

Dynamic Structural Analysis with Experimental Verification

by

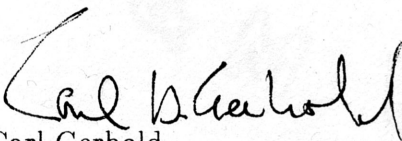
Robert A. Judge

Mechanical Engineering

Submitted in Partial Fulfillment of the Requirements of the
University Undergraduate Fellows Program

1985-86

Approved By:



Dr. Carl Gerhold

April 1986

Abstract

There are basically two methods of determining the dynamic response of structures. Analytical methods take advantage of theoretical equations to predict a system's response, while experimental methods use testing of prototypes or scale models of a structure to determine actual response.

In order to design structures efficiently, an engineer should be able to use theoretical tools to analyze and predict a system's response. This research project was designed to predict the accuracy of modeling structures subjected to dynamic loading using the finite element method. A finite element computer program was written and used to analyze two structures—a simple cantilevered beam and a cantilevered beam with another beam welded to it at a 90° angle. The structures that were modeled by the program were then constructed and tested using modal analysis techniques. The finite element model and experimental results were then compared to see how accurately the model predicted actual system response.

The results of this research showed that the finite element model was able to predict actual system behavior with a high degree of accuracy. The analytical model was able to predict natural frequencies and mode shapes within 19.1% of the experimentally-determined values for the cantilevered beam, and within 6.1% of the experimentally-determined values for the welded beams.

Acknowledgements

I would like to thank Dr. Carl Gerhold for his help in formulating, executing, and completing this project. I would also like to acknowledge the assistance of Dr. Alan Palazzolo for his help in explaining the use of some of the lab equipment.

I would especially like to thank Sandra Douglas for putting up with me (and the lack of me) during this project.

Table of Contents

Abstract	ii
Acknowledgements	iii
List of Figures	v
Introduction	1
Background	2
Finite Element Model Formulation	3
Experimental Validation of Finite Element Model–Cantilevered Beam	6
Results of First Test	11
Finite Element Model Formulation for Welded Beams	12
Experimental Validation of Finite Element Model for Welded Beams	17
Results of Second Test	17
Discussion of Results	21
Conclusions	25
References	26
Appendix A	27
Appendix B	29
Appendix C	31
Appendix D	33
Appendix E	37
Appendix F	39
Appendix G–Finite Element Computer Program	42
Vita	53

List of Figures

Figure 1–Theoretical Mode Shapes of Cantilevered Beam	7
Figure 2–1st Mode Shape Predicted by Finite Element Program	8
Figure 3–2nd Mode Shape Predicted by Finite Element Program	9
Figure 4–3rd Mode Shape Predicted by Finite Element Program	10
Figure 5–1st Mode Shape Determined Experimentally	13
Figure 6–2nd Mode Shape Determined Experimentally	14
Figure 7–3rd Mode Shape Determined Experimentally	15
Figure 8–Accelerometer Placement for Welded Beam Test	16
Figure 9–1st Mode Shape Predicted by Finite Element Program– Welded Beams	18
Figure 10–2nd Mode Shape Predicted by Finite Element Program– Welded Beams	19
Figure 11–3rd Mode Shape Predicted by Finite Element Program– Welded Beams	20
Figure 12–1st Mode Shape Determined Experimentally– Welded Beams	22
Figure 13–2nd Mode Shape Determined Experimentally– Welded Beams	23
Figure 14–3rd Mode Shape Determined Experimentally– Welded Beams	24

Introduction

Dynamic structural analysis is a term used to describe the prediction of a structure's behavior when it experiences motion. The analytical tools that are employed in this type of analysis range from cumbersome classical equations to modern finite element techniques. The theory behind all the available methods is supposedly well-proven, but unfortunately most of the proof exists only on paper.

In contrast to analytical methods of dynamic structural analysis are experimental methods. While the analyst can predict system behavior and argue that it is "theoretically correct", if experimental results contradict analytical ones, all the mathematic proofs in the world cannot dispute real-world occurrences. Many times, the result of such incongruities is the analyst saying "The lab guys don't know what they're doing," and the experimentalist saying "The analysts don't know what they're doing."

To eliminate the confusion that exists in the field of system modeling, more communication and feedback needs to take place between the analyst and the experimentalist. In particular, since finite element methods have become the major analytical tool in the dynamic structural analysis field, there should be experimental verification of results obtained using finite element modeling techniques. This would lead to improved analytical models, which in turn would result in more accurate design of structures that will be subjected to dynamic loading.

With the justification outlined above, the objective of the research performed was to develop a finite element model that could be used to predict

"The Journal of Sound and Vibration" was used for its style and format

dynamic system behavior with a degree of accuracy to be determined through experimentation.

Background

For Euler beams, the equations for calculating natural frequencies of simply supported, cantilevered, and pinned beams are well documented and thoroughly proven experimentally. However, when a structure becomes more complicated, such as two Euler beams welded together, there are no simple equations that can be used to calculate natural frequencies or mode shapes. Thus, classical analytical tools become extremely time-consuming and complex. For this reason, finite element techniques are usually employed for modeling any structure consisting of more than a simple beam.

The use of finite element theory to analyze dynamic response of structures is well documented. Unfortunately, most of the literature relates only how to model structures for input to programs [1, 2, 3, 4]. Other published works in the field of structural analysis tell how to obtain results through experimental testing of actual or scale models, but do not present any analytical predictions of system behavior before the model is tested [5, 6]. There are very few examples where a physical model was constructed and tested to see how accurately an analytical model performed. In fact, one paper suggested a method of modeling an offshore oil platform using finite elements, but went on to say that there was no way to evaluate how well the model predicted actual response, due to the lack of experimental data [7].

There were two papers, both by the same authors, that actually compared the results of a finite element model of a structure to the results of experimental testing of the actual model. In one of the papers, the authors stress that accurate

natural frequency prediction is an important part of finite element analysis, but admit that the definition of "accurate" is difficult to pin down. They said:

"The frequency measurement is the most basic data that all modal tests measure. The test measured data and the analytically predicted values can be compared directly. Frequency correlation is considered very essential such that a valid model must be able to predict accurate natural frequencies...Left unanswered is the question of to what degree of agreement must these comparisons achieve in order to validate the analytical model." [8]

The results of the search for background literature on the topic of experimental verification of finite element models is the conclusion that there is very little published information in this area. The two pieces that were found to be relevant [8,9] were results of analysis and testing of a Viking spacecraft, which is a very complex structure. The objective of this research was to obtain similar data for simpler structures that would correlate well with the previously published data for the more complex structure.

Finite Element Model Formulation

The first task in the research performed was to develop a computer program that used finite element theory to calculate natural frequencies and mode shapes of engineering structures. Although commercially available finite element programs already exist, many users do not understand what the program is doing. By studying the theory and formulating the program, the finite element method became more than just a "black box" which "magically" obtained results when the correct data was entered.

The basic equation for any dynamic system is (assuming no damping):

$$\left[M \right] \ddot{X} + \left[K \right] X = \left\{ F \right\}$$

Where

- $\left[M \right]$ = System Mass Matrix
- $\left[K \right]$ = System Stiffness Matrix
- F = System Forcing Function

If F is set to zero, the natural frequencies and mode shapes of the system can be found. For harmonic motion in a principal mode, all parts of the system will move in one mode with the same frequency. Therefore the x term in the above equation can be assumed to be of the form

$$x = X \sin \omega t$$

Substituting this form of x into the differential equation with $F = 0$ yields

$$\left[K \right] X = \omega^2 \left[M \right] X$$

Multiplying each side of the matrix by the inverse mass matrix $\left[M^{-1} \right]$ yields

$$\left[M^{-1} \right] \left[K \right] X = \omega^2 X$$

Now if the matrix $\left[A \right]$ is defined as the product of multiplying the inverse of the mass matrix $\left[M^{-1} \right]$ by the stiffness matrix $\left[K \right]$,

$$\left[M^{-1} \right] \left[K \right] = \left[A \right]$$

$$\left[A \right] X = \omega^2 X$$

This equation corresponds to the classic eigenvalue or characteristic problem, whose solutions correspond to natural frequencies and mode shapes of the system being analyzed.

Finite element theory was used to assemble the system mass and stiffness matrices. A complete description of the particular theory used can be found in the book Structural Dynamics [10]. After the matrices were assembled, an eigenvalue–eigenvector solving routine was employed to solve for the system natural frequencies and mode shapes [11]. This routine had nothing to do with finite elements, but simply used linear algebra to solve a matrix equation.

Once the program was written and debugged, the accuracy of its output was compared to classical equations by using it to model a cantilevered Euler beam. Appendix A contains the theoretical calculation for the natural frequencies of a $2'' \times \frac{1}{8}'' \times 14''$ cantilevered beam. The first three frequencies were calculated to be 20.1 Hz, 126.0 Hz, and 352.6 Hz. The finite element program predicted the first through third natural frequencies to be 19.9 Hz, 125.7 Hz, and 352.1 Hz, respectively. This led to the conclusion that the finite element program could be used to accurately predict natural frequencies for a cantilevered Euler beam. A table showing the number of elements required to obtain accurate results can be found in Appendix B. From this sensitivity analysis, it was concluded that as many elements were needed in the model as the number of desired natural frequencies to be calculated. In simpler terms, if one desired an accurate value

for the third natural frequency of a cantilevered beam, at least three elements must be used in the model.

Figures 1a, 1b, and 1c are the theoretical mode shapes for the 1st, 2nd, and 3rd natural frequencies, respectively [12]. To see if the finite element program correctly predicted these mode shapes, plots were made of the modal data obtained from the program. Figures 2, 3, and 4 are the mode shapes predicted by the model, and they match the classically determined mode shapes presented in Figure 1.

Experimental Validation Of Finite Element Model For Cantilevered Beam

Once the finite element model was established as acceptable for use on cantilevered beams, a testing method was established to experimentally verify what appeared to be correct on paper. The main objective behind testing what was already known to be correct was not to beat the simple case into the ground, but to establish correct experimental techniques so they could be used for more complex structures with confidence in the results.

A simplified diagram of the experimental set-up is shown in Appendix C. The set-up consisted of the test piece, a PCB impulse-force (modal) hammer (model 086B), a PCB model 307A accelerometer, two PCB transducer amplifiers (models 480B and 482A), and a Hewlett-Packard model 3582 Spectrum Analyzer. To simulate a true cantilever, the $2" \times \frac{1}{8}" \times 14"$ steel test piece was sandwiched between a table and another steel beam, and the three were clamped together. The accelerometer was mounted at the free end of the test piece, and the distance between the cantilevered and free ends of the beam were divided up into 1" increments called "stations".

