MODAL SEISMIC ANALYSIS OF A NUCLEAR POWER PLANT CONTROL PANEL AND COMPARISON WITH SAP IV

by

M. R. Pamidi
Sargent and Lundy Engineers
Chicago, Illinois

and

P. R. Pamidi
Computer Sciences Corporation
Hampton, Virginia

SUMMARY

This paper deals with the application of NASTRAN to seismic analysis by considering the example of a nuclear power plant control panel. A modal analysis of a three-dimensional model of the panel, consisting of beam and quadrilateral membrane elements, is performed. Using the results of this analysis and a typical response spectrum of an earthquake, the seismic response of the structure is obtained. ALTERs required to the program in order to compute the maximum modal responses as well as the resultant response are given. The results are compared with those obtained by using the SAP IV computer program.

INTRODUCTION

Current government and industry regulations [References 1 and 2] require that the safety-related systems, structures and components of nuclear power plants be designed to withstand specified seismic excitations without loss of capability to perform their safety functions. This requirement is necessary in order to ensure

(a) continued operation of the reactor without undue risk to the health and safety of the public during an Operating Base Earthquake (OBE), and,

(b) shutdown of the reactor and its maintenance in a safe shutdown condition following a Safe Shutdown Earthquake (SSE).

The design of such equipment to withstand seismic disturbances involves dynamic analysis, testing or a combination of both. Seismic qualification by analysis alone is deemed sufficient provided the safety function of a structure or component is assured by its structural integrity. Thus, for instance, most mechanical equipment, such as heat exchangers, tanks, pressure vessels, etc., are usually qualified by analysis. Qualification by testing is recommended in those cases where functional operability is not necessarily assured by structural integrity. Thus, most electrical equipment, such as switchgears, motor control centers, control panels, etc., are usually qualified by testing. However, in many instances, testing is impractical either due to the size of the equipment involved or due to the prohibitive cost entailed by such testing. In such cases, a detailed dynamic analysis of the equipment, such as a control panel, is performed and the various associated electrical instrumentation and devices are then tested to acceleration levels determined by the analysis.

In this study, the application of NASTRAN to seismic analysis has been discussed by considering the example of a nuclear power plant control panel. A modal analysis of a three-dimensional model of the panel, consisting of beam and quadrilateral membrane elements, is performed. Using the results of this analysis and a typical response spectrum of an earthquake, the seismic response of the structure is obtained. The results are compared with those obtained by using the SAP IV computer program.

FORMULATION OF THE SEISMIC ANALYSIS PROBLEM

Obtaining the Modal Responses

The dynamic behavior of a system having multiple (n) degrees of freedom and subjected to seismic excitation is described by a set of differential equations represented by

$$[M]\,\{\ddot{u}\} + [C]\,\{\dot{u}\} + [K]\,\{u\} = -\,[M]\,[D]\,\{\ddot{u}_f\} \tag{1}$$

where [M], [C] and [K] are the (n x n) mass, damping and stiffness matrices respectively of the system; {u} is the (n x 1) displacement vector of interest; $\{\ddot{u}_f\}$ is a (3 x 1) vector that represents the time-dependent floor acceleration in the three component (X, Y and Z) directions; and [D] is an (n x 3) direction cosine matrix consisting of ones and zeroes that selects the masses that are involved in the motion in the three directions. The negative sign in this equation merely indicates that the effective load due to the seismic disturbance is opposite to that of the floor acceleration.

The displacement vector {u} in Eq. (1) may be expressed in terms of the normal coordinates as

$$\{u\} = \sum_{i=1}^{n} \{\phi_i\}\, w_i = [\phi]\,\{w\} \tag{2}$$

where {w} is an (n x 1) vector that represents the normal (or generalized) coordinates w_i and $[\phi]$ is an (n x n) matrix whose columns $\{\phi_i\}$ are the n eigenvectors of the free, undamped system given by

$$[M]\,\{\ddot{u}\} + [K]\,\{u\} = 0 \tag{3}$$

By using the orthogonal properties of the eigenvectors and utilizing the relationships that exist among the generalized mass, damping and stiffness quantities at any mode, Eqs. (1) and (2) can be combined to give a set of n uncoupled equations in the normal coordinates as follows [Ref. 3]:

