change along the length of the bridge. How to represent a variable depth of web along a haunched girder or changes in the flange and web thickness along girders of constant depths is another consideration in the creation of the model. Three modeling techniques shown in Figure 1 were investigated to represent girders with varied cross sectional properties. The first consisted of using only one cross section per span whose properties were calculated by taking an average of the properties of several cross sections over the length of a span. A second technique employed distinct piecewise uniform cross sections to model the change in the cross section throughout the span with the location of the neutral axis fixed. The third procedure used different neutral axis locations for each section, which increased the complexity of the modeling. In all procedures, the mass was the same, with the only difference being a slight increase or decrease in moment of inertia. The second procedure was judged to best represent changes in the girder geometry with the moments of inertia of each calculated about their neutral axis.

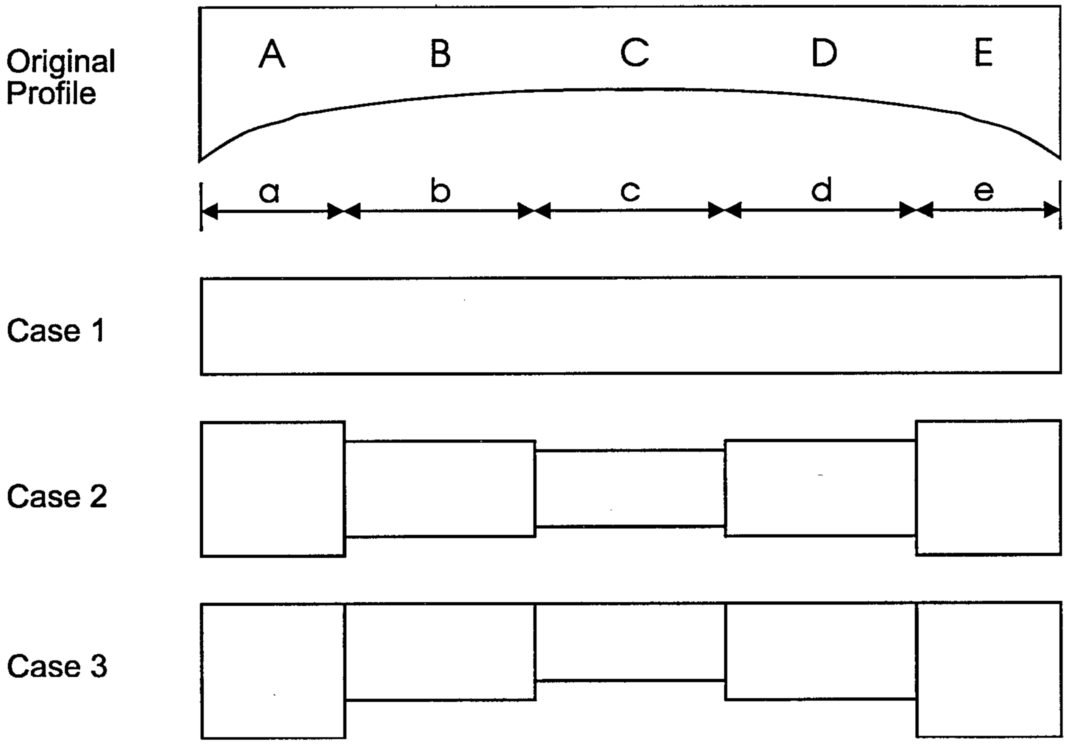


Figure 1. Three Modeling Cases for Girders with Varied X-Sections

The effect of diaphragms and cross bracing on the predicted dynamic response of a computer model is not readily apparent. The influence of these components depends not only on their configuration and orientation but also on the particular response component of interest. For example, diaphragms are not a factor in symmetric response such as longitudinal bending modes but may have a significant influence on transverse bending modes and torsional modes. Since the diaphragms and cross bracing act in an axial, flexural, and torsional capacity, beam elements were chosen to represent these components. Both *X* and *K* configurations of cross bracing were included. The bending stiffness of each of these configurations was determined, based on simple beam response, and an equivalent moment of inertia and area of these diaphragm elements were calculated based on these stiffnesses. The modeler is given a choice of these two types of cross bracing in the interactive modeling program and is prompted for the geometry of the members and must input the area, depth, width, and moments of inertia for the diaphragm. To determine spacing for modeling purposes, the number of cross supports specified in the plans can be divided into the length, assuming the supports are evenly spaced.

Parapets may also be significant components in a bridge model since they add considerable mass but relatively little stiffness. To calculate the area and moment of inertia of a parapet cross section, a simplified cross section was developed from the detailed cross section commonly specified in bridge plans. The area is the only important property because it is used in calculating the amount of mass the parapets will add to the structure. The moment of inertia will be calculated from the simplified cross section but will be arbitrarily decreased by a factor of about 25 percent to account for longitudinal discontinuity. The interactive program requests a percentage value so the value can be increased for instances where the parapets are integral with the deck along the length of the bridge. The option to leave out all effects of the parapets is also available.

The modeling program will automatically create supports at the end of each span and assume that all interior supports are pinned. It will ask if the end supports are fixed or pinned. Fixed supports affect both the beam and slab elements, whereas only the beam is affected when a pinned support is specified. The constraints of each pinned support must later be specified as rollers (*y* and *z* constrained) or fully pinned (*x, y,* and *z* constrained). The program cannot model rotation constraints or symmetry constraints, but both can be added within the input batch file. Because roller supports are the default condition, at least one support should be specified as fixed or fully pinned.