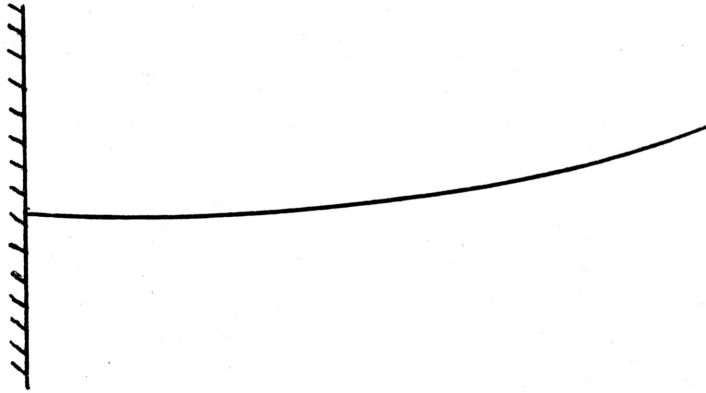


Figure 1a- 1st Mode Shape of Cantilevered Beam

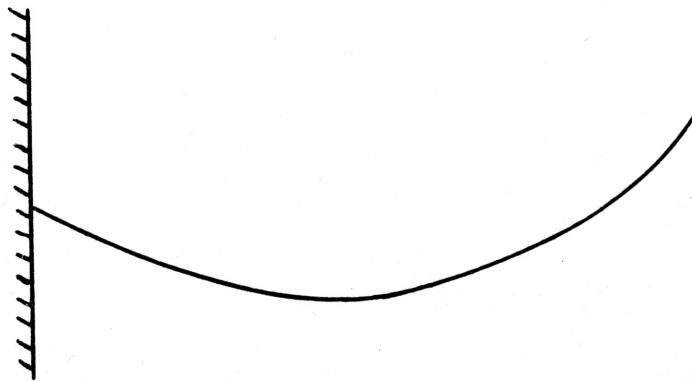


Figure 1b- 2nd Mode Shape of Cantilevered Beam

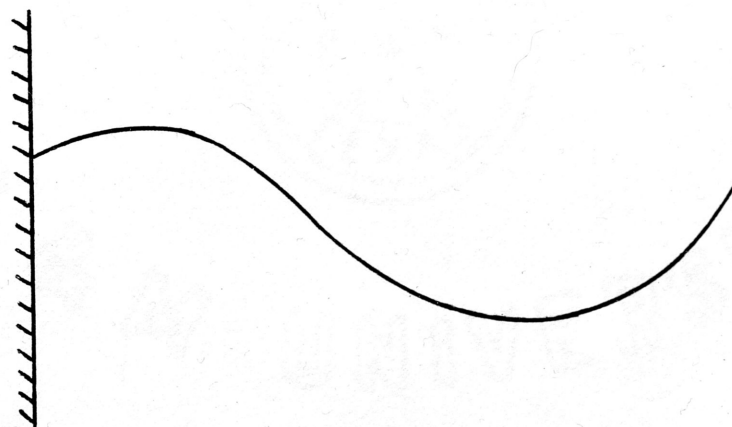


Figure 1c- 3rd Mode Shape of Cantilevered Beam

The modal hammer was used to input force into the beam by tapping at each of the stations along the test piece. The modal hammer's internal accelerometer provided the input signal to the spectrum analyzer, and the accelerometer mounted at the end of the beam provided the output signal for the system. After these signals were amplified by the charge amps, they were fed into the spectrum analyzer. The analyzer takes the input and output signals, derives the transfer function internally, and outputs a response contour. The peaks of the response contour correspond to the natural frequencies of the system being tested. At the contour peaks, the frequency, amplitude, phase, and coherence of the transfer function are calculated by the spectrum analyzer *for the station being tapped*. Thus, by plotting the peak amplitudes in their proper phase relations for each station against its position on the beam, an experimentally determined mode shape can be obtained. The coherence value is an indication of how free the transfer function was from extraneous signals, such as 60-cycle noise from power lines. The value of coherence is always from 0 to 1, and the closer to 1 the value, the clearer the transfer function is.

Results Of First Test

Appendix D is a table of the raw data obtained by testing the $2'' \times \frac{1}{8}'' \times 14''$ steel cantilevered beam. Also included in this appendix is a table showing the percent difference between the natural frequencies predicted by the finite element model and those found experimentally. The differences ranged from 14.1% to 19.6%. Although these were large differences, a great deal of the error was thought to have been introduced by not including the mass of the accelerometer in the finite element model. Also, in the paper by Chen, errors between analytically predicted and experimentally determined values were reported to be as high as 24.71%, so the results were not out of line with those from previously

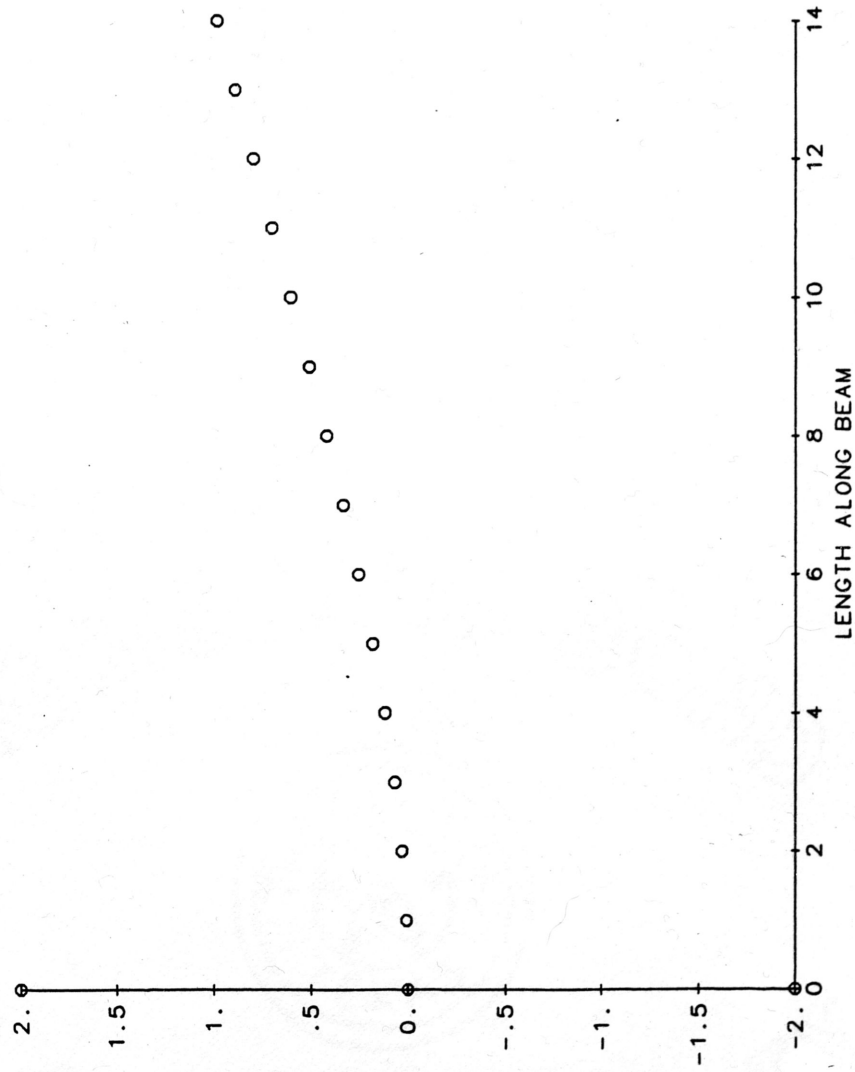


FIG. 2 THEORETICAL 1ST MODE SHAPE

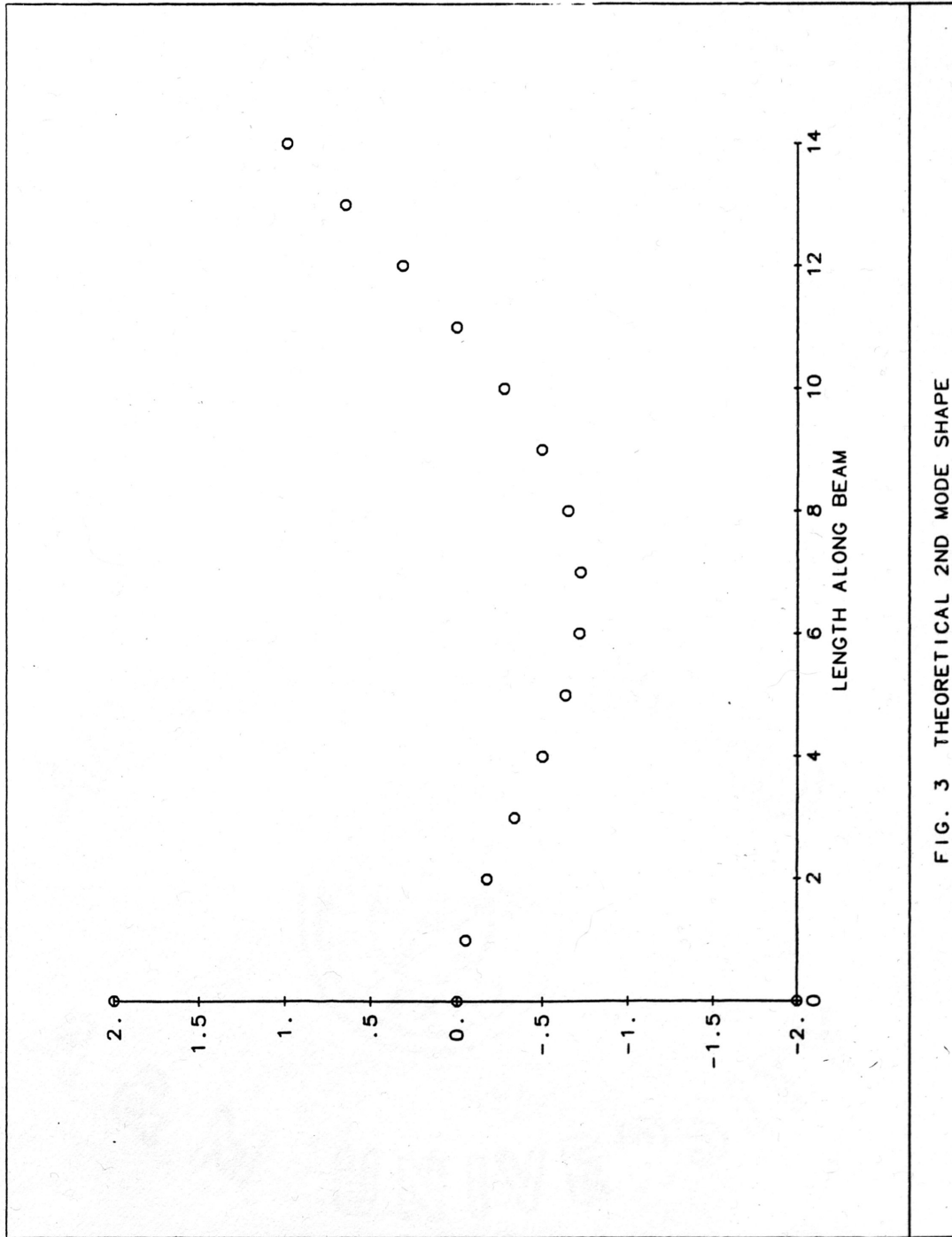


FIG. 3 THEORETICAL 2ND MODE SHAPE

conducted tests [8].

Figures 5, 6, and 7 are plots of the experimentally determined mode shapes for the 1st, 2nd, and 3rd natural frequencies, respectively. These matched up quite well with those predicted by the finite element model. Based upon these observations, it was decided that the model's performance was adequately confirmed by the experimental testing, and the model was next extended so as to be able to be used on welded beams.

Finite Element Model Formulation for Welded Beams

The first version of the finite element program was only able to be used to model beams whose elements were either all horizontal or all vertical. In order to model a beam consisting of a horizontal piece with another piece welded to it, in the same plane at any angle, a modification to the program was made. A transformation matrix, also called a cosine matrix, was formulated and used to convert all the local matrices into global ones, so that the elements of the matrices could be combined correctly. A transformation matrix simply modifies a local stiffness and mass matrix by multiplying it by the proper sine or cosine of the angle between the local coordinates and the global coordinates [10]. The basic equation, described previously, remains the same.

Once the program was modified and debugged, it was used to model a $1\frac{1}{2}'' \times \frac{1}{8}'' \times 12''$ cantilevered steel beam with a $1\frac{1}{2}'' \times \frac{1}{8}'' \times 7''$ steel beam welded to it at a 90° angle. Figure 8 shows a picture of this structure. The finite element program predicted the first three natural frequencies to be 12.8 Hz, 48.9 Hz, and 166.0 Hz. Appendix E contains the sensitivity analysis for the welded structure. Once again, it was shown that at a minimum, three elements

