LA-UR-02-2096
*Approved for public release;
distribution is unlimited.*

Title: **STRUCTURAL HEALTH MONITORING SYSTEM DESIGN USING FINITE ELEMENT ANALYSIS**

Author(s): Daniel W. Stinemates and Joel G. Bennett

Submitted to:

http://lib-www.lanl.gov/cgi-bin/getfile?00796851.pdf

STRUCTURAL HEALTH MONITORING SYSTEM DESIGN USING FINITE ELEMENT ANALYSIS

Daniel W. Stinemates[†] and Joel G. Bennett [†]

† Technical staff member, Engineering Sciences & Applications Division, Weapon Response Group, M/S P946, Los Alamos National Laboratory, Los Alamos, NM 87545

ABSTRACT

The project described in this report was performed to couple experimental and analytical techniques in the field of structural health monitoring and damage identification. To do this, a finite element model was constructed of a simulated three-story building used for damage identification experiments. The model was used in conjunction with data from the physical structure to research damage identification algorithms. Of particular interest was modeling slip in joints as a function of bolt torque and predicting the smallest change of torque that could be detected experimentally. After being validated with results from the physical structure, the model was used to produce data to test the capabilities of damage identification algorithms. This report describes the finite element model constructed, the results obtained, and proposed future use of the model.

1. INTRODUCTION

The ability to continuously monitor the structural health of civil structures and mechanical systems is desirable for economical and human safety reasons. Staff at Los Alamos National Laboratory (LANL), as well as students participating in the Dynamics Summer School at LANL, have recently been developing such health monitoring techniques. Some of their efforts have been focused on statistical vibration analysis for damage identification techniques using experimental acceleration data. One particular experiment used is based on acceleration data from a steel frame structure (“bookshelf”) that simulates a three-story building. The structure is subjected to vibration excitation, and the response is measured with accelerometers. Different damage cases are studied by loosening bolts in joints to introduce damage to the structure. The goal of this project was to create and utilize a finite element model of the test structure to provide an analytical compliment to the experimental work underway.

For most real-world applications, information from the damaged system will not be available. Numerical simulations will need to be utilized to define sensing system properties in order to deploy systems on real world structures. Examples of these properties are bandwidth, sensitivity, dynamic range, optimal location and possibly excitation source amplitude and waveform. The goal of this study is to demonstrate the ability to define sensing system properties a priori with a structure that can be "damaged" and "repaired" as necessary.

The specific goals pertaining to the model were to create a linear model of the structure if possible, investigate the sensing system requirements, determine preferential locations for sensors around the joints, predict the smallest experimentally detectable change of preload in joint bolts, provide acceleration data to be used to test damage identification algorithms, and investigate form and location to apply excitation. The details of the model and the results obtained are described in this report.

2. THE FINITE ELEMENT MODEL

The finite element model was constructed to match the geometry and physical properties of the bookshelf structure. Figure 1 provides a view of the model that also illustrates the components of the physical structure. At the bottom of the structure, the base plate is supported by four air bearings (not visible). Four vertical columns are attached to the top of the base plate with the column brackets. Three floor plates are attached to the columns with the floor plate brackets.