$$\ddot{w}_i + 2\,\xi_i\,\omega_i \dot{w}_i + \omega_i^2\,w_i = -\,\{F_i\}^T\{\ddot{u}_f\},\ i = 1,\ 2,\ \ldots\ldots,\ n \tag{4}$$

where ξ_i is the damping ratio and ω_i is the natural frequency of the i^{th} mode. $\{F_i\}$ is a (3 x 1) vector whose elements represent the so-called modal participation factors in the three component directions for the i^{th} mode and is given by

$$\{F_i\}^T = \begin{Bmatrix} F_{ix} \\ F_{iy} \\ F_{iz} \end{Bmatrix}^T = \frac{\{\phi_i\}^T\,[M]\,[D]}{\{\phi_i\}^T\,[M]\,\{\phi_i\}} \tag{5}$$

The denominator on the right hand side of the above equation is the generalized mass for the i^{th} mode.

Equations (4) can be solved for significant modes by direct integration if the time-history of the floor acceleration $\{\ddot{u}_f\}$ is known. The solutions for the various significant modes can then be superimposed as per Eq. (2) to obtain the total solution. However, from a design point of view, it is simpler and often more convenient and economical to obtain the maximum displacements in any given mode by response spectrum analysis. This approach involves the use of design spectra derived from past earthquake data.

Let $\{S_{di}\} = \begin{Bmatrix} S_{dix} \\ S_{diy} \\ S_{diz} \end{Bmatrix}$ represent the spectral displacements for the i^{th} mode for excitations in the X, Y and Z directions. The corresponding spectral velocities $\{S_{vi}\}$ and spectral accelerations $\{S_{ai}\}$ are related to the spectral displacements by

$$\{S_{di}\} = \frac{\{S_{vi}\}}{\omega_i} = \frac{\{S_{ai}\}}{\omega_i^2} \tag{6}$$

The maximum response for the i^{th} mode due to the individual spectral displacements S_{dix}, S_{diy} and S_{diz} in the X, Y and Z directions respectively is given by [Ref. 3]:

$$\{u_i\}_{x_{max}} = \{\phi_i\} F_{ix} S_{dix} \tag{7a}$$

$$\{u_i\}_{y_{max}} = \{\phi_i\} F_{iy} S_{diy} \tag{7b}$$

$$\{u_i\}_{z_{max}} = \{\phi_i\} F_{iz} S_{diz} \tag{7c}$$

The maximum response for the i^{th} mode due to simultaneous seismic excitations in the three component directions is obtained by combining Eqs. (7a), (7b) and (7c) and is represented by[1]

$$\{u_i\}_{max} = \{u_i\}_{x_{max}} + \{u_i\}_{y_{max}} + \{u_i\}_{z_{max}} \tag{8}$$

or

$$\{u_i\}_{max} = \{\phi_i\} G_i \tag{9}$$

where G_i is a scalar quantity given by the product

$$G_i = \{F_i\}^T\{S_{di}\} \tag{10}$$

Eq. (9), which gives the maximum response for the i^{th} mode, can be generalized to give the maximum response for any of m modes ($1 \leq m \leq n$) by the single matrix equation

$$[u']_{max} = [\phi'] \text{ diag}\left[[F']^T[S_d']\right] \tag{11}$$

where

$$[u']_{max} = [\{u_1\}_{max} \{u_2\}_{max} \cdots\cdots \{u_m\}_{max}] \tag{12a}$$

---

[1] Eq. (8) gives a conservative estimate for the maximum response for the i^{th} mode. It is acceptable to the regulatory authorities to compute this response by taking the square root of the sum of the squares of the maximum responses in the three component directions [Ref. 4].

$$[\phi'] = [\{\phi_1\}\ \{\phi_2\}\ \ldots\ldots\ \{\phi_m\}] \tag{12b}$$

$$[F'] = [\{F_1\}\ \{F_2\}\ \ldots\ldots\ \{F_m\}] \tag{12c}$$

and

$$[S_d'] = [\{S_{d1}\}\ \{S_{d2}\}\ \ldots\ldots\ \{S_{dm}\}] \tag{12d}$$

Eq. (11) can be rewritten as

$$[u']_{max} = [\phi']\,[G'] \tag{13}$$

where $[G'] = \text{diag}[F']^T[S_d'] = \begin{bmatrix} G_1 & 0 & \ldots\ldots & 0 \\ 0 & G_2 & \ldots\ldots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \ldots\ldots & G_m \end{bmatrix}$ (14)

The elements of the (m x m) diagonal matrix [G'] are given by Eq. (10).