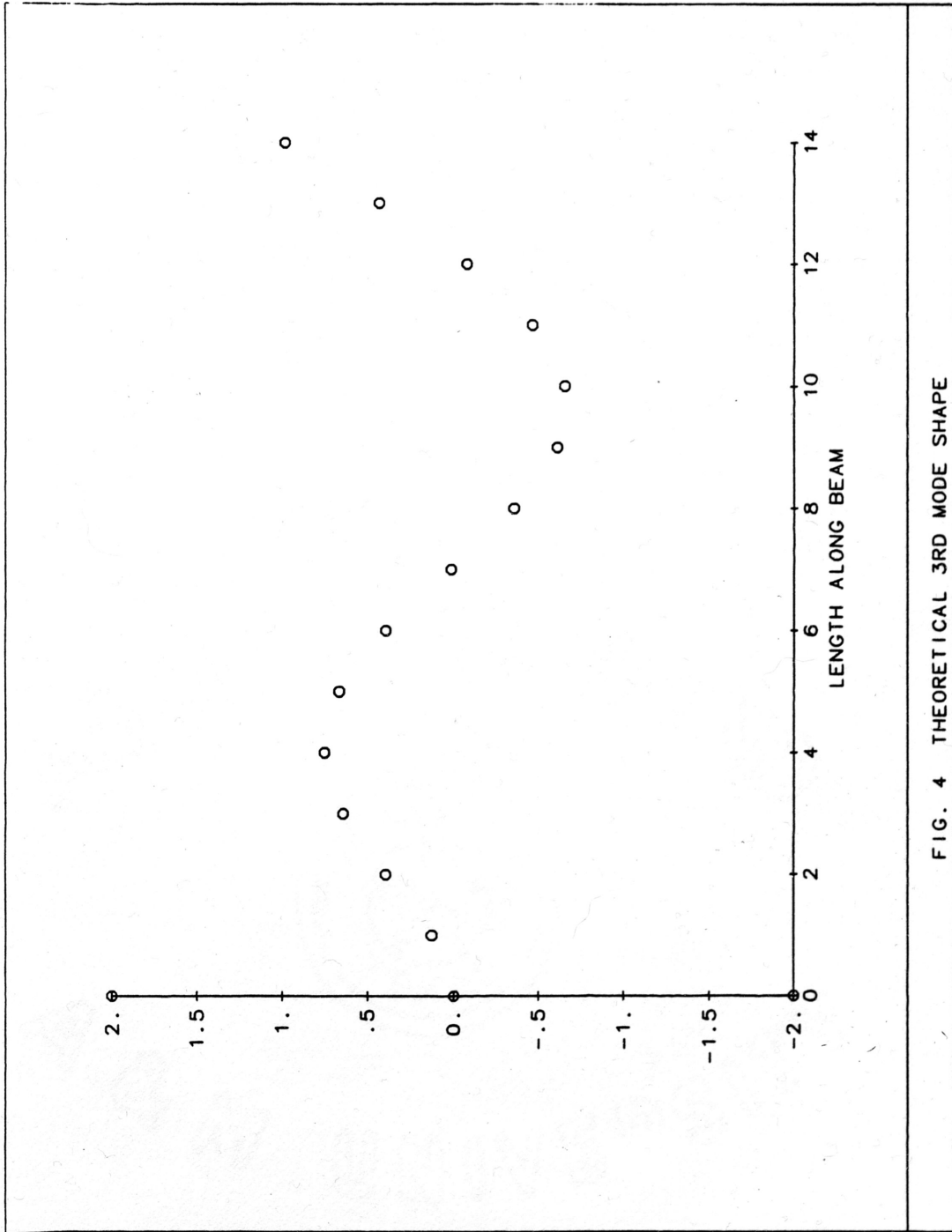


FIG. 4 THEORETICAL 3RD MODE SHAPE

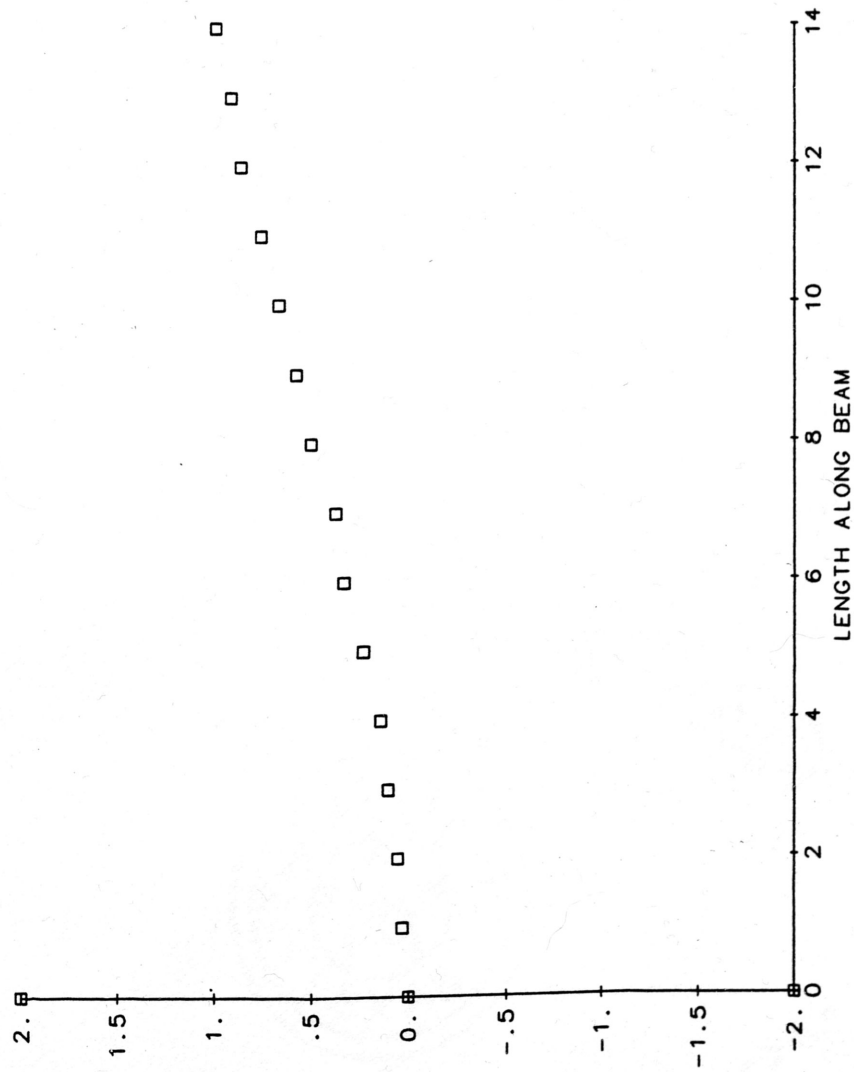


FIG. 5 EXPERIMENTAL 1ST MODE SHAPE

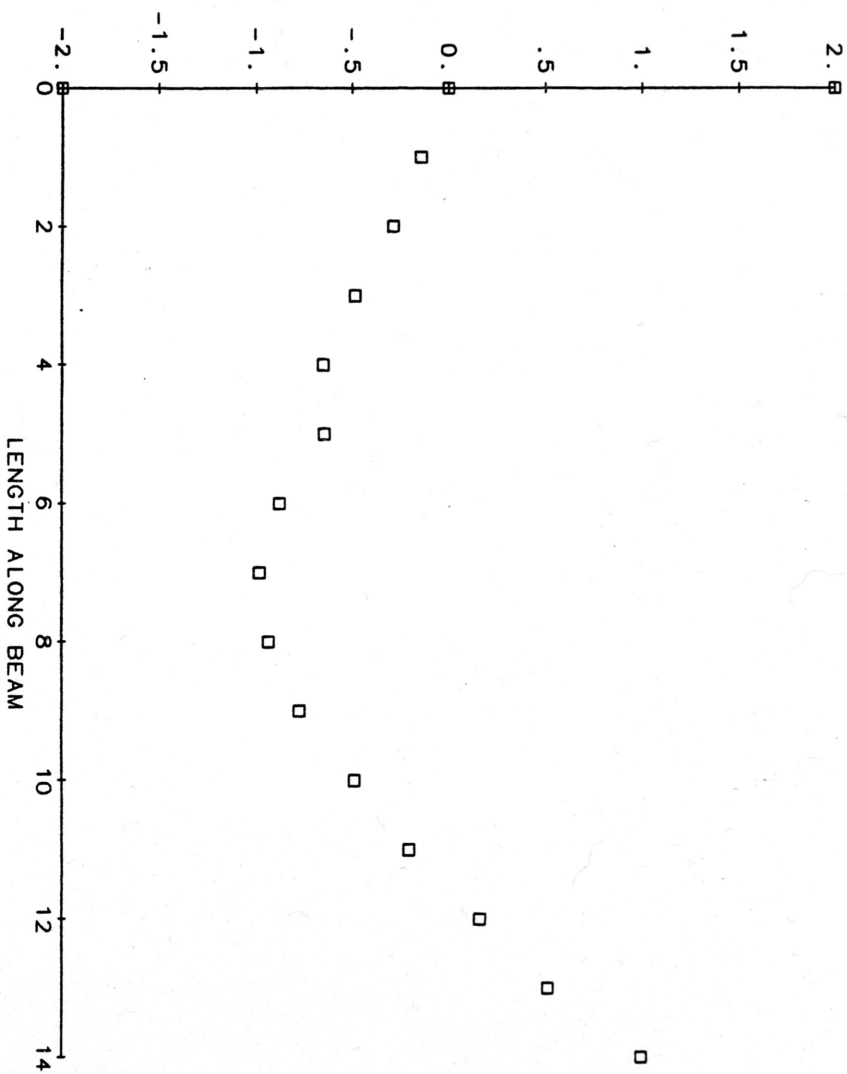


FIG. 6 EXPERIMENTAL 2ND MODE SHAPE

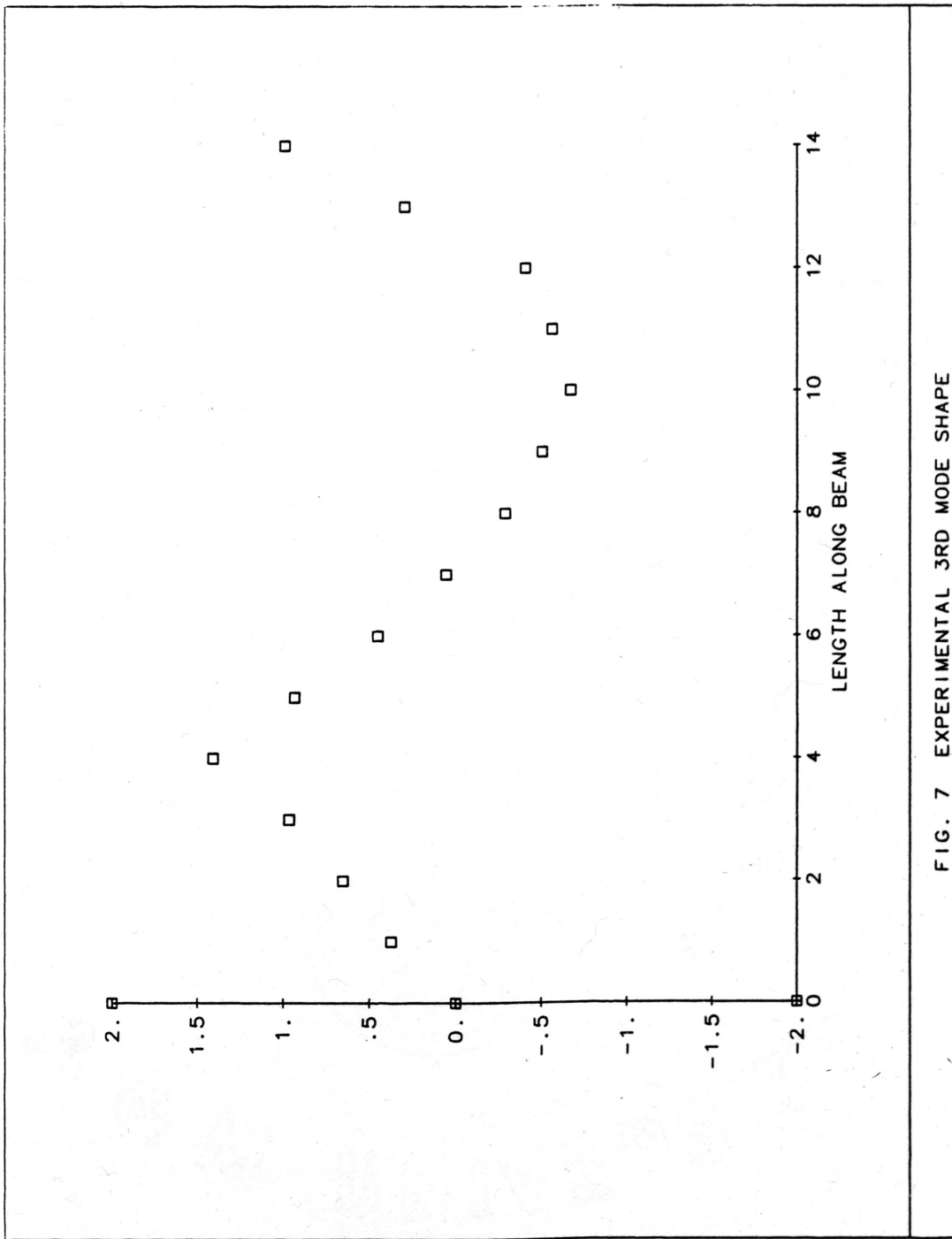


FIG. 7 EXPERIMENTAL 3RD MODE SHAPE

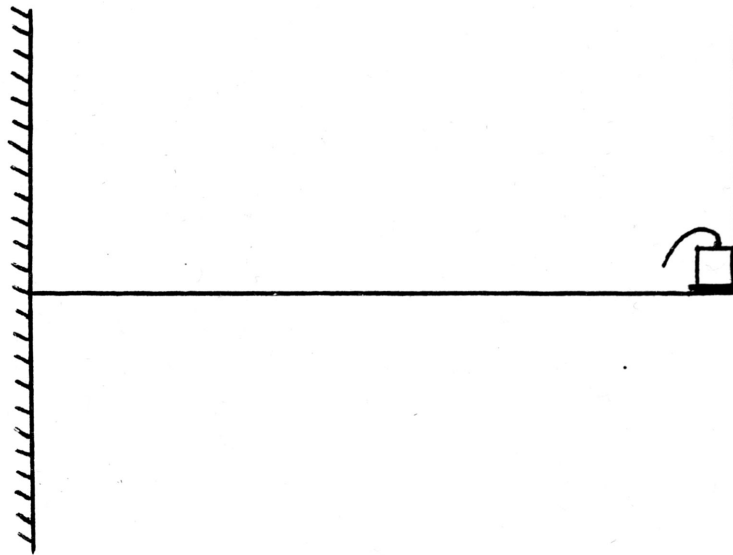


Figure 8a- Accelerometer Placement at Weld

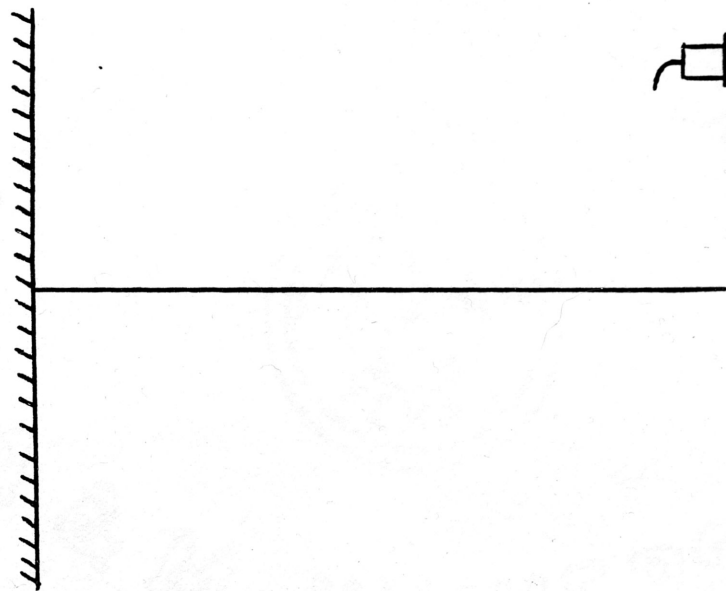


Figure 8b- Accelerometer Placement at End of Welded Beam

were needed in the model to accurately calculate the first three natural frequencies. Increasing the number of elements increased the program's CPU usage, but didn't increase the accuracy of the calculated values significantly.

Figures 9, 10, and 11 are plots of the predicted mode shapes for the welded beams. Since there were no classically determined "correct" examples for mode shapes of the type of structure modeled, it was left to experimentation to determine the validity of the model.

Experimental Validation Of Finite Element Model For Welded Beams

The same test set-up was used to test the welded beams as the one described earlier that was used to test the simple cantilevered beam. Of course, a different test piece was used. The test piece was a $1\frac{1}{2}$ " \times $\frac{1}{8}$ " \times 12" steel beam, cantilevered as described before, with a $1\frac{1}{2}$ " \times $\frac{1}{8}$ " \times 7" steel beam welded to it at a 90° angle. Two configurations of accelerometer placement were tested, one at the weld and one at the end of the welded piece (see Figures 8a and 8b). Appendix F contains the raw data for both configurations. The data obtained for accelerometer placement at the weld agreed better with the finite element model with regards to natural frequencies and mode shapes, so the data reported in this paper was obtained with this configuration.

Results of Second Test

Included in Appendix F is a table showing the percent difference between the natural frequencies predicted by the finite element model and those found experimentally. The differences ranged from 1.8% to 6.3%. The correlation between analytical and experimental results was much better for the case of the welded beams compared to the simple cantilevered beam.