Loading Representation

The modeling program can analyze a structure under four loading conditions: transient, modal, static, and harmonic. The routine will ask the user to choose one of these analyses and begin prompting for all necessary information. To compare the finite element model results to those obtained in field testing, it was necessary to model each loading as accurately as possible.

Although the procedures for applying any of the four loading conditions are straightforward, special techniques are employed for representing a moving load or vehicle. This method uses the transient solution analysis of the ANSYS 5.0 code and specifies the time increment to travel from one set of nodes to the next sequentially along the length of the bridge. The program steps across the bridge, placing the loads on one specified set of nodes at a time while removing the load on the previously loaded set of nodes. The dimensions of the vehicle crossing the bridge define the specified nodes, and the time steps are calculated from the length of the bridge, speed of the vehicle, and definition of the model.

This routine can vary the number of axles, width of vehicles, and axle spacing. This makes it possible to represent any vehicle loading, including AASHTO truck loading. The model's solution routine requires the input of the speed of the vehicle, dynamic state of the vehicle, and centerline location of the vehicle's travel path. The program divides the axle weights in half and applies two wheel loads at a distance equal to half of the vehicle width onto either side of the centerline of vehicle travel. If the wheel spacing in either length or width is less than the spacing of the plate elements, the sum of the front and rear wheel load or the full axle load will be placed on the closest node. Master degrees of freedom in the *y* direction are automatically chosen at slab nodes loaded with wheel loads because the program requires that the loads in a transient analysis be placed on master nodes for a reduced analysis option. Several girder nodes are also selected as *y*-displacement master degrees of freedom. Solutions may be obtained only from master degree of freedom nodes.

An experimental loading case was added to the program that enables a harmonic load to be added to the moving load. The increased load is calculated as a user-chosen percentage of the moving load. The user must input the forcing frequency of the harmonic load. This allows a resonant harmonic loading condition if the fundamental bending frequency has been calculated in a modal analysis and input as the forcing frequency. The routine calculates the time to cross the bridge at the specified speed and, based on the input forcing frequency, determines the number of cycles that will be experienced. This loading is intended to approximate the oscillation of vehicles as they traverse a bridge.

Static loads may be placed on the model at an unlimited number of user-specified nodes. Only concentrated forces may be applied when using the solution input module. The location is defined by specifying percentages of the length and width of the selected bridge span. For example, a midspan load would be at 50 percent of the length and 50 percent of the width. Gravity loads are optional but useful for calculating dead load deflections. Harmonic loading is made possible by placing loads in a manner similar to that for the static loading. The forcing frequency of the load must be specified by a frequency range.

Modal analysis is the final analysis procedure employed in the program and is used to calculate the natural frequencies and corresponding modes of vibration for the structure. For this analysis, master degrees of freedom must be specified. The default setting selects every third node of the girders as a master node for the *Y*-displacement degree of freedom. The alternative selection process selects the user-specified number of master nodes for computer selected degrees of freedom using the ANSYS TOTAL command. The output will be available only for the number of modes selected to expand.

If the chosen solution type is STATIC, the routine will ask upon which span to place a concentrated load and the position of that load on the span. The position is determined as a percentage of the length and a percentage of the width of the particular span of interest. If MODAL is the chosen solution type, the routine will ask the number of modes to expand and how the user wishes to specify the master degrees of freedom. If TRANSIENT is the chosen solution type, the routine will ask for the weight of the vehicle axles, the speed of the vehicle, and the vehicle's wheel spacing. If HARMONIC is the chosen solution type, the routine will ask for the magnitude and frequency of the forcing function. Like the static solution, the harmonic solution will ask for a span and location to place the harmonic forcing function.

DEVELOPMENT OF BRIDGE MODELING SOFTWARE

The bridge modeling program was developed at VTRC using the ANSYS 5.0 finite element code package to allow easy development of simple bridge models for static and dynamic loading. The program is an ASCII batch file written in ADPL language similar to FORTRAN or C++ but capable of being input directly into the ANSYS code. Loops and if-statements are used to ask the user for the needed information and to create a full finite element model of any bridge structure. A copy of the program may be obtained from W. T. McKeel, Jr., Research Manager, Virginia Transportation Research Council, 530 Edgemont Road, Charlottesville, VA 22903.

The bridge modeling program allows easy modeling of bridges consisting of steel I-section girders and concrete deck slabs. It allows a bridge of almost any geometry to be input through a series of questions for which the answers are readily attainable from any set of bridge plans. The program can handle skewed bridges with as many/few girders as needed at any spacing. To adjust the program's default parameters, the user must edit the input batch file. The file, default, is the input batch file in ASCII format and can be edited by any UNIX-based editor, such as Jove, vi, or upenove. For example, to use the vi editor, one would type "edit default" at any UNIX prompt. Once the program file is loaded, several parameters can be changed simply by editing the variables at the end of the *ASK command lines. Comments were inserted for easier navigation throughout the default file.

The program is organized in such a way that the bridge and modeling parameters can be entered interactively. The following are lists of the input parameters and a brief description of each.