Combining the Modal Responses

The total response of the system can be obtained by combining the maximum responses of the individual modes involved as given by Eq. (13). The exact manner in which these modal responses are combined is, however, a matter of judgment and there is no one way for obtaining the total response. When the modes are not closely spaced,[2] current regulatory practice [Ref. 4] requires that the resultant response (whether it be displacement, stress or other quantity) be obtained by taking the square root of the sum of the squares (SRSS) of the corresponding maximum responses for the individual modes involved. When the modes are closely spaced, it is required that the total response be computed by any one of three methods acceptable to the regulatory authorities. Referred to as the Grouping Method, the Ten Percent Method and the Double Sum Method, the details of these methods are given in Reference 4.

---

[2] Two modes are said to be closely spaced if their frequencies differ from each other by 10% or less of the lower frequency.

ANALYSIS

Analysis by NASTRAN Program

NASTRAN cannot directly perform seismic analysis, but can be adapted for the purpose. The procedure employed here corresponds to the theoretical development presented in the previous section. The method essentially involves two major steps as indicated below[3]:

Step 1. Perform a normal mode analysis using Rigid Format 3 and obtain the significant modes.[4]

Step 2. Once the significant natural frequencies are known, input the appropriate spectral displacement matrix $[S_d']$ [see Eq. (12d)] and the direction cosine matrix [D] [see Eq. (1)] via DMI bulk data cards and repeat the normal mode analysis using the following ALTERs to Rigid Format 3 (Level 16) and employing NØRM = MASS (normalization to unit value of the generalized mass) on the EIGR bulk data card [Ref. 7]:

```
ALTER 106 $
MPYAD     PHIA,MAA,/DUMMYA/C,N,1 $
MPYAD     DUMMYA,DIRCØS,/MØDEPF/C,N,0 $
MPYAD     MØDEPF,SEISMIC,/DUMMYB/C,N,0 $
DIAGØNAL  DUMMYB/DUMMYC/C,N,SQUARE $
MPYAD     PHIA,DUMMYC,/PHIASS/C,N,0 $
ALTER 108 $
SDR1      USET,,PHIASS,,,GØ,GM,,KFS,,/PHIG,,QG/C,N,1/C,N,REIG $
ALTER 121 $
TRNSP     PHIASS/PHIASST $
MPYAD     PHIASS,PHIASST,/DUMMYD/C,N,0 $
DIAGØNAL  DUMMYD/SRSSDISP/C,N,CØLUMN/C,N,0.5 $
MATPRN    SRSSDISP,,,,// $
ENDALTER  $
```

---

[3] The adaptation of NASTRAN for seismic analysis has also been discussed by other users [Refs. 5 and 6].

[4] For seismic analysis purposes, a mode is considered significant if its frequency is less than or equal to 33 Hz.

The data blocks used in the above ALTER package have the following correspondences to the matrices mentioned in the previous section:

$$\begin{aligned}
&\text{PHIA} && \equiv [\phi'] \\
&\text{MAA} && \equiv [M] \\
&\text{DIRCØS} && \equiv [D] \\
&\text{MØDEPF} && \equiv [F']^T \\
&\text{SEISMIC} && \equiv [S'_d] \\
&\text{DUMMYC} && \equiv [G'] \\
&\text{PHIASS} && \equiv [u']_{max}
\end{aligned}$$

The so-called eigenvectors printed out by the above analysis actually represent the maximum modal responses (displacements) as given by Eq. (13). The stresses obtained correspond to these displacements.

The output data block SRSSDISP in the above ALTER package is an (n x 1) vector that represents the resultant response obtained by taking the square root of the sum of the squares (SRSS) of the maximum modal displacements. The corresponding SRSS stresses can be obtained by combining the individual modal stresses.