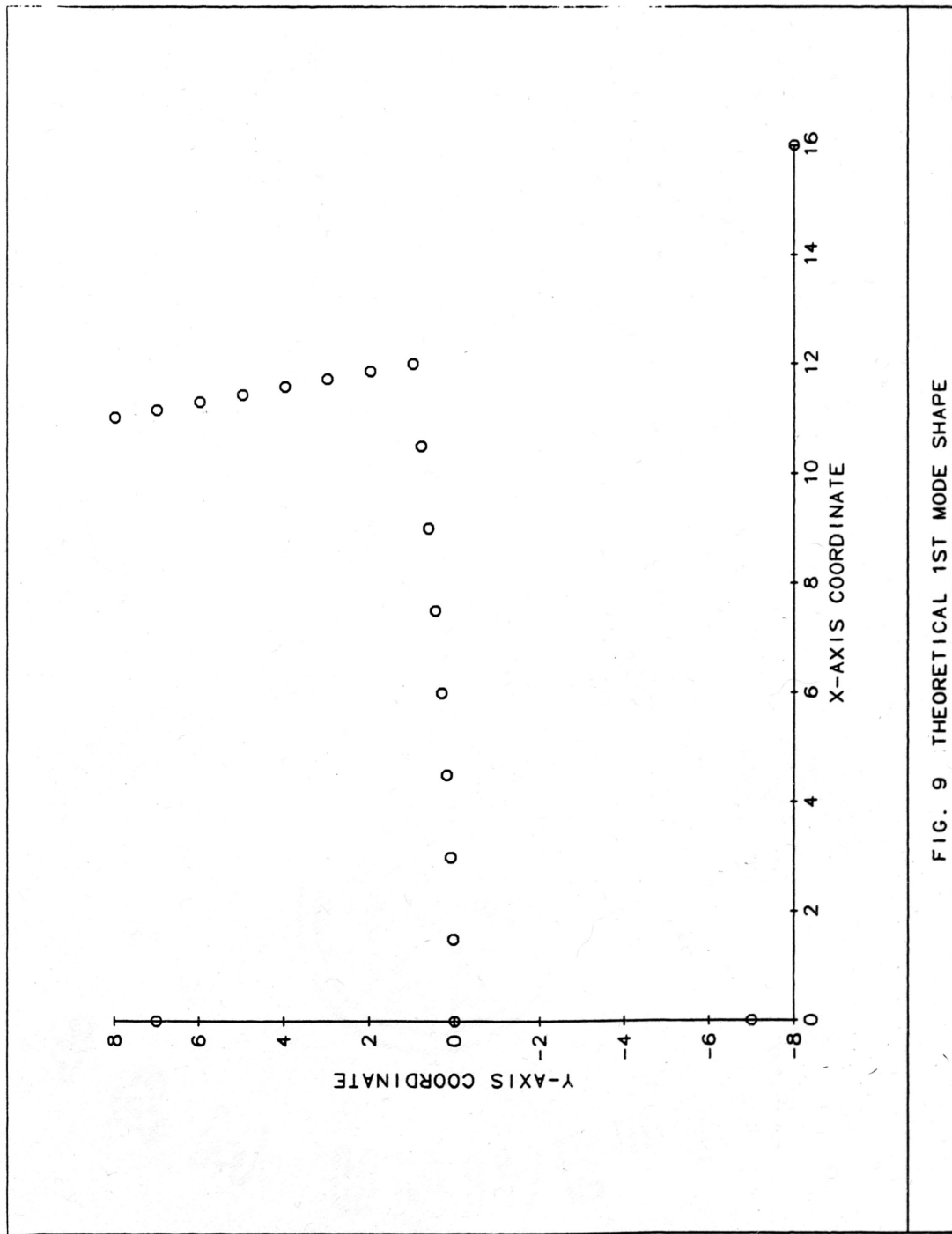


FIG. 9 THEORETICAL 1ST MODE SHAPE

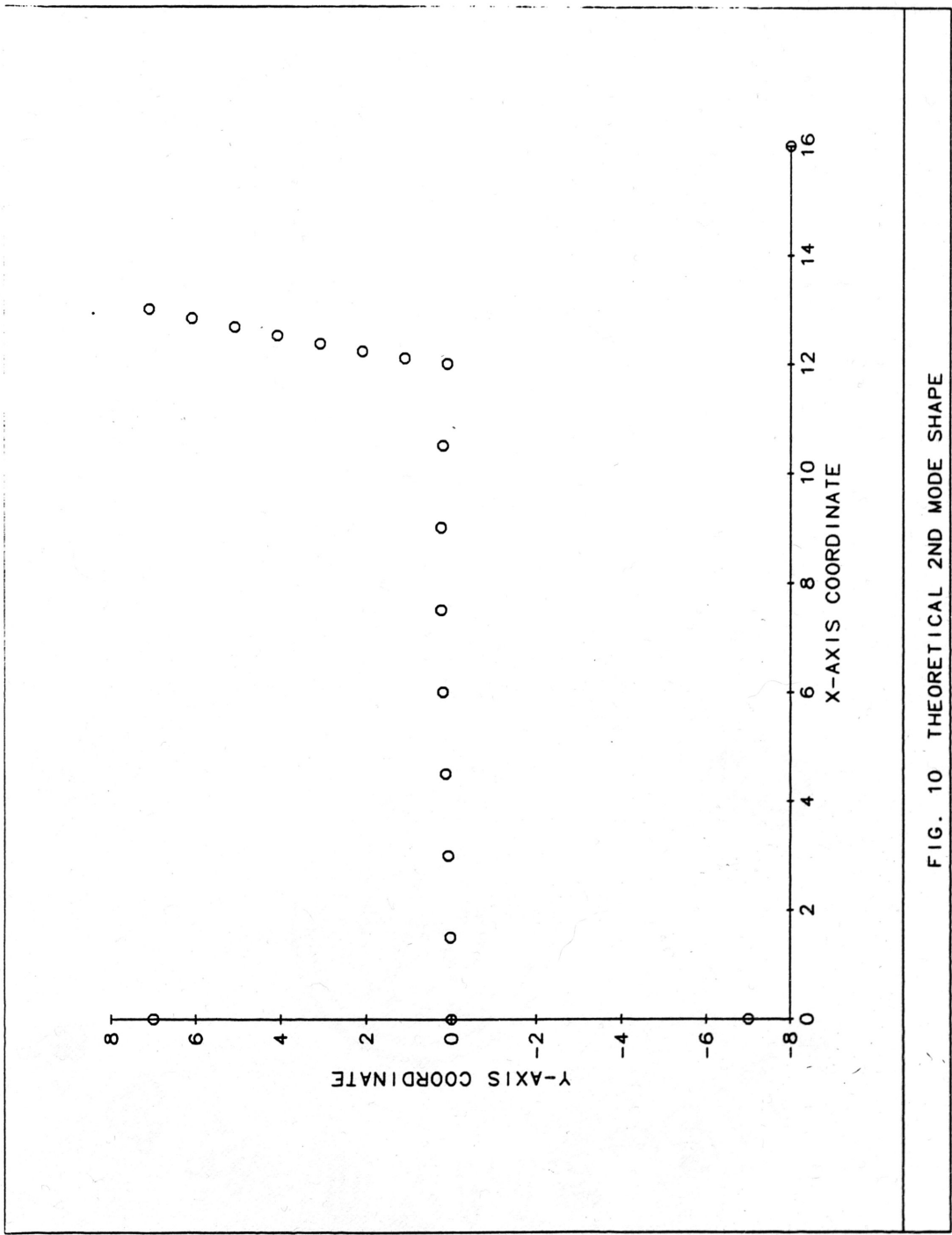


FIG. 10 THEORETICAL 2ND MODE SHAPE

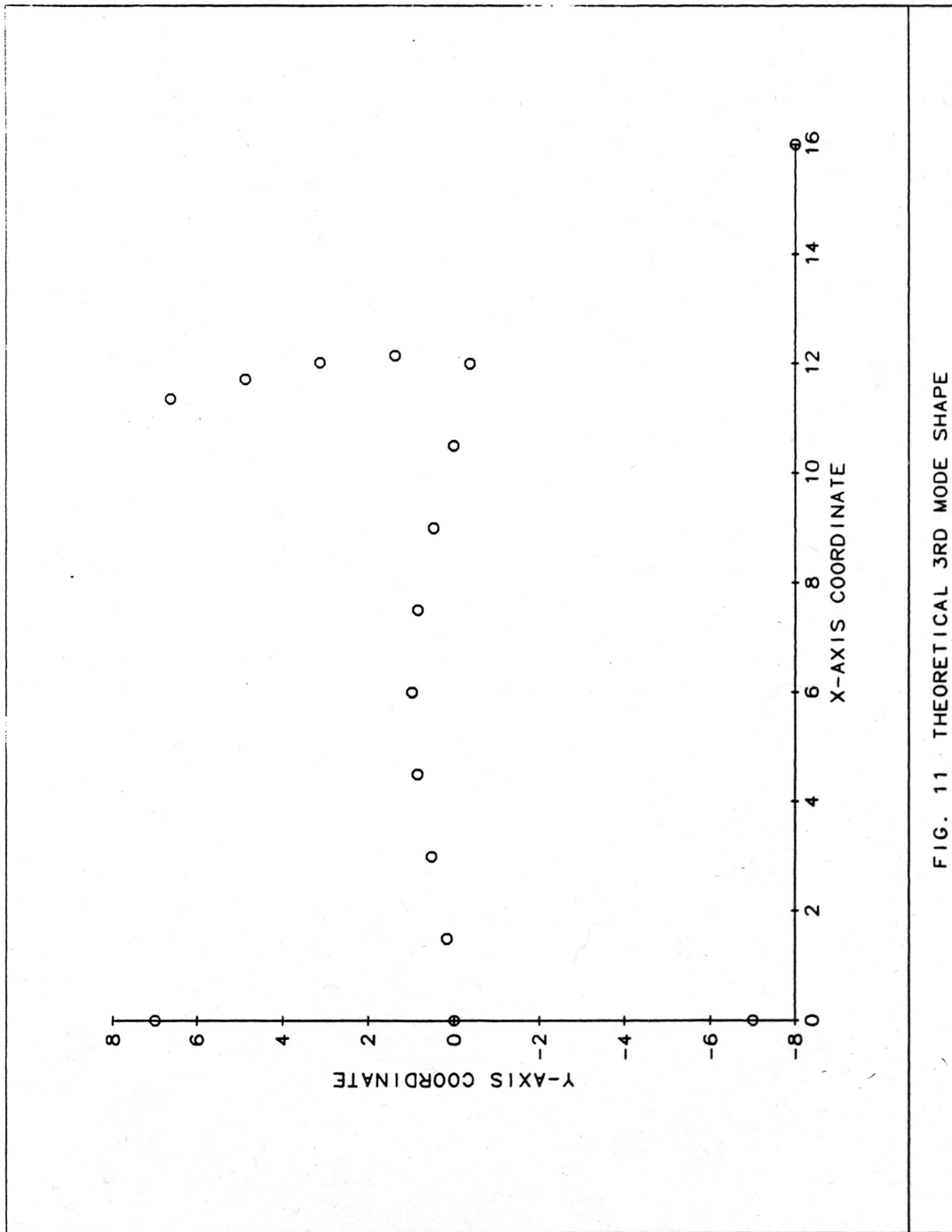


FIG. 11 THEORETICAL 3RD MODE SHAPE

Figures 12, 13, and 14 are plots of the experimentally determined mode shapes for the 1st, 2nd, and 3rd natural frequencies, respectively. Some obvious differences can be seen between the mode shapes predicted by the finite element model and those determined experimentally. On the whole, however, the general trends of the experimentally determined mode shapes were essentially correct.

Discussion of Results

The results obtained for the simple cantilevered beam showed a noticeable difference in analytically- and experimentally- determined natural frequencies. However, the analytical and experimental mode shapes exhibited remarkable similarity in their trends. Since the classical equations confirmed the finite element program results, there must have been some irregularities in the tested system. For example, the clamping method used to simulate a true cantilever may not have been completely adequate. Also, since the simple cantilever beam was 2" wide, the one-dimensional finite element theory used in the analysis may have been insufficient. Finally, the mass of the accelerometer was not modeled in the analytical system. Although the accelerometer's weight was admittedly small, its effect has been shown by a graduate student's work to change results by up to 4%.

On the other hand, the analytically- and experimentally- determined natural frequencies for the welded beams agreed closely, but the mode shapes did not correspond as well as those for the simple cantilevered beam. The test piece for the welded beam experiment was only $1\frac{1}{2}$ " wide, so modeling it with one-dimensional elements didn't introduce the error of modeling the 2" wide simple beam in the same manner. Also, the mass of the welded system was greater than that of the simple beam, so the effect on the system of not including the accelerometer's mass should have been smaller.