Bridge Parameters

1. SANG _ Angle between abutment and normal to centerline of bridge [degrees]
2. NS _ Number of spans in structure
3. NG _ Total number of girders
4. GRS _ Distance from centerline to centerline of girders [feet]
5. FRG _ Distance from outer edge of deck to centerline of first girder [feet]
6. BRL _ Total bridge length [feet]
7. TS _ Effective thickness of slab [inch]

8. WC _ Weight of concrete in slab [pcf]

9. CRFS _ Spacing of cross framing [feet]

10. FC _ Compressive strength of concrete in slab [psi]

11. SPANR _ Individual span length [feet]

12. NOXC _ Number of different girder cross sections in individual span length

13. COMPACT _ Composite action option (0 = yes, 1 = no)

Girder Parameters

1. DW _ Depth of web [inches]

2. TW _ Thickness of web [inches]

3. TTF _ Thickness of top flange [inches]

4. TBF _ Thickness of bottom flange [inches]

5. WTF _ Width of top flange [inches]

6. WBF _ Width of bottom flange [inches]

7. L _ Length of x-section currently being entered [feet]

Definition Parameters

1. NNL _ Number of nodes along length of bridge [dimensionless]. *Recommendation:* Pick a value close to the bridge length divided by 2 or 3.

2. NNG _ Number of nodes between girders [dimensionless]. *Recommendation:* Use 2 or 3 depending on the size of the model.

3. NNI _ Number by which nodes will be incremented in generation [dimensionless]. The recommended value is given by the routine. The defaults may be changed by editing the program. Keep in mind that shell elements, such as those used for the slab, perform better when the aspect ratio is less than 5 to 1.

Torsional/Parapet Parameters

1. PARAPETE _ Percentage of bending (moment of inertia) effect [0-100%]
2. TORSSUP _ Type of torsional support (none, cross framing, diaphragm)
3. DTYPE _ Type of *x*-framing (*X*-framing, *K*-framing)
4. SAREA _ Area of slanted member of cross frame [inch]
5. HAREA _ Area of horizontal member of cross frame [inch]
6. DMIX _ Strong moment of inertia of diaphragm [inch]
7. DMIY _ Weak moment of inertia of diaphragm [inch]
8. DIAREA _ Area of diaphragm [inch]
9. DDEPTH _ Depth of diaphragm's web [inch]
10. DWIDTH _ Width of diaphragm's flange [inch]
11. PARAPET _ Parapet option (0 = no parapets, 1 = add parapets)
12. SKEW _ Cross frame orientation option (1 = perpindicular, 2 = w/skew)

The process of running the program, viewing the model, and obtaining results was simplified with several prepared batch files. A bridge engineer should be able to analyze a typical bridge and obtain the desired results by following the instructions below, although it may be necessary to consult the ANSYS user's manual for particular options.

- *From an ANSYS directory,* type **ansys.e** and press enter.
- *If using an x-terminal,* type **/show,x11** to use a color monitor.
- *If using an x-terminal,* type **/menu,on** to turn on the menu system.
- *Once in the ANSYS menu setup,* type **/input,default** and press enter.
- Answer all questions.
- *To plot elements on the screen,* type **/input,elements.**
- *To plot nodes on the screen,* type **/input,nodes.**

Once the elements or nodes are plotted, change the views using the following batch files:

- *To view the model in an isometric orientation,* type **/input,iso_view**.

- *To view the model in an oblique orientation,* type **/input,obl_view**.

- *To view the model from the right side,* type **/input,rt_view**.

- *To view the model from the top side,* type **/input,top_view**.

Once the model has been generated and loads have been specified, it is next necessary to issue a command for the solution of the problem:

- *To solve,* type **/input,solution**.

At this stage, the basic problem has been solved and the results, such as stresses and displacements, have been calculated and stored in various files within ANSYS. These results can be displayed and/or printed using commands that are available in the various post-processors. The program will select the proper post-processor after the solution process is finished, and standard results can be easily obtained using prepared batch files:

- *To get the typical static numerical output,* type **/input,srslt_d**.

- *To get the typical static graphical output,* type **/input,srslt_g**.

- *To get the typical transient numerical output,* type **/input,trslt_d**.

- *To get the typical transient graphical output,* type **/input,trslt_g**.

- *To get the typical modal graphical output,* type **/input,mrslt_g**.

The menu system in ANSYS is convenient and self-explanatory. With limited exploration into the various menus, the user can become easily and comfortably acquainted with commands somewhat more difficult to explain in the raw code.

The program could be easily converted to model concrete bridges. Various degrees of parapet contributions are allowed, and composite or non-composite action can be modeled. However, the program can not model negative skew angles or skew angles greater than 60 degrees. Concrete girder bridges cannot be modeled, and a bridge cannot have partially restrained degrees of freedom at the supports. The program includes the following assumptions: both transient analysis and modal analyses are reduced analysis, the bridge has typical AASHTO parapets, the neutral axis of the girders is along the centerline of the deepest cross section, and physical and geometric linearity exists. Program warnings are produced when input values are unacceptable to continue with the construction of the model. Some warnings may occur for input of unrealistic parameter values, for excessive model definition, or for extremely large bridges where nodes may be defined out of an allotted range. The most common warnings occur when elements are defined with poor length-to-width ratios or when elements are assigned unusually low stiffnesses.

Program Verification

Effective use of any finite element model requires confidence in the reliability and accuracy of the model; thus, appropriate validation of any model is essential. Validation of models may be achieved by comparing model results either with the measurement of actual bridge response as determined by field tests or with response values calculated from analytical approximations or from current design procedures.