Analysis by SAP IV Program

SAP IV can perform seismic analysis directly without the need for a separate intermediate run just to obtain the significant modes. The proportions of the seismic excitations in the three directions are specified and the spectral information (displacements or accelerations) is input as a table of spectral values versus period. The maximum modal displacements and the resultant (SRSS) displacements and stresses are automatically output. The details of the method are explained fully in Reference 8.

FINITE ELEMENT MODEL

The basic details of the finite element model of the control panel considered are shown in Figures 1, 2 and 3. Beam elements of three different cross sections and quadrilateral membrane elements of two different thicknesses are used to model the structure. The beam elements are represented by the CBAR elements in NASTRAN and by the three-dimensional beam elements (element type 2) in SAP IV; the membrane elements are represented by the CQDMEM elements in NASTRAN and by the plane stress quadrilateral membrane elements (element type 3) in SAP IV. A total of 265 active degrees of freedom are involved in the analysis. The complete details of the model can be obtained from the authors.

RESULTS

Using essentially identical input data, the finite element model described above was analyzed by both NASTRAN and SAP IV programs following the procedure outlined earlier. Seismic excitations of equal magnitude in the three component directions were assumed. The spectral data used was based on the El Centro (Calif.) earthquake of 1940. A damping of 2% (see Reference 9 for guidelines in this regard) was assumed.

The results of the analysis are presented in Tables 1, 2 and 3. Table 1 lists the significant natural frequencies obtained by the two programs. Table 2 gives the corresponding modal participation factors. Table 3 shows some representative resultant (SRSS) displacements of significant magnitude.

It can be seen from Table 1 that the agreement between the significant natural frequencies obtained by the two programs is excellent. The modal participation factors given in Table 2 agree well too, except when their magnitudes are small; this is due to small differences that exist in the various eigenvector components (not shown) obtained by the two programs. The representative resultant (SRSS) displacements of significant magnitude shown in Table 3 also agree well, but the same is not true when they are of smaller magnitude. This is due not only to the small differences in the modal participation factors involved, but also to the different manner in which the spectral data is input to the two programs. The agreement in the results, on the whole, is quite good.

SUMMARY AND CONCLUSIONS

The application of NASTRAN to seismic analysis has been discussed by considering the example of a nuclear power plant control panel. A modal analysis of a three-dimensional model of the panel, consisting of beam and quadrilateral membrane elements, is performed. Using the results of this analysis and a typical response spectrum of an earthquake, the seismic response of the structure is obtained. ALTERs required to the program in order to compute the maximum modal responses as well as the resultant (SRSS) response are given. The results are compared with those obtained by using the SAP IV program. The agreement, on the whole, is quite satisfactory.

The paper demonstrates the adaptability and suitability of NASTRAN for seismic analysis. The greater choice of elements offered by NASTRAN as well as the availability of such desirable features as the CNGRNT capability (which can result in significant reductions in running times particularly for large problems) [Ref. 7] and Guyan reduction [Ref. 10] make this versatile program an attractive tool for seismic analysis of large structures.

REFERENCES

1. Regulatory Guide 1.29, Seismic Design Classification, U. S. Atomic Energy Commission, Directorate of Regulatory Standards, Washington, D. C., Revision 1, August 1973.

2. IEEE STD 344-1975, IEEE Recommended Practices for Seismic Qualification of Class 1E Equipment for Nuclear Power Generating Stations, IEEE, New York, 1975.

3. Wiegel, R. L., (ed.), Earthquake Engineering, Prentice-Hall, Englewood Cliffs, N. J., 1970.

4. Regulatory Guide 1.92, Combining Modal Responses and Spatial Components in Seismic Response Analysis, U. S. Nuclear Regulatory Commission, Office of Standards Development, Washington, D. C., Revision 1, February 1976.

5. Go, J. C., "Seismic Analysis of Nuclear Power Plant Structures", NASTRAN: Users' Experiences, NASA TM X-2893, September 1973, pp. 243-249.