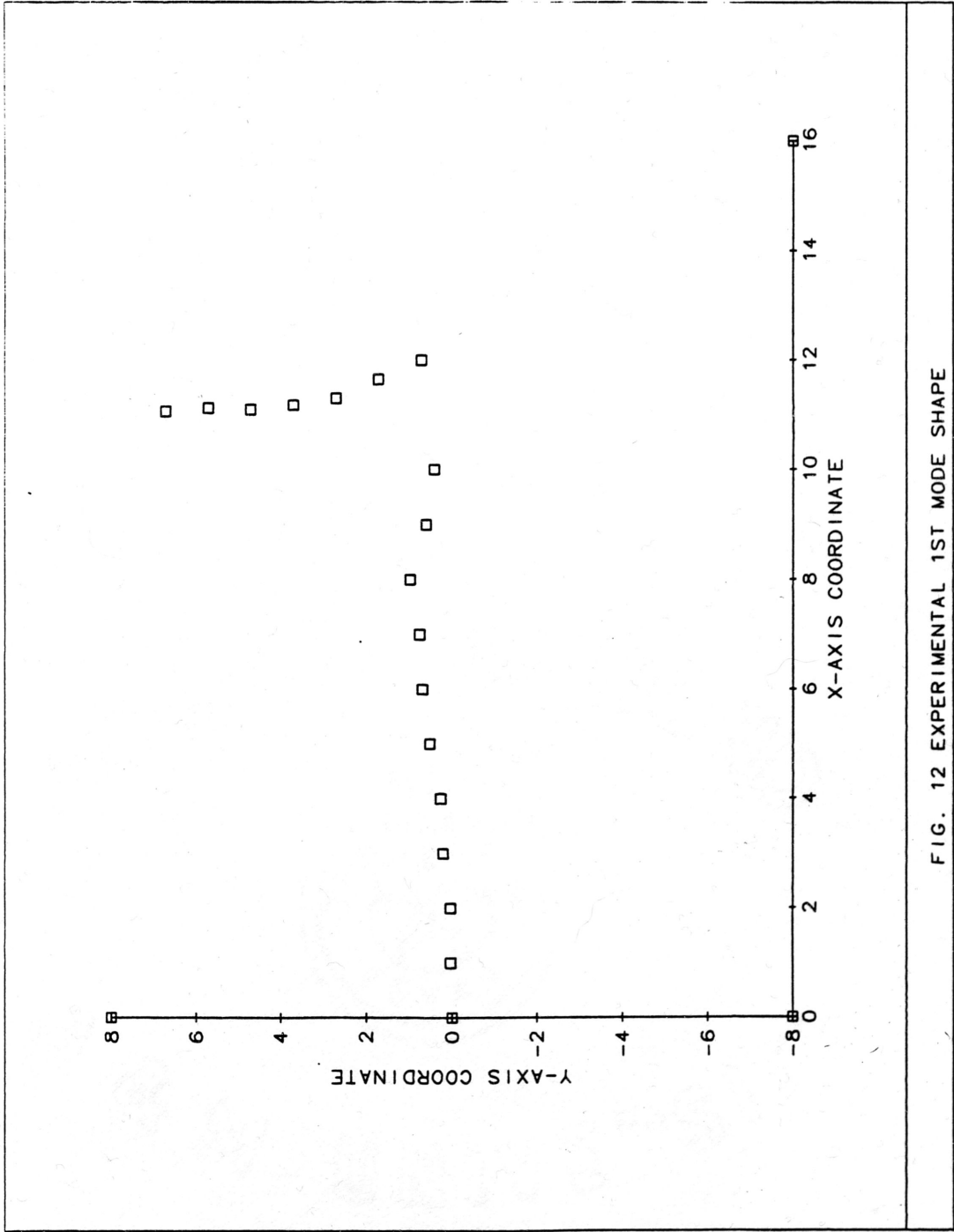


FIG. 12 EXPERIMENTAL 1ST MODE SHAPE

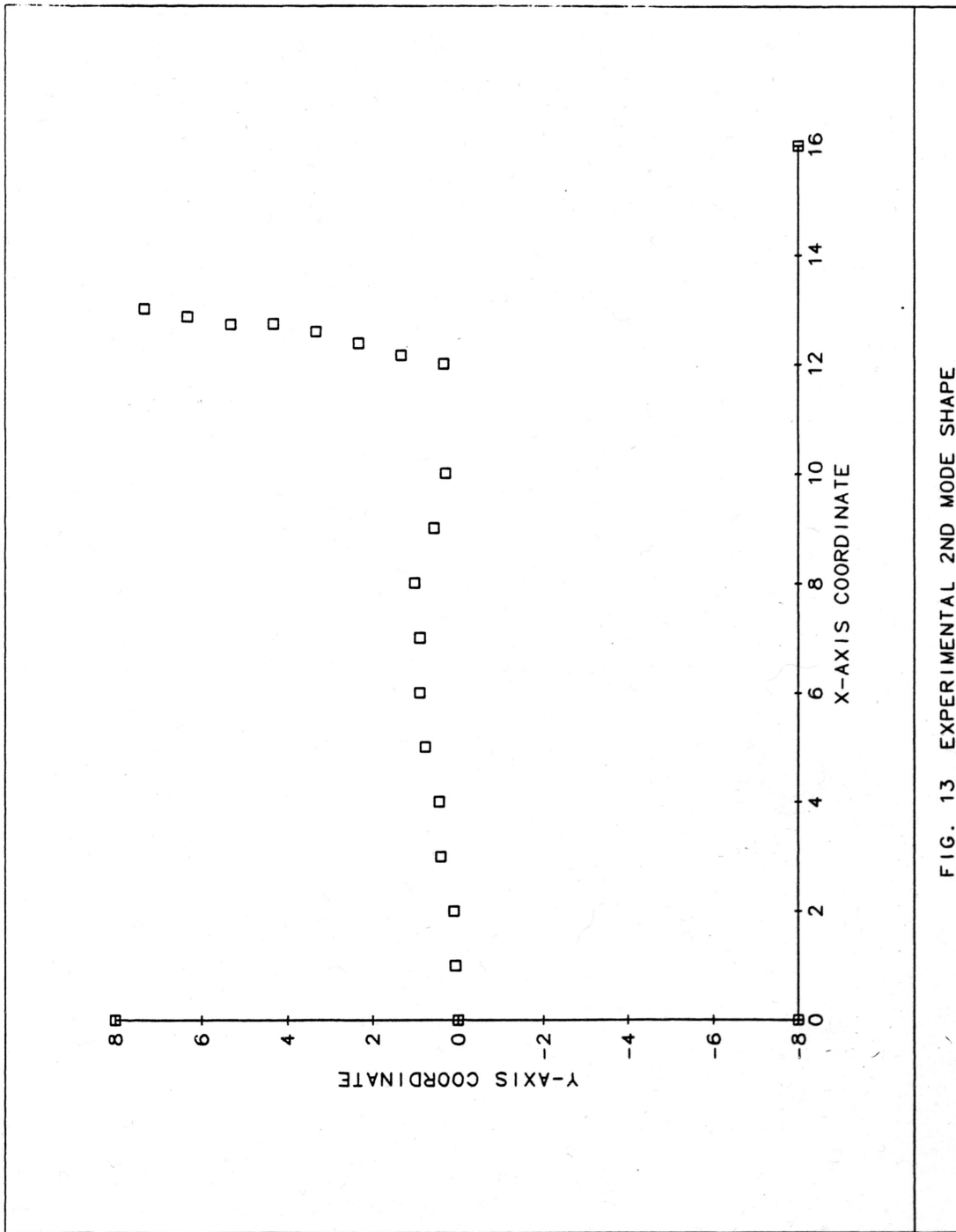


FIG. 13 EXPERIMENTAL 2ND MODE SHAPE

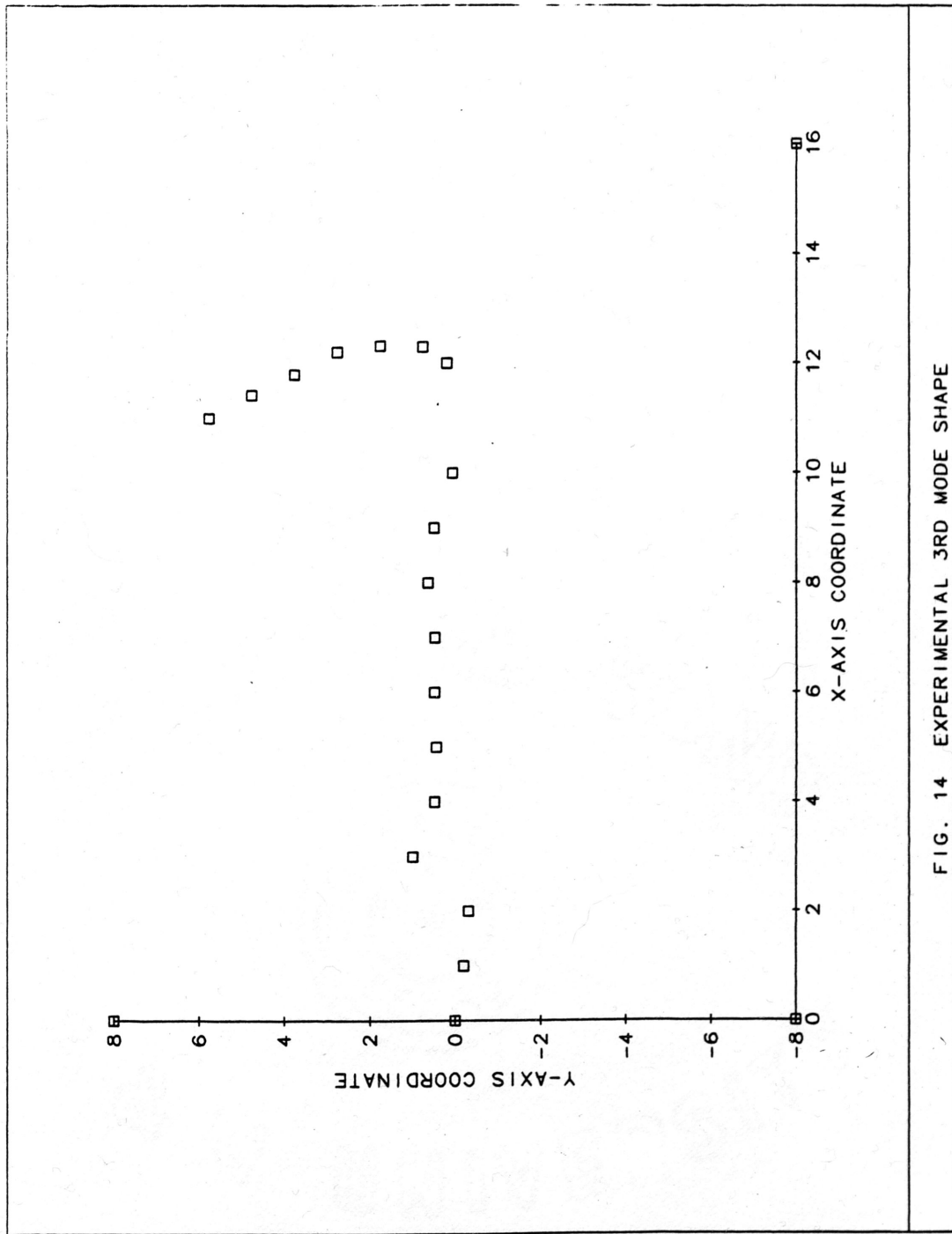


FIG. 14 EXPERIMENTAL 3RD MODE SHAPE

The experimentally determined mode shapes for the welded beam system exhibited some irregularities near the cantilever. This led to the conclusion that the method of achieving the cantilever was not entirely sufficient. Further experimentation needs to be done to determine what exactly would constitute an acceptable cantilever.

Conclusions

On the whole, considering actual data and data trends between the analytical model and the experimental test results, correct finite element modeling of structures can be used to accurately predict their natural frequencies and mode shapes. Experimental verification of analytical models of complicated structures may be difficult to obtain, but it is not impossible.

As far as remaining work that can be done in this area, the final version of the finite element program written for this project could be used to model more complex systems, up to and including planar trusses. The same experimental equipment needed to test the models in this research project could be used for a truss, but the testing technique may need to be modified.

While it is not economically feasible to construct and test every designed structure's dynamic response, as more analytical models are verified experimentally, engineers will get a better "feel" for constructing finite element models of their intended designs. More efficient analysis will ultimately lead to better and safer designs the first time around.

References

1. Childers, Donald G., ed., Modern Spectrum Analysis, an IEEE Publication, 1978.

2. Kardestuncer, H., ed., Unification of Finite Element Methods, a North Holland Mathematics Series Publication, 1984.

3. Schaeffer, Harry G., MSC/NASTRAN Static and Normal Modes Analysis, Schaeffer Analysis, Inc., 1979, pp. 305-343.

4. Wachspress, Eugene L., A Rational Finite Element Basis, Academic Press, Inc., 1975.

5. Armer, G.S.T., Garas, F.K., ed., Design for Dynamic Loading—The Use of Model Analysis, Construction Press, 1982.

6. Baker, Wilfred E., ed., Use of Models and Scaling in Shock and Vibration, an ASME Publication, 1963.

7. Dobbs, M.W., Nelson, R.B., "Parameter Identification of Large Structural Models—Concept and Reality," Modal Testing And Model Refinement, an ASME Publication, 1983, pp. 119-133.