A model can be quickly validated for dynamic comparison by simply comparing the natural frequencies of the actual and model bridges. Because many influencing factors are specific to one type of loading, this may not be the most appropriate indicator of validity if static deflections or girder strains are of primary interest. Therefore, different methods of validation may be required. The primary purpose of developing a model in this study was to predict modal and dynamic response and, consequently, validation of the model could be sufficiently accomplished by comparing model response with response determined from experimental natural frequency analysis, and closed form and approximate solutions of dynamic displacement. Nevertheless, comparisons with static response values were also included.

To validate the model using actual field test data, it was desirable to use at least two bridges having different span lengths and support conditions. Two Virginia bridges were selected and tested to provide data that could be used to calibrate and validate the finite element models of these structures. One of the bridges was located on Route 265 over the Dan River near Danville and consisted of two four-span continuous segments with each of the eight spans having a length of 37 m (120 ft). The second bridge tested was the Route 220 bridge over the James River about 8 km (5 mi) south of Clifton Forge. The structure has seven spans with the central portion consisting of a three-span continuous segment with individual span lengths of 55, 82, and 55 m (180, 270, and 180 ft). Using the program developed in this study, a finite element model was created to represent each of these bridges.

Additional validation was performed by comparing predicted response from the model with response determined from other sources, such as strength of materials equations, the VDOT girder design program, and other theoretical and analytic calculations. For this phase of validation, default finite element models, each consisting of a 31-m (100-ft) simply supported span, were created and used for comparison. These models consisted of only a single girder line that included a portion of the slab.

Static response comparisons were based on predicted response from the default bridge model in which the equivalent moment of inertia of the composite cross-section was assumed to be 23,900 E6 mm^4 (57,430 in^4) and the weight was assumed to be 18 N/mm (102.78 lb/in). The static live load analysis was performed by placing a 40,000 N (9,000 lb) concentrated force at midspan of the default model and using the static analysis mode within ANSYS. The maximum vertical deflection of 4.93 mm (0.194 in) at midspan was obtained from the model's output of nodal displacements. The corresponding deflection calculated from a strength of materials simple beam model was 4.95 mm (0.195 in) and from the VDOT girder design program was 4.6 mm (0.180 in). A dead load analysis was performed by applying a gravity acceleration to the default model, once with the effects of composite action and a second time without composite action. The maximum predicted displacements were 42.7 mm (1.68 in) and 117 mm (4.63 in), respectively. The strength of materials beam approximation for the same two cases indicated maximum displacements of 42.4 mm (1.67 in) and 107 mm (4.20 in), whereas the VDOT girder program predicted displacements of 42 mm (1.65 in) and 111 mm (4.37 in), respectively.

Next, a transient analysis was executed using a second default bridge model in which a constant force of 8,900 N (2,000 lb) was caused to move across the bridge at a constant velocity of 35 km/h (22 mph). For this problem, the moment of inertia was assumed to be 2,740 E6 mm^4 (6,580 in^4) and the weight was assumed to be 1.6 N/mm (109 lb/ft). The results of this analysis were compared with an exact, closed form solution derived in an earlier phase of the study (Massarelli & Baber, 1994) using a beam model of the bridge and Mathcad software. The maximum dynamic displacement at midspan, as determined from both the exact solution and the default model, was approximately 9 mm (0.36 in). The results from the computer model, as seen in Figure 2, compare very well to results calculated from the closed form solution.