6. Setlur, A. V. and Valathur, M., "Some Studies on the Use of NASTRAN for Nuclear Power Plant Structural Analysis and Design", NASTRAN: Users' Experiences, NASA TM X-2893, September 1973, pp. 585-594.

7. The NASTRAN User's Manual, (Level 16.0), NASA SP-222(03), March 1976.

8. Bathe, K. J., Wilson, E. L., and Peterson, F. E., SAP IV: A Structural Analysis Program for Static and Dynamic Response of Linear Systems, University of California, Earthquake Engineering Research Center, Berkeley, California, June 1973.

9. Regulatory Guide 1.61, Damping Values for Seismic Design of Nuclear Power Plants, U. S. Atomic Energy Commission, Directorate of Regulatory Standards, Washington, D. C., October 1973.

10. The NASTRAN Theoretical Manual, (Level 16.0), NASA SP-221(03), March 1976.

Table 1. Natural Frequencies of Significant Modes

Mode no.	Natural frequencies (Hz)	
	NASTRAN results	SAP IV results
1	0.6352E+01	0.6352E+01
2	0.6563E+01	0.6563E+01
3	0.6699E+01	0.6700E+01
4	0.1013E+02	0.1013E+02
5	0.1180E+02	0.1180E+02
6	0.1401E+02	0.1401E+02
7	0.1648E+02	0.1649E+02
8	0.1652E+02	0.1652E+02
9	0.1665E+02	0.1665E+02
10	0.1706E+02	0.1706E+02
11	0.2371E+02	0.2370E+02
12	0.2578E+02	0.2579E+02
13	0.3082E+02	0.3082E+02

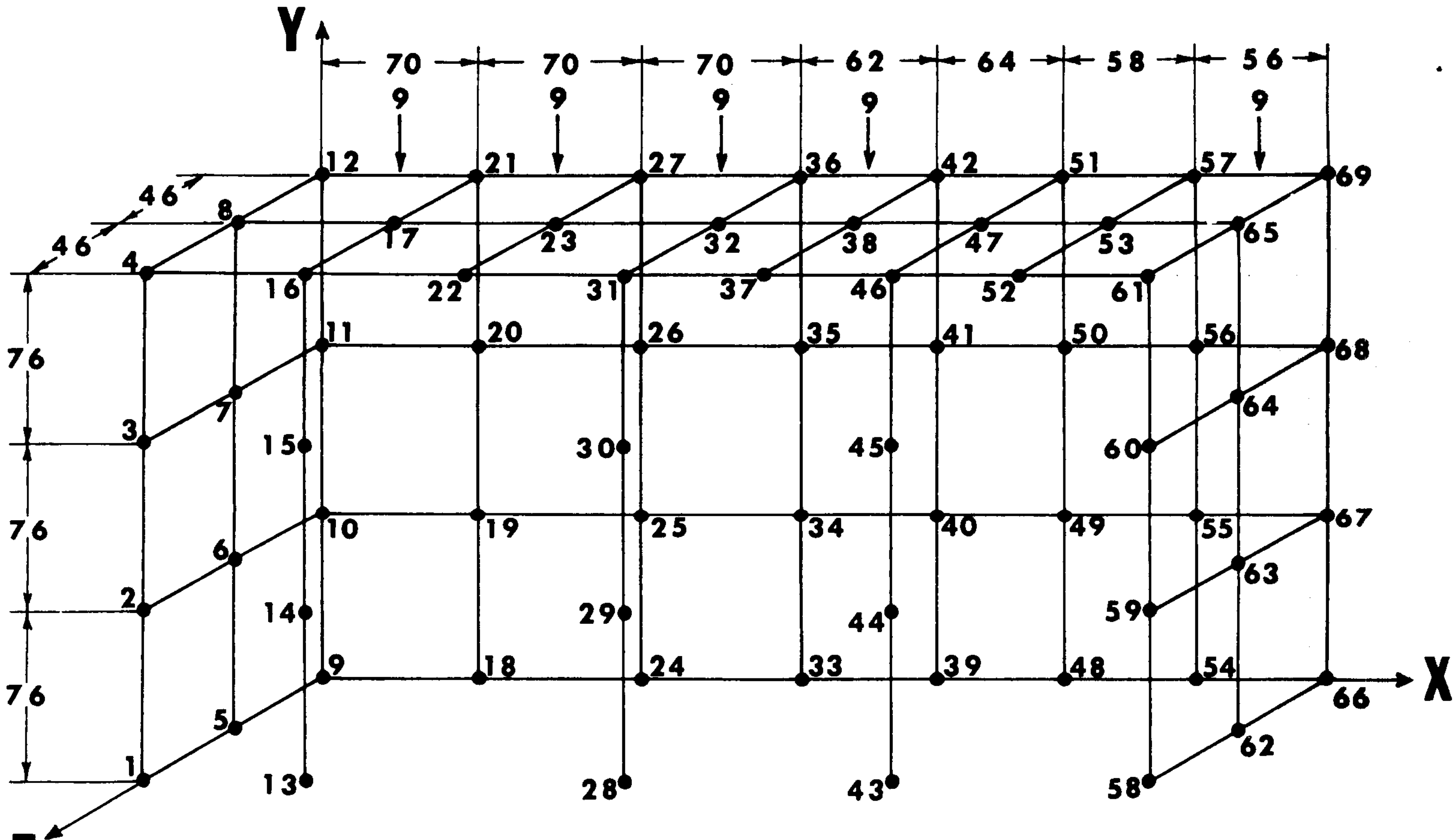
Table 2. Modal Participation Factors for Significant Modes

