Non Destructive Evaluation/Testing Methods – 3D Finite Element Modeling of Bridges

Divyachapan S. Padur, Xiaoyi Wang, Ahmet Türer , James A. Swanson, Arthur J. Helmicki, Victor J. Hunt

University of Cincinnati Infrastructure Institute
Cincinnati, OH 45221-0030 USA

ABSTRACT

3D Finite Element Modeling (FEM) is very useful in condition assessment of highway bridges and can help ratify actual field data obtained through truck tests or modal tests. Since manually generating or revising a typical model is time-consuming, the 3D FE models are generated using a preprocessor written in Visual Basic to make the entire process of modeling more efficient. The preprocessor, Bridge Modeler, has a Graphical User Interface (GUI) which lets users input meaningful data from bridge plans and generates the FE model. The model is then fed as input to SAP90™ (Structural Analysis Program) for analysis, generation of modes and generation of expected influence lines. What used to take weeks and days to write manually has been automated using this preprocessor to be done in a couple of hours. Several bridges with special features were analyzed by FEM. Bridge Modeler has proven to be an efficient tool for generating FE models. FE model calibration has been started. The initial comparisons between experimental and theoretical results have been completed.

INTRODUCTION

3D Finite Element Modeling (FEM) has the potential to significantly improve the design and analysis of highway structures, especially bridges. This method uses finely meshed elements to construct the model of a bridge in three dimensions, as in real life. Supports can be defined with springs, or models of supports (such as piers) can be generated and integrated into the model. The geometry of the bridge can be best modeled with 3D-FEM, getting closest to the representation of the actual structure. The stresses in members can be calculated directly by a well-defined geometrical model, which incorporates a variety of element types such as shell and frame elements. The only downside of this type modeling is the large size of the analysis problem and the difficulty in calibration of several thousands of variables. With ever-advancing computer technology and capacity, the 3D-FEM is becoming an easily affordable solution.

A bridge can be modeled in many different ways based on the level of modeling. Although three-dimensional (3D) analyses supply the best possible numerical simulation of a structure, they, too, require approximations because it is too costly to model every detail. For instance, the concrete deck is usually modeled by shell elements instead of solid elements. Several shell elements (with a longitudinal aspect ratio of 1.5) are used between girders to represent the deck.

Girders are defined by using a combination of shell and frame elements: webs being composed of shell elements and flanges made out of frame elements. Composite action between the deck and girders is simulated by rigid links (stiff frame elements) connecting the deck to girders. Three-dimensional FE modeling is a great step in minimizing the assumptions made by lower level analytical models. Computing member stresses directly in 3D-FEM, instead of extrapolating them from the bundled member forces and sectional properties as in 1D and 2D analysis methods, is a significant improvement. A figurative comparison of the three modeling techniques is shown in Figure I below.

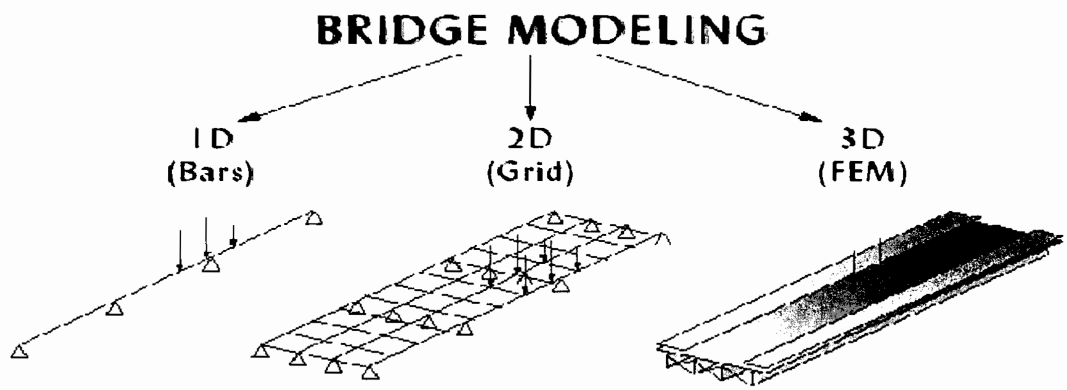

Fig I. Overview of Modeling Techniques.