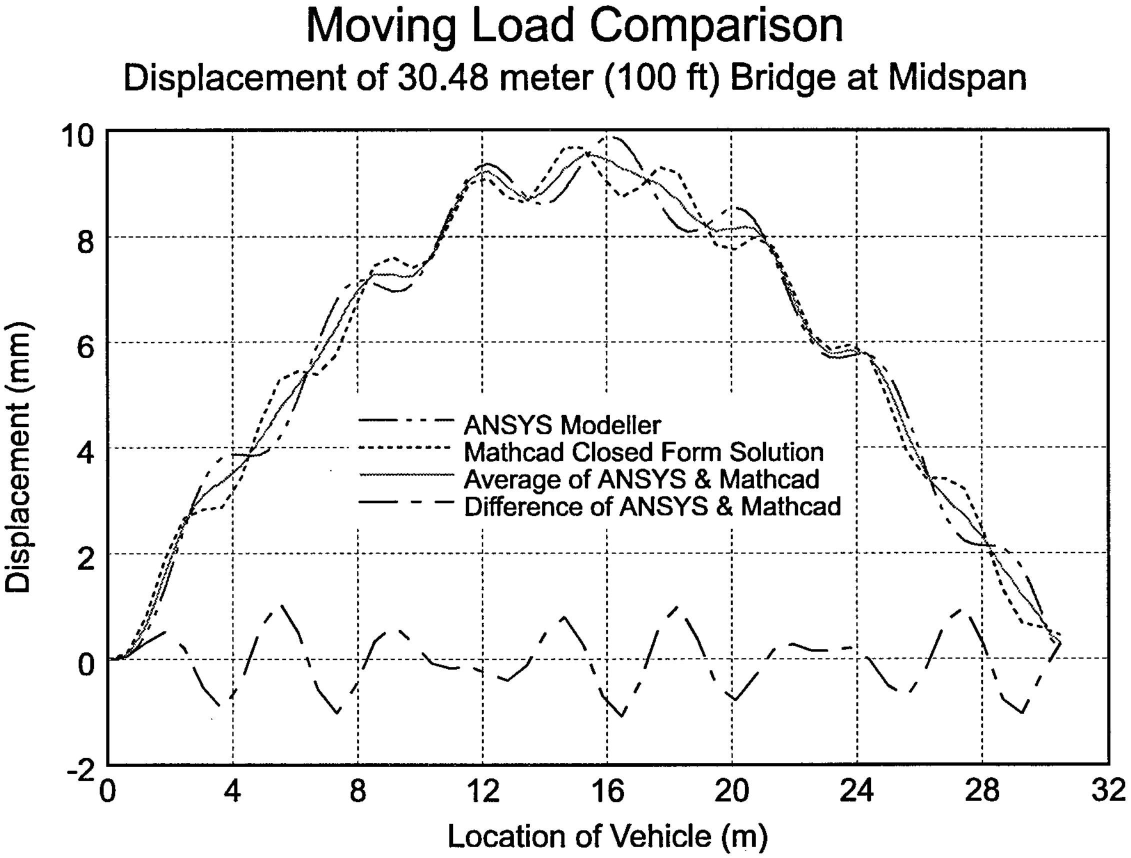


Figure 2. Comparison of SNSYS Moving Load Technique to Mathcad's Closed Form Solution

ANSYS models of the Dan River and Route 220 bridges were built using the interactive modeler described previously. Elevation views for both bridges are shown in Figures 3 and 4. Diaphragms and parapets were included in the development of these models, although the parapets were included with a reduced stiffness. These models were used primarily to validate the predicted dynamic response, although static results were also considered. A static dead load analysis was run on models of the Dan River and Route 220 bridges under construction loading where concrete strength and composite action were ignored. Deflected shapes were calculated and matched to those specified by the deflection profiles in the original plans. The maximum dead load displacements were calculated as 60.5 mm (2.38 in) and 151.6 mm (5.97 in) for the Dan River and the Route 220 bridges, respectively.

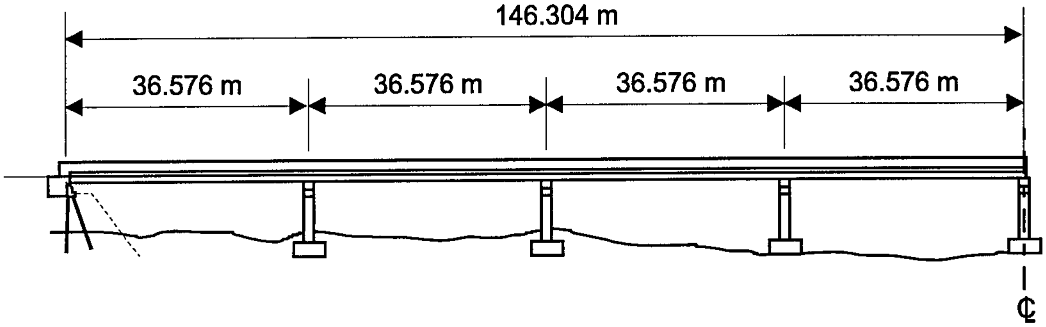


Figure 3. Elevation of Dan River Bridge

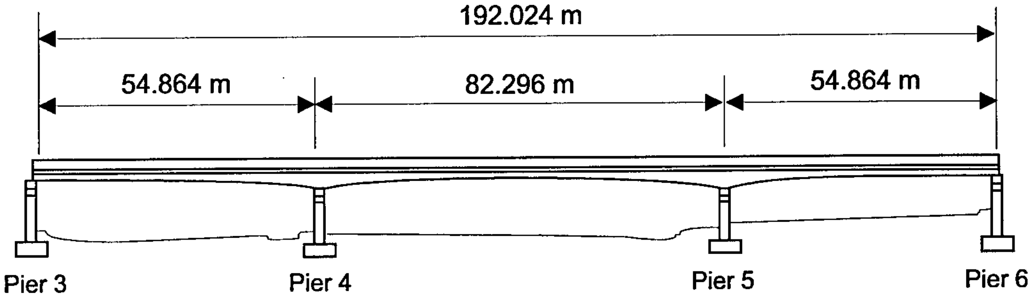


Figure 4. Elevation of Main Unit of Route 220 Bridge

An ANSYS modal analysis was performed on the models of the Dan River and Route 220 bridges to determine the fundamental frequencies and mode shapes of these structures. The computer models for these bridges were the same models used for evaluating behavior under static loadings and are shown in Figures 5 and 6. Because it was not possible to simulate actual traffic loadings in the dynamic analysis of the computer models, comparisons were made primarily on the basis of frequencies and mode shapes.