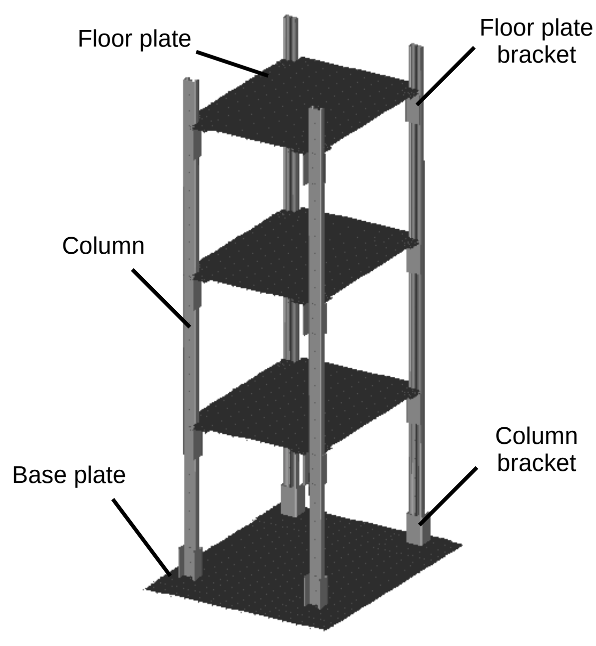

Figure 1: Finite element model of experimental structure

2.1 Components of the Structure

Floor plates

Each of the three floor plates is a rectangular aluminum plate with squares cut out of each corner for the columns. The dimensions of the aluminum plate are 24.188" x 18.188" x 0.5" thick. The floor plates were modeled with shell elements using aluminum material properties and the same thickness as the physical floor plates.

Base plate

The base plate is a 1.5-in.-thick rectangular aluminum plate, 24-in. wide by 30-in. long. The base plate was also modeled with shell elements and aluminum material properties.

Air bearings

The base plate was mounted on four air bearings to isolate the structure from incident vibrations. To model the air bearings, springs were attached from nodes on the base plate to ground. The nodes were chosen at locations corresponding to the locations where the air bearings were attached in the physical model. Values for the spring constants were obtained from the air bearing manufacturer's specifications. These values were updated during modal analysis validation to achieve proper rigid body modal frequencies associated with the structure rocking on its base.

Columns

The columns are 60-in.-long B-line brand stainless steel channels. Figure 2 shows the cross section shape and major dimensions of the columns. Beam elements with the same geometric cross section as the physical columns were used in the model.

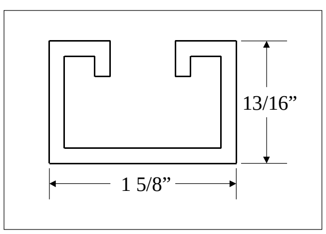

Figure 2: Cross section of column

Column brackets

To attach the columns to the base plate, B-line brackets were used. These brackets are referred to as column brackets throughout this report. Each bracket is comprised of a flat plate that is bolted to the base plate and a U-shaped channel that extends perpendicular to the base plate and encompasses the bottom 3-1/2" of the column. The column is bolted to the U-shaped channel of the bracket. The plate portion of each bracket was modeled with shell elements. To model the U-shaped channel, the cross section of the column was adjusted in that length to include the geometry of the bracket. The cross section of the column including the bracket is shown in Figure 3.

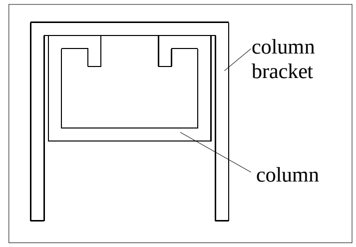

Figure 3: Cross section of column with column bracket

Floor plate brackets

Brackets are used to attach each floor plate to the four columns. Each bracket, referred to as a floor plate bracket, has two flat plates to attach to the floor plate and an L-shaped channel to attach to the column. Each of the flat plates is rectangular with dimensions of 1.5" x 1.875". The L-shaped channel is 3.625" tall, and fits with the column as shown in Figure 4. As was done with the column brackets, the cross section of the column is adjusted in the regions where the floor plate brackets are attached to include the cross section geometry of the brackets. The plates of each bracket are rigidly connected to the columns with beam constraints.

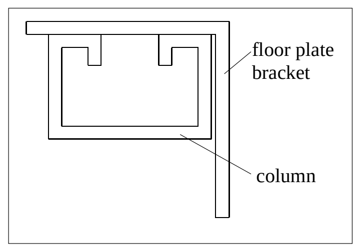

Figure 4: Cross section of column with floor plate bracket

2.2 Contact Surfaces

The original intent of the project was to create a linear finite element model of the structure, therefore contact between the floor plates and the floor plate brackets was simulated by connecting linear spring elements from nodes on the floors to nodes on the brackets. The placement of these spring elements can be seen in Figure 5.