The Bridge Modeler Software

Typically, model creation is the most time-consuming part bridge design and analysis since a large number of nodes and elements have to be generated to form bridge structure. Generating the model of a standard three span bridge of approximately 150 feet length could take days or weeks. After the generation comes the analysis of the model to provide the deformations and stresses that are needed. The modeling of a bridge goes through numerous iterations of model creation and analysis. Manually incorporating the changes in iterations is also time consuming making the entire process terribly lengthy. Any technology becomes irrelevant unless it can deliver results in an acceptable time-frame. To ensure that we get good results in a reasonable amount of time using the 3D FEM for modeling of bridges, the University of Cincinnati Infrastructure Institute (UCII) has developed a software GUI (Graphical User Interface) for creating the FE model called the "UCII Bridge Modeler".

This program is written in Microsoft Visual Basic™ and is primarily meant to simplify the process of creating a FE model of a bridge. It is designed in such a way that any layman can input the required data from the bridge plans to create a FE model of the bridge. The output from this software is the input file to SAP90™. SAP90™ is a widely used 3D FE model analysis program developed by CSI, Berkeley. The initial objective of this project was to automate the model creation process of the bridges involved in a six bridge project that UCII was contracted for. Currently, the modeler software is designed to handle steel stringer bridges only.

Assumptions

The 3D-FE Bridge Modeler program developed for modeling steel stringer bridges involves certain assumptions that are listed below.

1. The Bridge Modeler program is developed to prepare 3D FE models of straight concrete-deck-on-steel-stringer bridges.
2. The bridge modeler program generates input files for SAP90 program, which can also be loaded into SAP2000 software using its internal import command.
3. Although the properties of each girder can be modified independently, the girders are assumed to be equally spaced. For unevenly spaced girders, the bridge model generation can be carried out normally with the spacing modified manually. A variable girder spacing option will be added in future versions.
4. The girders are defined by using shell elements for the web, and frame elements for the flanges.
5. The deck is modeled using shell elements. Two layers of shell elements that are connected with rigid links are defined for the thicker portions of the deck, such as sidewalks.
6. The connections between the deck and the girders are simulated by using rigid links.
7. The supports are restrained for lateral movement. Vertical springs are defined to simulate the flexibility at the supports.
8. Moving loads (i.e., HS20-44 truck and point loads) are generated and saved in separate files by the Bridge Modeler program. Moving loads are not initially included in the main input because at the calibration stage such information is not needed. After arriving at the final calibrated model, the user will copy and paste the files into the main input file for bridge rating purposes.

Even though 3D-FE models are the best tools to represent the structural properties and behavior of a bridge, 3D modeling and analysis still require certain simplifications and engineering judgment. The 3D-FE model of a bridge can be generated in a number of different ways based on the level of desired complexity and accuracy.

Bridge Modeler – Program Description and GUI

The Bridge Modeler software is designed for Windows NT/2000/9.x versions. It creates an output file as per the standards required for SAP90™, though the input file can be used in SAP2000 also. The program has an interface as shown in the Figure II below. As the Figure II shows, the Bridge Modeler program has a pleasant user-friendly interface that collects the input in the major categories shown in the tab sequences. A picture associated with each of the tabs is shown in the upper half of the interface to enhance the understanding of the model creation process. This way the user knows the effect of the input variables on the model immediately.