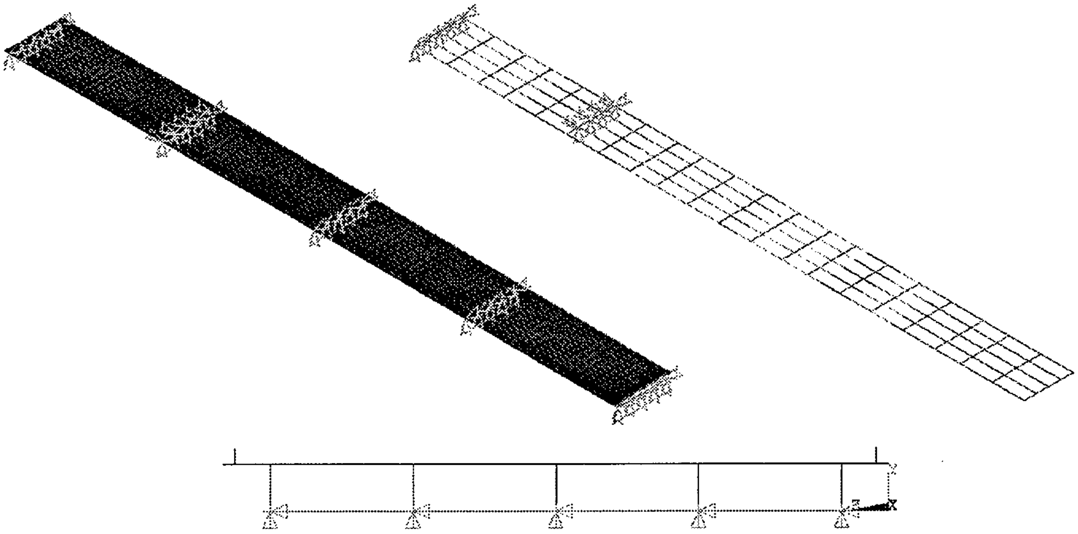


Figure 5. Model of Dan River Bridge with Framing Plan and X-Sectional View

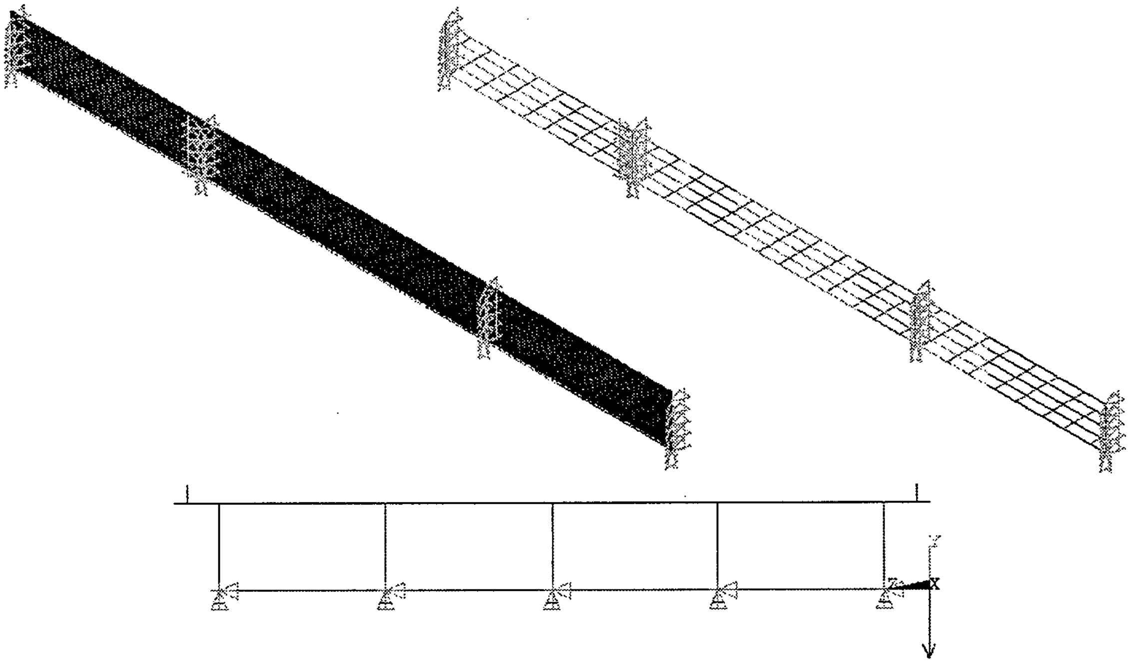


Figure 6. Model of Route 220 Bridge with Framing Plan and X-Sectional View

For the Dan River Bridge, the fundamental frequency was calculated to be 2.26 Hz, and the corresponding mode shape, which is the first bending mode, is shown in Figure 7. Based on