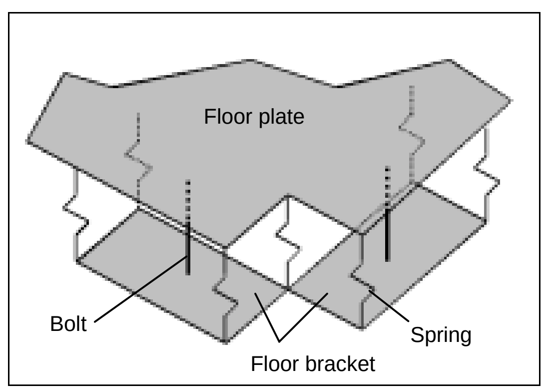

Figure 5: Isometric view of corner joint (with column removed) showing location of contact springs

The results from the linear model were not as desired, so the springs were changed to nonlinear springs. The nonlinear springs resist motion with linearly increasing force up to a certain magnitude, after which the resisting force remains constant. In this way, the nonlinear springs model friction and slippage in the joint. The spring constants for the nonlinear springs were estimated based on the motion in the joint in the linear model. These spring constants should be updated and validated in future work with this model.

2.3 Bolts and Damage Cases

The bolts in the joints of the structure were modeled as beam elements connecting nodes on the floor plates to nodes on the floor brackets. The location of the bolt beam elements can be seen in Figure 5. The tightness of the bolts was introduced by applying a stress initial condition in the axial direction of the bolts. The magnitude of the stress was calculated from the equation:

$$F_i = \frac{T}{0.20d} \tag{1}$$

where F_i is the preload in the bolt, T is the torque to which the bolt is tightened, and d is the fastener size. The stress can then be calculated as:

$$\sigma = \frac{F_i}{\frac{\pi}{4}d^2} \tag{2}$$

For damaged cases, the preload in the bolts at one of the joints was reduced to model loosening of the bolts. The damage was introduced in three different ways during this study. The first method was to reduce the preload of the bolts at the beginning of the analysis and leave them constant throughout the analysis. This method models a joint that has been damaged, but is not being further damaged. The second method was to reduce the stress in the bolt linearly throughout the analysis. This method models a joint that is progressively loosening or being damaged. The third method of damage introduction is a step reduction of stress at some point during the analysis. Many other damage cases could be introduced to the model in further studies. The ability to easily apply different types and magnitudes of damage to the structure is one of the main advantages this analytical method has over experimental data collection.

2.4 Labeling Conventions and Axes Orientation

To effectively communicate about the model, a labeling convention was chosen for identifying the joints. Figure 6 shows the numbering convention for the columns. The beams were labeled 1-4 as shown in the top view.

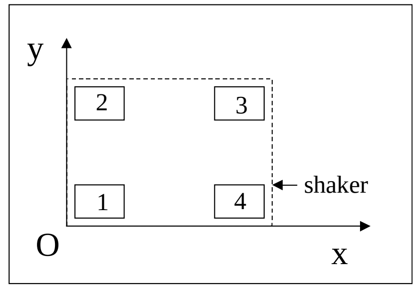

Figure 6: Top view showing numbering of columns

Figure 7 shows the labeling convention of the floor plates. The plates are labeled a, b, and c starting from the top floor.

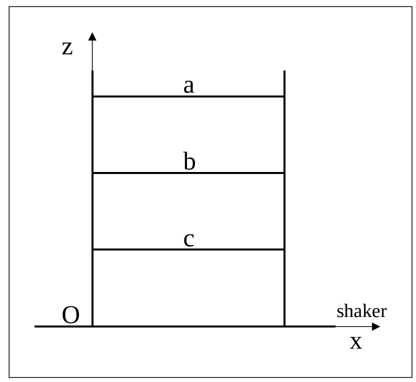

Figure 7: Side view showing lettering of floors

To specify a particular joint, the column number and floor letter are used. For example, in the finite element model, the bolts were loosened at joint 3a, the joint at the top floor on the corner counterclockwise from the shaker location.

2.5 Accelerometer Locations

Accelerometers were attached at the joints of the physical structure and were oriented to measure acceleration in the x-direction (as shown in Figure 6 and Figure 7). Figure 8 shows the accelerometer layout on one of the floor plates, which is typical of all three floors. At each joint, one accelerometer was attached to the column and one accelerometer was attached to the floor plate.