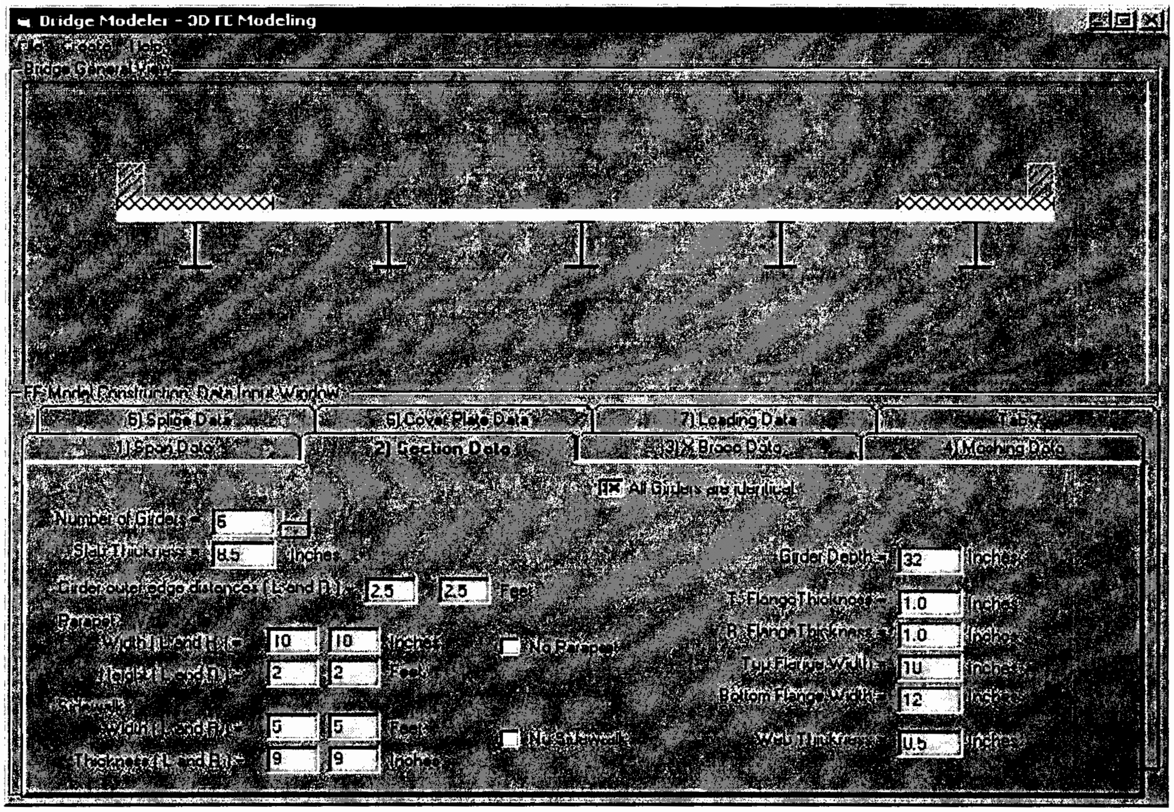

Fig II. GUI of Bridge Modeler Software.

It also helps to check for errors in input by simply verifying the picture for apparent anomalies. Appropriate units are indicated next to each input box to help the user determine the values that have to be entered. A conversion table is included in the Help section of the drop down menu for quick reference. Holding the mouse pointer over any section or input box will reveal a ToolTip™ text box with relevant information. Extensive help regarding the usage of this software is included in the Help menu. The software is also made robust enough to handle many errors in input. All variables are assigned default values and the user has to change only those that differ from the bridge plans. This feature is also helpful in having a successful run every time, just in case the user forgets to input a value in a particular input box. Also the input data of a bridge can be saved as a file that can be reloaded whenever needed.