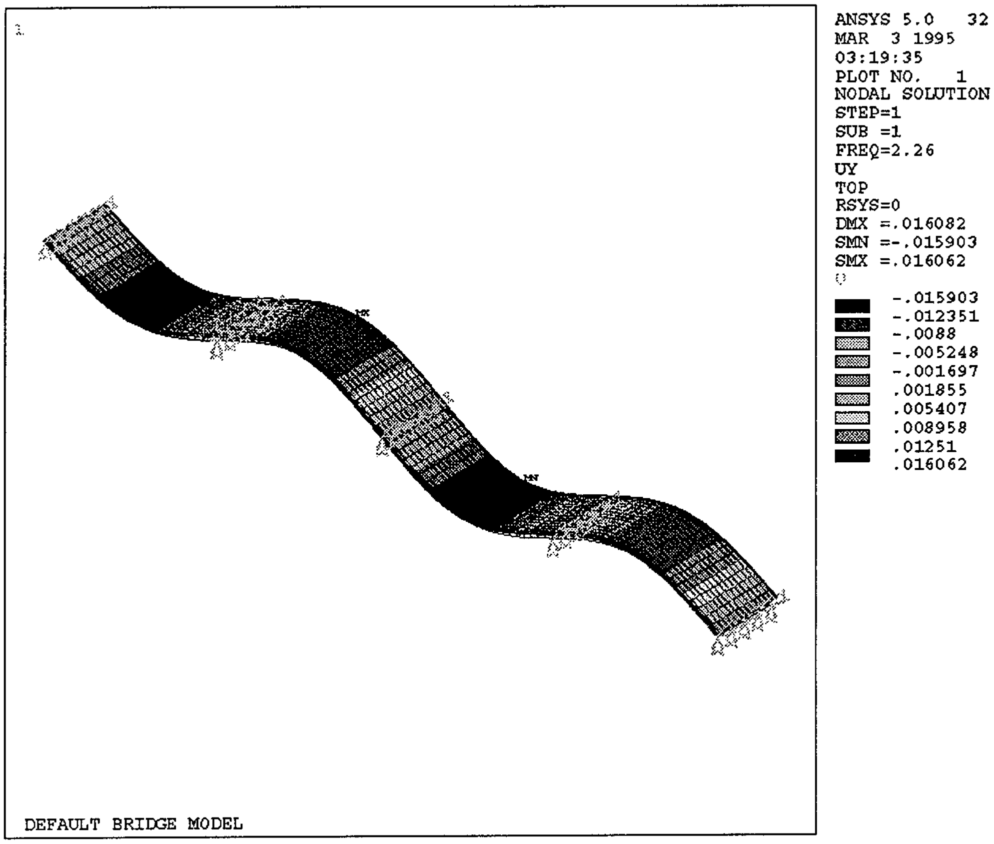


Figure 7. Fundamental Bending Frequency (2.26) of Dan River Bridge

the analysis of measured response from the field test, the experimental fundamental frequency was found to be 2.34 Hz, slightly higher than that predicted by the model. Frequencies and mode shapes for the first six modes of the Dan River Bridge model were calculated and compared with the frequencies measured during the field test. The frequency values are tabulated in Table 1, and the agreement between measured and predicted frequencies is excellent.

The computer model of the Route 220 bridge predicted a fundamental frequency of 1.39 Hz, whereas the frequency determined from the measured acceleration response was 1.40 Hz (see Figure 8). Frequencies and mode shapes for the first six modes were also calculated using this bridge model and compared with measured frequencies from the field tests. These frequency comparisons are presented in Table 2, and, again, the agreement is seen to be excellent.

Table 1. Natural Frequencies of the Dan River Bridge

Mode/Method	Results from Field (Hz)	ANSYS Modeler (Hz)
1	2.34	2.26
2	2.78	2.73
3	3.66	3.65
4	3.82	3.78
5	--	3.99
6	4.59	4.59

Table 2. Natural Frequencies of the Route 220 Bridge

Mode/Method	Results from Field (Hz)	ANSYS Modeler (Hz)
1	1.40	1.39
2	1.65	--
3	2.23	2.37
4	2.83	2.88
5	3.08	3.04
6	3.85	3.82

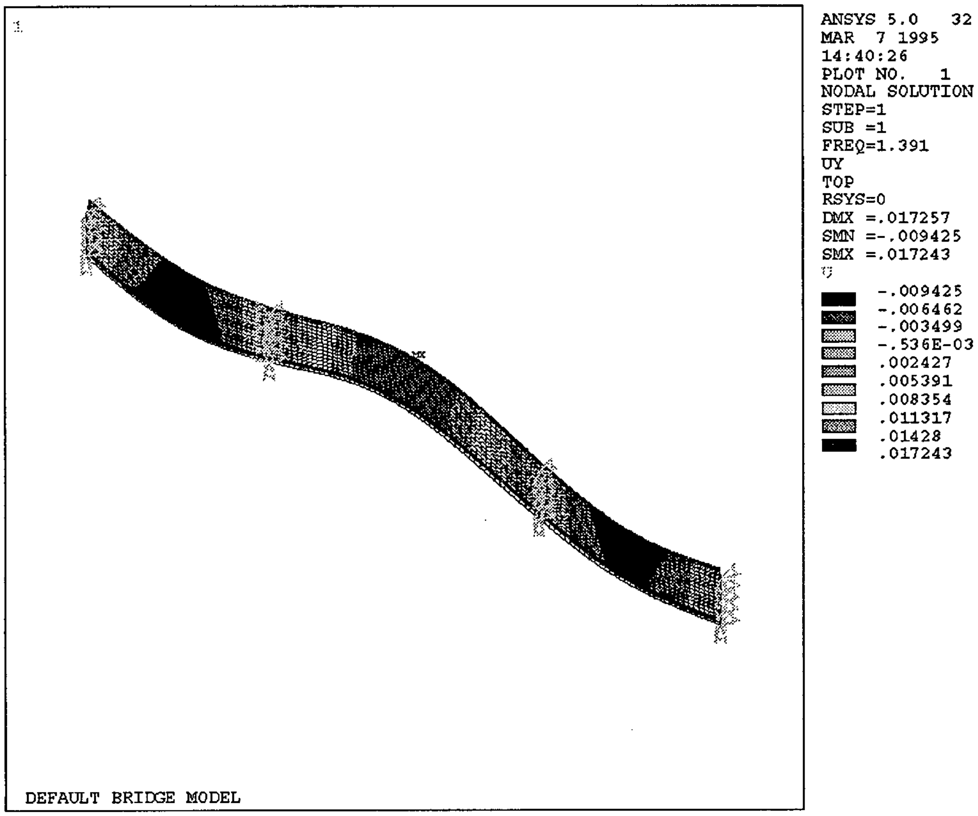


Figure 8. Fundamental Bending Frequency (1.39) of Route 220 Bridge

CONCLUSIONS

- Reliable and accurate finite element models of a variety of bridges can be readily developed using the software package developed in this study coupled with the ANSYS analysis code.
- Parapets and diaphragms should be included in the finite element models of bridges if reliable dynamic response, particularly torsional response, is desired.