8. Chen, J.C., Garba, J.A., "Structural Analysis Model Validation Using Modal Test Data," Combined Experimental/Analytical Modelling of Dynamic Structural Systems, an ASME Publication, June 1985, pp. 109-133.

9. Chen, J.C., Garba, J.A., Wada, B.K., "Modal Test and Analysis Correlation," Modal Testing And Model Refinement, an ASME Publication, 1983, pp. 85-99.

10. Craig, Roy R., Structural Dynamics—An Introduction to Computer Methods, John Wiley & Sons, 1981, pp.381-445.

11. Gerald, Curtis F., Applied Numerical Analysis, Second Edition, Addison-Wesley Publishing Company, 1978, pp.319-328.

APPENDIX A

**THEORETICAL CALCULATION FOR NATURAL
FREQUENCIES OF CANTILEVERED BEAM**

$$L = 14''$$

$$E = 27.6 \times 10^6 \text{ lb/in}^2$$

$$W = 2''$$

$$\gamma = .28 \text{ lb/in}^3$$

$$H = 1/8''$$

$$I = 1/12bh^3 = 3.2552 \times 10^{-4} \text{ in}^4$$

$$\omega_n = (\beta_n L)^2 \sqrt{\frac{EIg}{WL^3}}$$

Where $(\beta_n L)^2$ for a Cantilevered Beam

$$= 3.516 \quad (1\text{st Nat'l Freq})$$

$$= 22.04 \quad (2\text{nd Nat'l Freq})$$

$$= 61.7 \quad (3\text{rd Nat'l Freq})$$

$$\omega_n = (\beta_n L)^2 \sqrt{\frac{(27.6E6)(3.2252E - 4)(386)}{(14)(2)(.125)(.28)(14)^3}}$$

$$= 35.91(\beta_n L)^2$$

$$f_i = \frac{\omega_n}{2\pi} = \frac{35.91}{2\pi} (\beta_n L)^2 = 5.7153(\beta_n L)^2$$

$$f_1 = 20.1 \text{ Hz}$$

$$f_2 = 126.0 \text{ Hz}$$

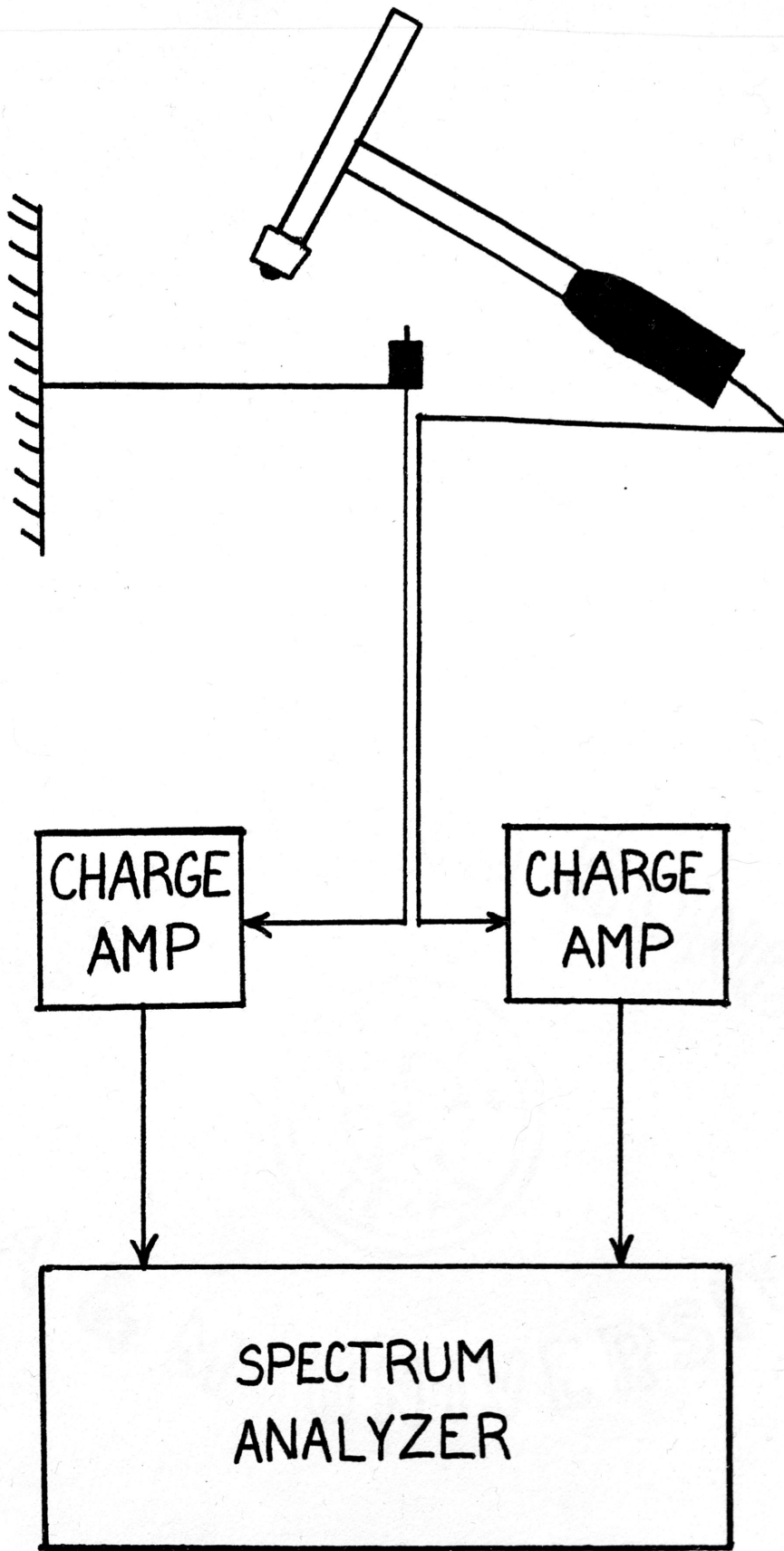
$$f_3 = 352.6 \text{ Hz}$$

APPENDIX B

**SENSITIVITY ANALYSIS FOR
CANTILEVERED BEAM**

<u>No. of Elements</u>	<u>1st Freq</u>	<u>2nd Freq</u>	<u>3rd Freq</u>
1	20.2	198.6	3834.8
2	20.0	126.8	428.9
3	20.1	126.2	356.4
4	19.9	125.9	354.8
7	20.1	125.9	352.5
14	19.9	125.7	352.1

APPENDIX C



APPENDIX D

Raw Data for Cantilevered Beam

<u>STATION</u>	<u>HZ</u>	<u>AMP</u>	<u>PHASE</u>	<u>COH</u>
1	16	8.8	118	.88
	108	74.1	-86	1.0
	288	37.5	106	.98
2	16	15.7	114	.87
	108	151	-94	1.0
	312	65.4	84	.97
3	16	28.5	123	.79
	108	253	-95	1.0
	296	96.8	114	.97
4	16	39.2	118	.88
	108	340	-93	1.0
	292	141	101	.90
5	16	63.5	123	.94
	108	336	-89	1.0
	296	93.7	126	.99
6	16	90.7	127	.94
	108	458	-86	1.0
	296	45.4	132	1.0
7	16	102	122	.87
	108	513	-83	1.0
	296	5.9	178	.78
8	16	136	121	1.0
	108	488	-79	1.0
	296	29.0	-86	.99
9	16	157	121	.99
	108	403	-78	1.0
	296	50.4	-74	.97
10	16	181	124	1.0
	108	253	-74	1.0
	296	66.7	-59	1.0

Raw Data for Cantilevered Beam, Cont.