Modeling Procedure and Results

The 3D FE modeling process using the UCII Bridge Modeling Software begins with the data collection process. The exact steps involved in arriving at plots of modes or UILs (Unit Influence Lines) is as follows:

1. Obtain the data from blue prints of the bridge to feed into the Bridge Modeler program
2. Create the SAP input file using the Bridge Modeler and feed it into the SAP90™ program

3. Obtain the output files, i.e. files with the extension "sol", "eig" etc. depending on the type of output expected.
4. Extract the necessary values from the files and use Microsoft Excel™ to plot and compare with test results.
5. The model may have to be revised by changing the input parameters to measured values instead of design values from the plans. For example, the deck thickness of the bridge may have been reduced due to erosion or there could be retrofits to the bridge that were not a part of the original bridge plan.
6. The nominal model then has to be tuned to match the model test results so that the model can be used for rating the bridge.
7. The rating process can include changes like partially composite action between deck and girders, adding springs to supports, etc.

The initial model of the bridge can be used to verify the modal test characteristics of the bridge. The first model can give an idea as to what to expect from the bridge in terms of frequency response and the initial results can greatly help understanding the relevance of impact points of a modal test. Several bridges have been modeled since inception and the program is constantly being modified to be able to handle special cases as and when they are encountered. Two examples of bridges modeled using this program is shown in Figures III and IV.

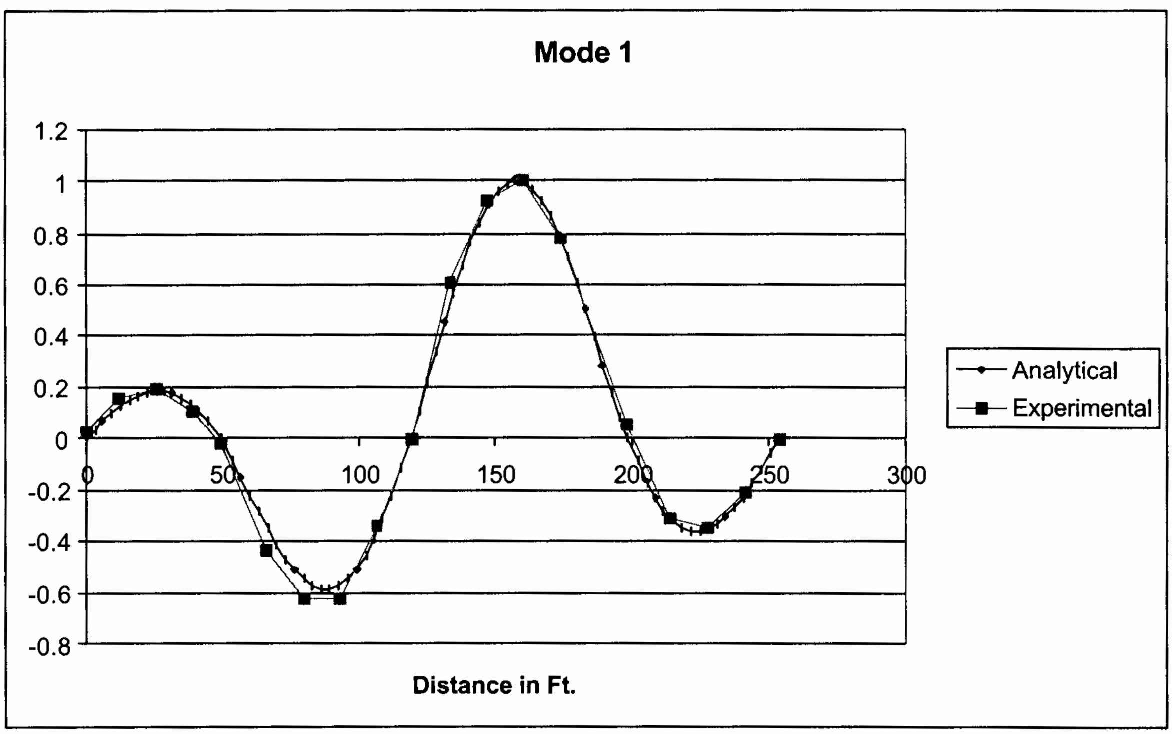

Fig. III – First Mode Shape of RIC-30-1384 Bridge from Simulation and Experimental data

The plot in Figure III is the first mode shape of the RIC-30-1384 Bridge from the simulation results and experimental results. The diagram in Figure IV is the comparison of actual response of the WAS339 Bridge to a super load and the reconstituted response of the bridge from the UILs obtained from the 3D FE model created by the Bridge Modeler software.