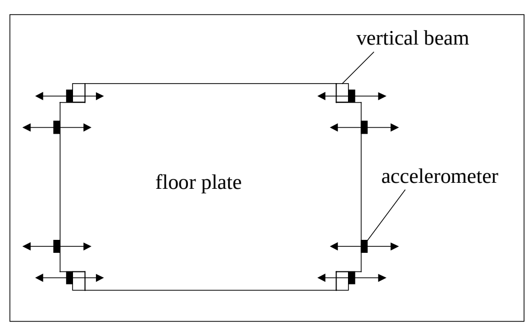

Figure 8: Top view of a floor plate showing typical location and orientation of accelerometers

2.6 Shaker Location and Input Time Histories

A shaker attached to one side of the base plate supplied the excitation input for the physical experiment. The location is illustrated in Figure 6 and Figure 7. Force time histories measured during the experiment were used as the excitation input to the finite element model. The force was applied at a node corresponding to the location of the shaker attachment on the physical structure. The excitation was 0–200 Hz random signal with rms level of 5.9 pounds.

3. MODEL VALIDATION

Before being used to generate acceleration response data, the model was validated by comparing it to the physical structure. The two areas compared were weights of the components for the model and the physical structure and modal analysis results for each.

3.1 Comparison of Weights of Components

The weights of the components of the finite element model and the physical test structure are shown in

Table 1. As seen in the table, the weights of individual components are not exactly equal, but they are close, and the total weights of the two cases are almost exactly equal. One discrepancy that needs clarification is the weight of the base plate and air bearings. When the physical structure was weighed, the air bearings were left attached to the base plate, so their weight is included in the weight listed for the base plate in the table.

Table 1: Weights of structure components

	Weight (lb.)	
Component	Model	Physical
floor plates	63.0	65.0
base plate	105.8	120.0
air bearings	10.1	
vertical brackets	10.2	14.0
floor brackets	15.3	13.0
columns	37.8	25.8
base bolts	1.5	2.8
floor bolts	0.5	3.7
Total weight	244.3	244.2

3.2 Comparison of Modal Analysis Results

Experimental and analytical modal analyses were performed on the physical structure and with finite element model, respectively. The frequencies (in Hertz) for the first several modes of each are shown in Table 2. The mode shapes were also compared, and they, like the frequencies, were similar for each case. The correlation was not formally quantified.

Table 2: Frequencies (Hz) of mode shapes from experimental data and finite element model

Mode number	Experimental	FE Model
1	2.29	3.03
2	3.04	3.87
3	12.57	6.76
4	13.90	7.27
5	14.46	11.00
6	24.87	20.12
7	32.04	34.95
8	40.08	39.89
9	49.82	50.84
10	69.10	57.59
11	73.42	71.06
12	74.30	83.55
13	120.33	126.27
14	138.89	130.10
15	145.04	134.28
16	187.59	146.38
17		158.17
18		174.86
19		176.74
20		176.80

4. DATA GENERATED

Acceleration data were calculated with the finite element model to correlate with the experimental data. Nodes on the model were chosen that were located close to where the accelerometers were attached to the physical structure (as seen in Figure 8 above). The acceleration time history in the direction measured by the accelerometers was recorded for each of these nodes.

4.1 Desired Data

The damage identification algorithms under consideration at the time of this project compare acceleration time histories of a baseline undamaged case and a test (damaged) case to determine if, and where, damage is present in the test case. Data generated by the finite element model were manipulated to be in the form of a difference of two accelerations. The difference of acceleration across each joint was calculated by subtracting the acceleration history of the accelerometer node on the plate at a joint from the acceleration history of the accelerometer node on the column at the same joint. This difference time history was then examined and analyzed to detect damage at the joint.

4.2 Example Data

Figure 9 and Figure 10 show plots of typical acceleration time history data generated by the finite element model. Figure 9 shows data from the model that used linear springs to simulate the contact between plates at the joints. One of the two lines plotted is the difference between the accelerations of the accelerometer nodes at joint 3a in the undamaged case, and the second line is the difference in the damaged case when the pretension in the bolts at joint 3a was reduced by one-half. As seen in the figure, no noticeable difference is evident between the two lines. This plot illustrates the problem with the linear model and the need to expand to the nonlinear spring model.