<u>STATION</u>	<u>HZ</u>	<u>AMP</u>	<u>PHASE</u>	<u>COH</u>
11	16	206	125	1.0
	108	105	-58	1.0
	296	56.2	-59	.98
12	16	234	125	1.0
	108	87.9	88	.99
	296	40.6	-30	.99
13	16	248	125	1.0
	108	272	113	1.0
	296	30.3	24	1.0

**Comparison of Natural Frequencies
For Cantilevered Beam**

	As Predicted By			
	<u>FEM</u>	<u>Theory</u>	<u>EXP</u>	<u>% DIFF</u>
Cantilevered	19.9	20.1	16	19.6 %
(PI = .46497)	125.7	126.0	108	14.1 %
	352.1	352.6	296	15.9 %

Note: PI stands for performance index. Since it is less than 1.0, the performance of the eigenvector-eigenvalue routine is excellent.

APPENDIX E

**SENSITIVITY ANALYSIS FOR
WELDED BEAMS**

<u>No. of Elements</u>	<u>1st Freq</u>	<u>2nd Freq</u>	<u>3rd Freq</u>
1 / 1	13.6	49.2	270.6
2 / 1	13.7	49.0	167.8
2 / 2	13.5	49.1	167.8
4 / 7	12.9	48.9	166.5

The notation for number of elements is N_c / N_w where N_c is the number of elements the cantilevered part of the structure was modeled as having and N_w is the number of elements the welded part of the structure was modeled as having.

APPENDIX F

Raw Data for Welded Beams

<u>STATION</u>	<u>HZ</u>	<u>AMP</u>	<u>PHASE</u>	<u>COH</u>
1	12	17.6	-73	.83
	48	32.9	-92	.83
	156	30.4	148	.85
2	12	21.3	-70	.95
	48	52.2	-79	.98
	156	45.8	115	.93
3	12	98.5	-64	.93
	48	223	-58	.91
	156	152	-152	.85
4	12	125	-66	.98
	48	243	-65	.99
	156	74.1	-160	.96
5	12	239	-69	.95
	48	427	-64	.96
	154	68.1	-135	.95
6	12	318	-69	.91
	48	496	-62	.93
	154	75.1	-139	.99
7	12	347	-68	.99
	48	492	-66	.99
	154	74.3	-141	.98
8	12	443	-74	.97
	48	562	-62	.97
	154	99	-126	.96
9	12	280	-79	.85
	48	306	-91	.82
	154	77.2	-129	.76
10	12	195	-81	1.0
	48	155	-102	1.0
	154	12.5	-197	.99

Raw Data for Welded Beams Cont.

<u>STATION</u>	<u>HZ</u>	<u>AMP</u>	<u>PHASE</u>	<u>COH</u>
12	12	330	-82	.97
	48	175	-111	.92
	156	32.8	-121	.95
13	12	81.6	-71	.96
	48	131	105	.94
	156	12.3	-53	.94
14	12	137	-70	.96
	48	413	113	.95
	154	13	-7	.91
15	12	104	-70	.98
	48	496	108	.89
	154	8.41	-71	.87
16	12	124	-75	.99
	48	431	119	.96
	152	8.67	115	.97
17	12	139	-77	1.0
	48	421	109	.98
	152	24.0	121	.98
18	12	130	-82	.99
	48	555	81	.97
	152	41.6	101	1.0

**Comparison of Natural Frequencies
For Welded Beams**

	As Predicted By			
	<u>FEM</u>	<u>Theory</u>	<u>EXP</u>	<u>% DIFF</u>
Welded	12.8	--	12	6.3 %
(PI = .8202772)	48.9	--	48	1.8 %
	166.0	--	156	6.0 %

Note: PI stands for performance index. Since it is less than 1.0, the performance of the eigenvector-eigenvalue routine is excellent.

APPENDIX G

Finite Element Computer Program


```

C...
C...DYNAMIC FINITE ELEMENT ANALYSIS PROGRAM
C...
C...WRITTEN BY: ROBERT A. JUDGE
C...COURSE      : MEEN 485H
C...
C...DIMENSION ARRAYS AND DECLARE VARIABLES
C...
          DIMENSION ES(12,12,50),EM(12,12,50)
          DIMENSION GSTIFF(50,50),GMASS(50,50)
          COMMON E,G,IX,IY,POL,P
          REAL IX,IY,IZ,POL,L,AREA,M,ES,EM,GSTIFF,GMASS,X(25),Y(25)
C...
C...THIS SECTION ALLOWS THE USER TO INPUT THE NUMBER OF ELEMENTS
C...
          WRITE (6,100)
100      FORMAT ('$','PLEASE INPUT THE NUMBER OF ELEMENTS IN YOUR
          $ STRUCTURE=====>')
          READ (5,*)NEL
C...
C...THIS SECTION PROMPTS THE USER FOR THE MATERIAL PROPERTIES
C...
          WRITE (6,110)
110      FORMAT ('$','INPUT YOUNG'S MODULUS, E=')
          READ (5,*)E
          WRITE (6,120)
120      FORMAT ('$','INPUT MOMENT OF INERTIA, Iz=')
          READ (5,*)IZ
          WRITE (6,140)
140      FORMAT ('$','INPUT MATERIAL DENSITY, p=')
          READ (5,*)P
C...
C...THIS DO LOOP CALLS SUBROUTINE 'ELEMENT' TO GENERATE THE ELEMENTAL
C...STIFFNESS AND MASS MATRICES FOR ELEMENT "NEL"
C...
          DO 10 I=1,NEL
              WRITE (6,150)(I)
150          FORMAT('$','FOR ELEMENT ',I2,' INPUT NODAL COORDINATES X
          & Y')
              WRITE (6,155)I
155          FORMAT(' ',1ST NODE OF ELEMENT ',I2)
              READ(5,*)X(I),Y(I)
              WRITE (6,156)I
156          FORMAT(' ',2ND NODE OF ELEMENT ',I2)
              READ (5,*)X(I+1),Y(I+1)
              WRITE (6,160)(I)
160          FORMAT(' ',INPUT AREA OF ELEMENT ',I2)
              READ (5,*)AREA
              DX = X(I+1) - X(I)
              DY = Y(I+1) - Y(I)
              L = SQRT(DX*DX+DY*DY)
              CALL ELEMENT(IZ,L,M,AREA,I,ES,EM)
              CALL TRANSF(ES,EM,DX,DY,L,I)
10      CONTINUE

```

```

C...
C...THIS SECTION CALLS SUBROUTINE GLOBAL, WHICH USES A LOCATOR VECTOR
C...TO ASSEMBLE THE GLOBAL (SYSTEM) MASS AND STIFFNESS MATRICES.
C...
      CALL GLOBAL (NEL,GSTIFF,GMASS,ES,EM)
C...
C...
C...THIS SECTION CALLS SUBROUTINE EIGEN, WHICH WILL CALCULATE THE NATURAL
C...FREQUENCIES AND MODE SHAPES FOR THE GIVEN BEAM
C...
      CALL EIGEN (GSTIFF,GMASS)
      STOP
      END
C...
C...THIS IS SUBROUTINE 'TRANSF', WHICH WILL CALCULATE THE ANGLE BETWEEN
C...THE LOCAL ELEMENT COORDINATES AND THE GLOBAL COORDINATES, PLACE THE
C...VALUES IN A MATRIX, TRANPOSE THE MATRIX, PREMULTIPLY THE ELEMENTAL
C...MATRICES BY THE TRANPOSE OF THE COSINE MATRIX AND THEN POSTMULTIPLY
C...BY THE COSINE MATRIX.
C...
      SUBROUTINE TRANSF (ES,EM,DX,DY,L,I)
C...
C...DECLARATION OF VARIABLES
C...
      INTEGER A,B,C,D
      REAL ES(12,12,50),EM(12,12,50),ESTIFF(6,6),EMASS(6,6),
      $      TRANS(6,6),TRANSP(6,6),PRES(6,6),PREM(6,6)
      $      ,DX,DY,L,COSTH,SINTH
C...
C...THIS DO LOOP WILL ZERO OUT THE COSINE MATRIX
C...
      DO 10 A=1,6
          DO 20 B=1,6
              TRANS(A,B)=0.
              PRES(A,B)=0.
              PREM(A,B)=0.
              ESTIFF(A,B)=0.
              EMASS(A,B)=0.
          20      CONTINUE
      10      CONTINUE
C...
C...THIS SECTION FILLS IN THE COSINE MATRIX WITH THE APPROPRIATE
C...VALUES
C...
      COSTH=DX/L
      SINTH=DY/L
      TRANS(1,1)=COSTH
      TRANS(1,2)=SINTH
      TRANS(2,1)=-SINTH
      TRANS(2,2)=COSTH
      TRANS(3,3)=1.
      TRANS(4,4)=COSTH
      TRANS(4,5)=SINTH
      TRANS(5,4)=-SINTH
      TRANS(5,5)=COSTH

```

```

        TRANS(6,6)=1.
C...
C...THIS SECTION CREATES THE TRANSPOSE OF THE COSINE MATRIX 'TRANSP'
C...
        DO 30 C=1,6
            DO 40 D=1,6
                TRANSP(C,D)=TRANS(D,C)
            40         CONTINUE
        30         CONTINUE
C...
C...THIS SECTION PREMULTIPLIES THE ELEMENT MATRICES BY THE TRANSPOSE
C...OF THE COSINE MATRIX
C...
        DO 60 A=1,6
            DO 70 B=1,6
                DO 80 C=1,6
                    PRES(A,B)=PRES(A,B)+TRANSP(A,C)*ES(C,B,I)
                    PREM(A,B)=PREM(A,B)+TRANSP(A,C)*EM(C,B,I)
                80         CONTINUE
            70         CONTINUE
        60         CONTINUE
                WRITE (10,1000)I
                WRITE (10,1100)(PRES(A,B),B=1,6),A=1,6)
1000     FORMAT (////////' ,T35,'PREMULTIPLIED STIFFNESS MATRIX FOR
                SELEMENT ',I2)
1001     FORMAT (A)
1100     FORMAT (' ',T10,7X,6E10.2)
C...
C...THIS SECTION POSTMULTIPLIES THE 'PRE' MATRIX PRODUCTS BY THE COSINE
C...MATRIX
C...
        DO 90 A=1,6
            DO 100 B=1,6
                DO 110 C=1,6
                    ESTIFF(A,B)=ESTIFF(A,B)+PRES(A,C)*TRANS(C,B)
                    EMASS(A,B)=EMASS(A,B)+PREM(A,C)*TRANS(C,B)
                110         CONTINUE
            100         CONTINUE
        90         CONTINUE
                WRITE (10,1200)I
                WRITE (10,1300)(ESTIFF(A,B),B=1,6),A=1,6)
1300     FORMAT (' ',T10,7X,6E10.2)
1200     FORMAT (////////' ,T37,'POSTMULTIPLIED STIFFNESS MATRIX
                SFOR ELEMENT ',I2)
C...
C...THIS SECTION RENAMES ESTIFF AND EMASS BACK TO ES AND EM SO THEY CAN
C...BE PASSED BACK TO THE MAIN PROGRAM
C...
        DO 120 A=1,6
            DO 130 B=1,6
                ES(A,B,I)=ESTIFF(A,B)
                EM(A,B,I)=EMASS(A,B)
            130         CONTINUE
        120         CONTINUE
                RETURN

```

```

      END
C...
C...THIS SUBROUTINE WILL GENERATE THE ELEMENTAL STIFFNESS AND MASS
C...MATRICES FOR THE STRUCTURE BEING MODELED.
C...
      SUBROUTINE ELEMENT (MOMIZ,L,M,AREA,I,ES,EM)
C...
C...COMMON BLOCK AND DECLARATION OF VARIABLES
C...
      DIMENSION ES(12,12,50),EM(12,12,50)
      COMMON E,G,IX,IY,POL,P
      CHARACTER FLAG*3
      INTEGER A,B,C
      REAL MOMIZ,L,E,G,POL,ES,EM
C...
C...THIS SECTION FILLS IN THE STIFFNESS MATRIX WITH THE APPROPRIATE
C...CALCULATED VALUES.
C...
      ES(1,1,I)=E*AREA/L
      ES(2,2,I)=12*E*MOMIZ/(L**3)
      ES(3,3,I)=4*E*MOMIZ/L
      ES(4,4,I)=ES(1,1,I)
      ES(5,5,I)=ES(2,2,I)
      ES(6,6,I)=ES(3,3,I)
      ES(1,2,I)=0.
      ES(1,3,I)=0.
      ES(1,4,I)=-ES(1,1,I)
      ES(1,5,I)=0.
      ES(1,6,I)=0.
      ES(2,3,I)=6*E*MOMIZ/(L**2)
      ES(2,4,I)=0.
      ES(2,5,I)=-ES(2,2,I)
      ES(2,6,I)=ES(2,3,I)
      ES(3,4,I)=0.
      ES(3,5,I)=-ES(2,3,I)
      ES(3,6,I)=2*E*MOMIZ/L
      ES(4,5,I)=0.
      ES(4,6,I)=0.
      ES(5,6,I)=-ES(2,3,I)
C...
C...THIS SECTION FILLS IN THE MASS MATRIX WITH THE APPROPRIATE
C...CALCULATED VALUES.
C...
      EM(1,1,I)=P*AREA*L/3.
      EM(2,2,I)=13.*P*AREA*L/35.
      EM(3,3,I)=P*AREA*(L**3)/105.
      EM(4,4,I)=EM(1,1,I)
      EM(5,5,I)=EM(2,2,I)
      EM(6,6,I)=EM(3,3,I)
      EM(1,2,I)=0.
      EM(1,3,I)=0.
      EM(1,4,I)=P*AREA*L/6.
      EM(1,5,I)=0.
      EM(1,6,I)=0.
      EM(2,3,I)=22.*P*AREA*(L**2)/420.