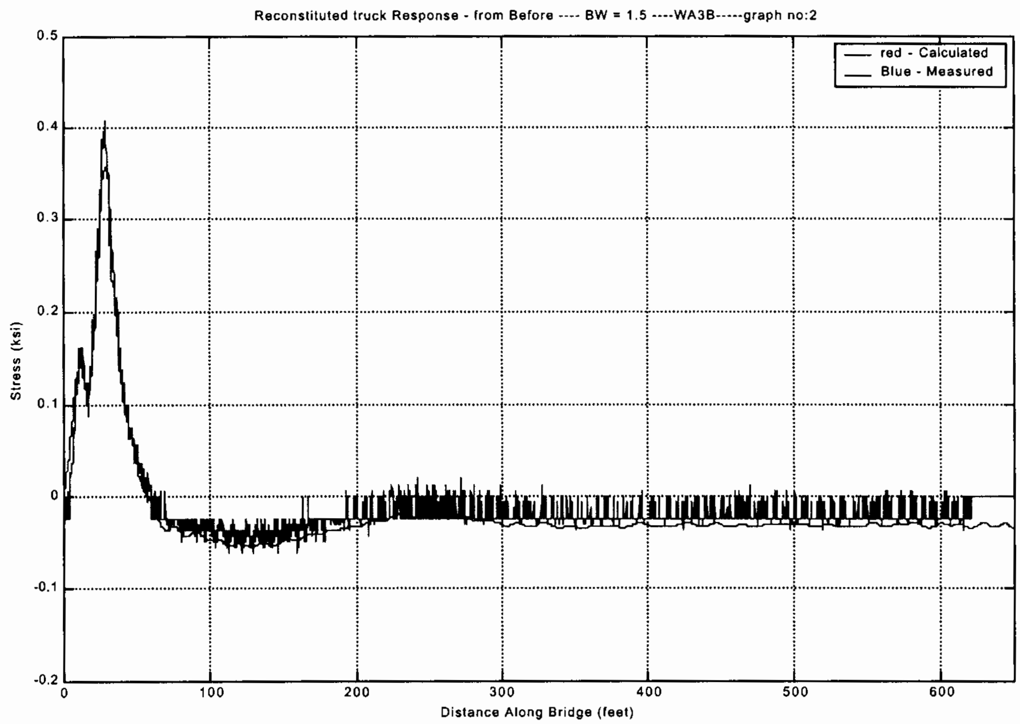

Fig. IV - Comparison of Actual Bridge Response with Predicted Bridge Response from Simulation UIL's

Current Capacity and Scope of Program

Though the bridge modeler was originally designed to handle six simple steel-stringer bridges, it has now evolved to handle a wide range of bridges with special features. Particularly the Bridge Modeler can now handle bridges with the following features

- Cover plates (moment plates) of different lengths and different sections on each pier.
- Cross braces with equal or unequal spacing and also with special end-braces
- Multiple section types within same girder line.

Many more changes will be made to make the model creation process easier, better and robust. Apart from the pre-processing of the model, UCII is also venturing into automation of the post-processing of the output files from SAP90™. The output files generated by the SAP90 analysis program have many headers, page breaks, and titles etc that are sometimes unnecessary. The information needed from the output files is very specific and requires manual searching and copying for further use. Once automated, the model creation and tuning process is much more efficient.

FE Model Calibration

If the bridge analyzed is a new structure, a large fluctuation is not expected in the local element properties throughout the bridge. This would reduce the level of effort needed for calibration by having the same material properties and geometry for the nominal through calibrated models. If the objective bridge is an aged and deteriorated one, possible damage and deterioration should be incorporated in grouping the parameters.