Figure 10 shows data from a damaged and undamaged case of the model with nonlinear springs to simulate the plate contact at the joints. Although the differences between the two lines are small, they are evident, and when statistical methods are used to analyze the data, the damage to the joint is detectable.

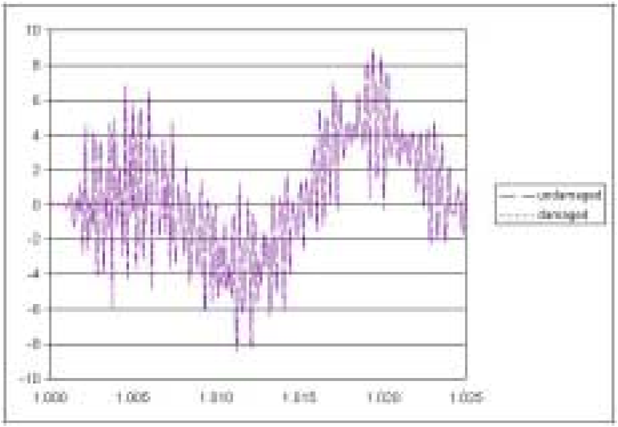

Figure 9: Acceleration data from model with linear spring contact

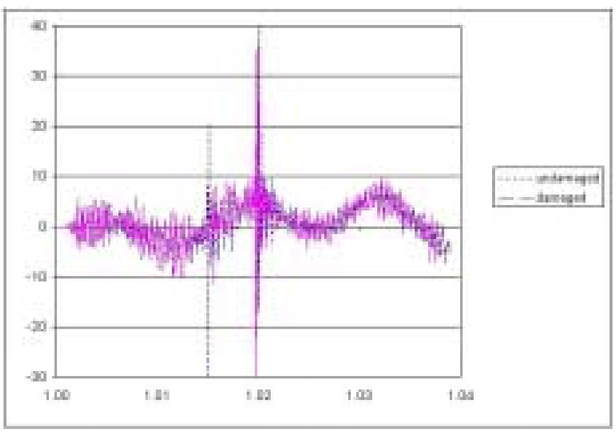

Figure 10: Acceleration data from model with nonlinear spring contact

5. INITIAL SENSING SYSTEM REQUIREMENTS

In this project, the number of sensors (accelerometers) and their locations were predefined. Based on the results from the finite element analysis, some conclusions about the requirements of the sensing system can be drawn. The results show that in order to detect a loss of preload of about 90%, the sensor has to detect signal changes on the order of 0.05 g's amplitude. The sensing system must be able to sample data at 20 kHz in order to capture the frequency bands necessary to distinguish the healthy system from the damaged system.

When further analysis is complete, the results obtained from FE simulations will be experimentally verified regarding required sensing parameters.

6. FUTURE APPLICATIONS

The finite element model constructed was effectively used to generate acceleration data to be used in damage identification research. The model can be used to generate data for a wide range of damage cases. Some of the variables that can be adjusted are as follow:

- magnitude of input excitation
- frequency content of input excitation
- location of input (where the shaker is attached)
- location of damage (which joint is loosened)
- magnitude of damage
- different damage cases

The model could be used to determine optimal excitation levels and locations to excite the structure in the most efficient manner for detecting and locating damage.

This method can also be expanded to aid in specifying the health monitoring system for other structures, including experimental structures as well as real civil structures.

7. CONCLUSION

The project was a good first step to adding an analytical approach to sensor system definition for the damage detection research. The model closely resembles the physical bookshelf structure in geometry, physical properties, and modal response to vibration. When nonlinear springs are used to model the contact between plates at the joints, differences can be detected in acceleration data from nodes corresponding to accelerometer locations on the floor plates and the columns of the structure. The model can be used to generate acceleration data to be used to test damage identification algorithms and to explore other excitation and sensing experimental configurations.

8. ACKNOWLEDGEMENTS

Other members of the Engineering Sciences & Applications Division, Weapon Response Group contributed to this project. Thank you to Chuck Farrar, Hoon Sohn, Dave Allen, Francois Hemez, and any others that provided insight or support for this project.