```

```

EM(2,4,I)=0.
EM(2,5,I)=54.*P*AREA*L/420.
• EM(2,6,I)=-13.*P*AREA*(L**2)/420.
EM(3,4,I)=0.
EM(3,5,I)=-EM(2,6,I)
EM(3,6,I)=-P*AREA*(L**3)/140.
EM(4,5,I)=0.
EM(4,6,I)=0.
EM(5,6,I)=-EM(2,3,I)

C...
C...THIS SECTION FILLS IN THE LOWER TRIANGULAR PORTION OF BOTH THE MASS
C...AND STIFFNESS MATRICES.
C...
      DO 20 A=1,6
          DO 30 B=1,6
              ES(A,B,I)=ES(B,A,I)
              EM(A,B,I)=EM(B,A,I)
              IF (B.EQ.A) GOTO 20
          30      CONTINUE
      20      CONTINUE

C...
C...THIS SECTION ALLOWS THE USER TO PRINT OUT THE ELEMENTAL STIFFNESS AND
C...MASS MATRICES FOR ELEMENT NUMBER "I"
C...
      WRITE (6,*)'DO YOU WANT TO PRINT OUT THE STIFFNESS AND MASS
$ MATRICES FOR ELEMENT ',I
      READ (5,1001)FLAG
      IF (FLAG.EQ.'YES'.OR.FLAG.EQ.'yes') THEN
          WRITE (10,1000)I
          WRITE (10,1100)(ES(A,B,I),B=1,6),A=1,6)
          WRITE (10,1200)I
          WRITE (10,1300)(EM(A,B,I),B=1,6),A=1,6)
          RETURN
      ENDIF
1000  FORMAT (////////' ',T35,'ELEMENTAL STIFFNESS MATRIX FOR
SELEMENT ',I2)
1001  FORMAT (A)
1100  FORMAT (' ',T10,7X,6E10.2)
1300  FORMAT (' ',T10,7X,6E10.2)
1200  FORMAT (////////' ',T37,'ELEMENTAL MASS MATRIX FOR ELEMENT ',
SI2)
      RETURN
      END

C...
C...THIS IS SUBROUTINE GLOBAL, WHICH USES A LOCATOR VECTOR TO ASSEMBLE
C...THE GLOBAL (SYSTEM) MASS AND STIFFNESS MATRICES FROM THE ELEMENTAL
C...MATRICES. IT USES THE DIRECT ASSEMBLY METHOD.
C...
      SUBROUTINE GLOBAL (NEL,GSTIFF,GMASS,ES,EM)

C...
C...DIMENSION ARRAYS AND DECLARE VARIABLE TYPES
C...
      COMMON E,G,IX,IY,POL,P
      DIMENSION ES(12,12,50), EM(12,12,50), GSTIFF(50,50),
$          GMASS(50,50)

```

```

CHARACTER FLAG2*3
REAL GSTIFF,GMASS,ES,EM,E,G
INTEGER*4 N(6),A(6),GF,FF
DO GF=1,30
    DO FF=1,30
        GSTIFF(FF,GF)=0.
        GMASS(FF,GF)=0.
    ENDDO
ENDDO

```

C...

C...THIS LOOP WILL GET DIRECTON NO.'S FROM USER, DETERMINE ACTIVE
C...DEGREES OF FREEDOM,ESTABLISH THE LOCATOR VECTOR FOR EACH
C...ELEMENT, AND ASSEMBLE THE GLOBAL (SYSTEM) MATRICES

C...

```

DO 10 I=1,NEL
    WRITE (6,*)'INPUT DIRECTION NUMBERS FOR EACH NODE,
$ MAKING SURE YOU GIVE A DISPLACEMENT DIRECTION FIRST, FOLLOWED
$ BY A ROTATION, ETC.'
    WRITE (6,*)'1ST DIRECTION NO. FOR 1ST NODE OF ELEMENT',I
    READ (5,*)N(1)
    WRITE (6,*)'2ND DIRECTION NO. FOR 1ST NODE OF ELEMENT',I
    READ (5,*)N(2)
    WRITE (6,*)'ROTATION NO. FOR 1ST NODE OF ELEMENT',I
    READ (5,*)N(3)
    WRITE (6,*)'1ST DIRECTION NO. FOR 2ND NODE OF ELEMENT',I
    READ (5,*)N(4)
    WRITE (6,*)'2ND DIRECTION NO. FOR 2ND NODE OF ELEMENT',I
    READ (5,*)N(5)
    WRITE (6,*)'ROTATION NO. FOR 2ND NODE OF ELEMENT',I
    READ (5,*)N(6)
    DO 20 Q=1,6
        WRITE (6,*)'IS DIRECTION NUMBER',N(Q),'ACTIVE?'
        READ (5,1002)FLAG2
        FORMAT(A)
        IF (FLAG2.EQ.'NO'.OR.FLAG2.EQ.'no')THEN
            A(Q)=0
        ELSE
            A(Q)=N(Q)
        ENDIF
    CONTINUE
    DO 30 J=1,6
        DO 40 K=1,6
            NO=0
            DO 50 L=1,6
                IF(N(J).EQ.A(L))NO=NO+1
                IF(N(K).EQ.A(L))NO=NO+1
                IF(NO.EQ.2)THEN
                    GSTIFF(N(J),N(K))=GSTIFF(N(J),N(K))+
$SES(J,K,I)
                    GMASS(N(J),N(K))=GMASS(N(J),N(K))+
$SEM(J,K,I)
                NO=0
            ENDIF
        CONTINUE
    CONTINUE
40
50

```

```

30         CONTINUE
10        CONTINUE
          KOUNT=NEL*3+3
          DO 60 I=1,KOUNT
          DO 70 J=1,KOUNT
          WRITE(20,120) I,J,GSTIFF(I,J),I,J,GMASS(I,J)
120       FORMAT(' ',5X,'STIFF(',I2,',',I2,')= ',G15.7,
1         1      5X,'MASS(',I2,',',I2,')= ',G15.7)
          CONTINUE
60        CONTINUE
          RETURN
          END

C...
C...
C...
C
*****
C
*
C
*          THIS PROGRAM PROVIDES CALCULATION OF          *
C
*          FREQUENCIES AND MODE SHAPES.                    *
C
*
C
*****
C
-----
SUBROUTINE EIGEN (GSTIFF,GMASS)
COMPLEX EIGVAL,EIGVEC,ALPHA,SCALE
CHARACTER*20 ANS
DIMENSION GMASS(50,50),GSTIFF(50,50),EIGVAL(50),EIGVEC(50,50),
1         ALPHA(50),BETA(50),WK(5000)
REAL GMASS,GSTIFF

C...
C...
C...
-----
Begin data input phase.

LUNIT=6          ! DEFAULT OUTPUT TO TERMINAL
40        CONTINUE
          WRITE(6,50)
50        FORMAT('$','PLEASE INPUT THE NUMBER OF DEGREES OF FREEDOM, N= ')
          READ(5,*)N
          IF(N .LE. 50) GO TO 80
          WRITE(6,98)
          WRITE(6,99)
98        FORMAT('0',' ERROR!!! THE NUMBER OF DEGREES OF FREEDOM MUST BE')
99        FORMAT(' ';50 OR LESS, TRY AGAIN ')
          GOTO 40          ! REENTER DOF
80        CONTINUE

C...
C...INTERACTIVE PROMPTS
C...
          WRITE (6,150)
150       FORMAT ('1','SELECT OPTION:')
          WRITE (6,170)
170       FORMAT (' ',' 1)SET OUTPUT')
          WRITE (6,175)
175       FORMAT (' ',' 2)RUN')
          WRITE (6,180)
180       FORMAT ('$','TYPE OPTION #:')
          READ (5,*)OPT

```

```

IF (OPT.EQ.2)GOTO 600
IF(OPT.EQ.1) THEN
  WRITE(6,181)
181  FORMAT ('$','WRITE TO TERMINAL OR FILE?')
      READ (5,186) ANS
      IF(ICHAR(ANS).EQ.70) THEN
        LUNIT=10
        WRITE(6,186) ' '
        WRITE(6,186) ' *** OUTPUT SET TO MOD5.DAT ***'
        WRITE(6,186) ' '
      ELSE
        LUNIT=6
      END IF
      GOTO 500
    END IF
    GOTO 900      ! ABORT
C...
185  FORMAT('$','WRITE TO TERMINAL OR FILE?')
186  FORMAT(A)
C...
499  WRITE(6,999)
999  FORMAT(' INVALID INPUT, TRY AGAIN')
500  WRITE (6,186) ' HIT RETURN FOR MENU'
      READ (5,186,ERR=499)ANS
      GOTO 80
600  CONTINUE      ! RUN
-----
C...
C...      Determine the eigenvalues and vectors of:
C...      [GSTIFF] [X] = E [GMASS]
C...
C...      IJOB = 2 will cause routine to find eigenvalues,
C...      eigenvectors, and performance index.
C...
C...      IJOB=2
      CALL EIGZF(GSTIFF,50,GMASS,50,N,IJOB,ALPHA,BETA,EIGVEC,50,WK,IER)
C...
C...      If there is an error, EIGZF will print an error
C...      message.
C...
C...      Compute the eigenvalues.
C...
      DO 3 I=1,N
      IF(BETA(I).EQ.0.0)GOTO 3
      EIGVAL(I)=ALPHA(I)/BETA(I)
C...      The natural frequencies are the square root
C...      of the eigenvalues.
      EIGVAL(I)=CSQRT(EIGVAL(I))*3.122
3    CONTINUE
C...
C...      Normalize the eigenvectors (where possible) so
C...      that elements in row 1 of the modal
C...      column matrix are all 1.
C...
      DO 4 I=1,N
      SCALE=EIGVEC(1,I)

```



```

IF(SCALE.EQ.0.)GO TO 4
DO 5 J=1,N
EIGVEC(J,I)=EIGVEC(J,I)/SCALE
5 CONTINUE
4 CONTINUE
C...
C.....
C... Print out the results.
C... The standard output form for complex numbers is:
C... ( Real_part , Imaginary_part )
C...
IF(LUNIT.EQ.10) THEN
OPEN(10,FILE='MOD5.DAT',STATUS='NEW')
END IF
C...
C...
C...PRINT THE PERFORMANCE INDEX
C...
WRITE(LUNIT,200)WK(1)
200 FORMAT('0',/,',', THE PERFORMANCE INDEX IS ',G15.7,/)
C...
C...PRINT THE FREQUENCIES AND MODE SHAPES
C...
DO 6 I=1,N
WRITE(LUNIT,102)I,EIGVAL(I),EIGVEC(1,I)
102 FORMAT('0',/,',16X,'FREQUENCY ',I2,24X,'MODE SHAPE',/,
1 5X,('G15.7,',',G15.7,')',5X,('G15.7,',',G15.7,')')
DO 6 J=2,N
WRITE(LUNIT,103)EIGVEC(J,I)
103 FORMAT(43X,('G15.7,',',G15.7,')')
6 CONTINUE
C
CLOSE(10)
900 RETURN
END

```