The sequence of parameter variation during manual calibration followed the general relationships established during sensitivity studies. The sensitivity studies examined the impact of the variation of parameters upon the dynamic and static responses of the bridge. For example, the order of the modes was affected by both the stiffness of the rigid links connecting the shell and frame elements and the stiffness of the linear and rotational springs representing the boundary conditions at abutments and piers. Reasonable variations in the stiffness of the girder and slab elements minimally influenced the frequencies and certainly affected the deflected shape. Reasonable estimates for the stiffness of the linear and rotational springs, which simulated the stiffness provided to the bridge at the abutments, were based on indirect measurements of the corresponding displacements and rotations during modal tests. After the FE model is constructed based on nominal values and idealized boundary conditions, eigenvalue analysis results of the nominal model are compared with the experimental results. The objective of this calibration is to quantify frequency errors and MAC values. The entire process is summarized below.

1. **Data Post-Processing**
   The parameter-estimation technique involves two steps: estimation of frequencies and damping and determination of mode shapes. After the modal testing, the collected data is analyzed to obtain frequencies and mode shapes. For the theoretical analysis, the modal z-direction displacements are obtained at every nodal location at the deck elevation.

2. **Comparison of Model Parameters**
   Although the analytical model had not been calibrated yet, it was developed from the basic theory of FE technique. Since all the stiffness and geometry were considered and included, the results from SAP90 should provide reasonable results. If not, all the stiffness and geometry should be reconsidered.

3. **Comparison of Natural Frequencies**
   The differences between the measured and predicted natural frequencies are calculated.

4. **Comparison of Mode Shapes**
The comparisons for the mode shapes are accomplished using MAC matrices. The Modal Assurance Criterion (MAC) is defined as

$$MAC(\psi_E,\psi_A)=\frac{[\sum_{i=1}^{N}\sum_{j=1}^{N}(\psi_{Ei})(\psi_{Aj})^T]^2}{\sum_{i=1}^{N}\sum_{j=1}^{N}(|\psi_{Ei}|^2|\psi_{Aj}|^2)}$$

The experimentally measured mode shape ψ_{Ei} was obtained from experimental results. The analytical mode shape was obtained from the SAP90 program. After the plots of z-direction displacements vs. nodal location along the length of the bridge in Microsoft Excel files were obtained, quadratic or linear interpolation was used to get the displacements according to every sensor location in the bridge test. The modal displacements of every tested girder are connected end to end to form both an analytical and an experimental vector for each mode for the calculation of MAC matrix. Then, a simple routine is used to complete the MAC calculation.

CONCLUSION

As is apparent from the discussion above, the UCII Bridge Modeler is certainly a milestone in the process of 3D FE modeling of bridges. We now can have models of bridges developed in a matter of few hours and once these models are calibrated against test results, they become the perfect platforms to implement and analyze modifications and special conditions right on the desktop. The horizon for expansion for this software is far and wide.

REFERENCES

1. Turer, Ahmet, Ph.D Dissertation, "CONDITION EVALUATION AND LOAD RATING OF STEEL STRINGER HIGHWAY BRIDGES USING FIELD CALIBRATED 2D-GRID AND 3D-FE MODELS", 2000
2. Cook, Robert; Malkus, David; Plesha, Michael; Witt, Robert , "CONCEPTS AND APPLICATIONS OF FINITE ELEMENT ANALYSIS", 4th Ed., 2002
3. Computer & Structures Inc. SAP90 User Manual, Volume I, 1991
4. "STRUCTURAL IDENTIFICATION: ANALYTICAL ASPECT", by Emin Aktan, Necati Catbas, Ahmet Türer and Zongfen,Zhang. "JOURNAL OF STRUCTURAL ENGINEERING", July 1998.