- The predicted response from finite element models generated using the software developed in this study compares very favorably with the response obtained from other methods of analysis and from experimental data recorded during field tests.

RECOMMENDATIONS

- *Bridge engineers should employ the modeling capability developed through this study to predict the response of bridge structures in those cases where there might be a possibility of undesirable dynamic response.* Even for those bridges where large dynamic response is not a factor, the use of finite element models derived from the software provided through this study would seem to be an excellent way to confirm the behavior predicted by the design procedures.

- *Additional work should be undertaken to more accurately represent the effects of secondary elements on the dynamic response of bridges and to more reliably model various supports found in the field.* Such a follow-up study could also include simple procedures for modeling moving vehicle loads.

REFERENCES

Bakht, B. & Pinjarkar, S.G. (1989). Review of Dynamic Testing of Highway Bridges. *Transportation Research Record No. 1223.* Transportation Research Board, Washington, DC.

Cantieni, R. (1992). *Dynamic Behavior of Highway Bridges Under the Passage of Heavy Vehicles.* EMPA Report No. 220. EMPA, Dubendorf, Switzerland.

Casas, J.R. (1995). *Dynamic Modeling of Bridges: Some Thoughts Derived from Field Testing.* Preprint for the 74th Annual Meeting of the Transportation Research Board, Washington, DC.

Gaunt, J.T. & Sutton, C.D. (1981). *Highway Bridge Vibration Studies.* JHRP-81-11. Department of Civil Engineering, Purdue University, West Lafayette, IN.

Inbanathan, M.J. & Wielaand, M. (1987). Bridge Vibrations Due to Vehicles Moving Over Rough Surface. *Journal of Structural Engineering,* Vol. 113, No. 9. pp. 1,994-2,008.

Leonard, D.R. (1974). *Dynamic Tests on Highway Bridges: Test Procedures and Equipment.* TRRL LR 654. The Road Research Laboratory, Crowthorne, England.

Massarelli, P.J. & Baber, T.T. (1994). *Analytical Solution to Moving Load Problems.* VTRC 95-R6. Virginia Transportation Research Council, Charlottesville.

Nowak, A.S. (1990). *Bridge Evaluation, Repair and Rehabilitation.* NATO ASI Series E: Applied Sciences. Vol. 187. Kluwer Academic Publishers, Boston.

Robson, B.N. (1993). *Modeling of Field-Tested Highway Bridges.* Presented at the 72nd Annual Meeting of the Transportation Research Board, Washington, DC.

Sawyer, D.A. (1994). *Development of a Finite Element Model to Predict Maximum Strains in Steel I-Girder Bridges.* Master's thesis, University of Virginia, Charlottesville.

Wang, T., Huang, D. & Shahawy, M. (1992). Dynamic Response of Multigirder Bridges. *Journal of Structural Engineering,* Vol. 118, No. 8, pp. 2,222-2,238.

Wolek, A.L. (1992). *The Analytical and Experimental Evaluation of the Dynamic Behavior of Bridges.* Master's thesis, University of Virginia, Charlottesville.

Reproduced by NTIS

National Technical Information Service
Springfield, VA 22161

NTIS does not permit return of items for credit or refund. A replacement will be provided if an error is made in filling your order, if the item was received in damaged condition, or if the item is defective.

This report was printed specifically for your order from nearly 3 million titles available in our collection.

For economy and efficiency, NTIS does not maintain stock of its vast collection of technical reports. Rather, most documents are printed for each order. Documents that are not in electronic format are reproduced from master archival copies and are the best possible reproductions available. If you have any questions concerning this document or any order you have placed with NTIS, please call our Customer Service Department at (703) 605-6050.

About NTIS

NTIS collects scientific, technical, engineering, and business related information — then organizes, maintains, and disseminates that information in a variety of formats — from microfiche to online services. The NTIS collection of nearly 3 million titles includes reports describing research conducted or sponsored by federal agencies and their contractors; statistical and business information; U.S. military publications; multimedia/training products; computer software and electronic databases developed by federal agencies; training tools; and technical reports prepared by research organizations worldwide. Approximately 100,000 *new* titles are added and indexed into the NTIS collection annually.

For more information about NTIS products and services, call NTIS at 1-800-553-NTIS (6847) or (703) 605-6000 and request the free *NTIS Products Catalog,* PR-827LPG, or visit the NTIS Web site **http://www.ntis.gov**.

NTIS
Your indispensable resource for government-sponsored information—U.S. and worldwide

U.S. DEPARTMENT OF COMMERCE
Technology Administration
National Technical Information Service
Springfield, VA 22161 (703) 605-6000