Mode no.	Modal participation factors					
	NASTRAN results			SAP IV results		
	X-direction	Y-direction	Z-direction	X-direction	Y-direction	Z-direction
1	0.4692E-04	0.2313E-01	0.6498E+00	0.4879E-04	0.2316E-01	0.6524E+00
2	0.1033E-03	-0.1002E-01	-0.4356E+00	0.1305E-03	-0.9960E-02	-0.4348E+00
3	0.1467E-03	0.1909E-01	0.5135E+00	0.1679E-03	0.1902E-01	0.5128E+00
4	-0.2529E-03	-0.1750E-01	0.4024E+00	-0.3235E-03	-0.1755E-01	0.4035E+00
5	0.3132E+00	0.1331E-02	0.3729E-03	0.3132E+00	0.1340E-02	0.4287E-03
6	-0.1957E-03	0.5041E-01	-0.2872E+00	-0.1730E-03	0.5043E-01	-0.2885E+00
7	0.2507E+00	-0.6438E-02	0.3732E-03	0.2507E+00	-0.6425E-02	0.2668E-03
8	-0.1324E-02	-0.2445E-01	0.1439E-01	-0.1446E-02	-0.2465E-01	0.1291E-01
9	0.1969E-02	0.3657E-01	-0.5804E-01	0.1849E-02	0.3639E-01	-0.5702E-01
10	-0.7096E-03	0.1421E-01	-0.3724E-01	-0.7052E-03	0.1409E-01	-0.3580E-01
11	-0.3842E+00	0.2454E-01	-0.2009E-02	-0.3844E+00	0.2452E-01	-0.2307E-02
12	-0.5905E-03	-0.1363E+00	0.1533E+00	-0.6860E-03	-0.1363E+00	0.1564E+00
13	-0.1407E+00	-0.2246E-01	-0.2920E-02	-0.1407E+00	-0.2247E-01	-0.3697E-02

Table 3. Representative Values of Resultant (SRSS) Displacements

Grid point no.	Direction of motion	Resultant (SRSS) displacements (cm.)	
		NASTRAN results	SAP IV results
19	Z	0.53861E+00	0.54059E+00
20	Z	0.65093E+00	0.65283E+00
30	X	0.16941E+00	0.17834E+00
34	Z	0.67732E+00	0.68189E+00
35	Z	0.85773E+00	0.86271E+00
45	X	0.26523E+00	0.26662E+00
49	Z	0.44626E+00	0.44775E+00
50	Z	0.56609E+00	0.56746E+00

FINITE ELEMENT MODEL
(Numbers Designate Nodes)



Dimensions in cm   Loads in kg

FIGURE 1

FINITE ELEMENT MODEL
(Numbers Designate Beam Elements)

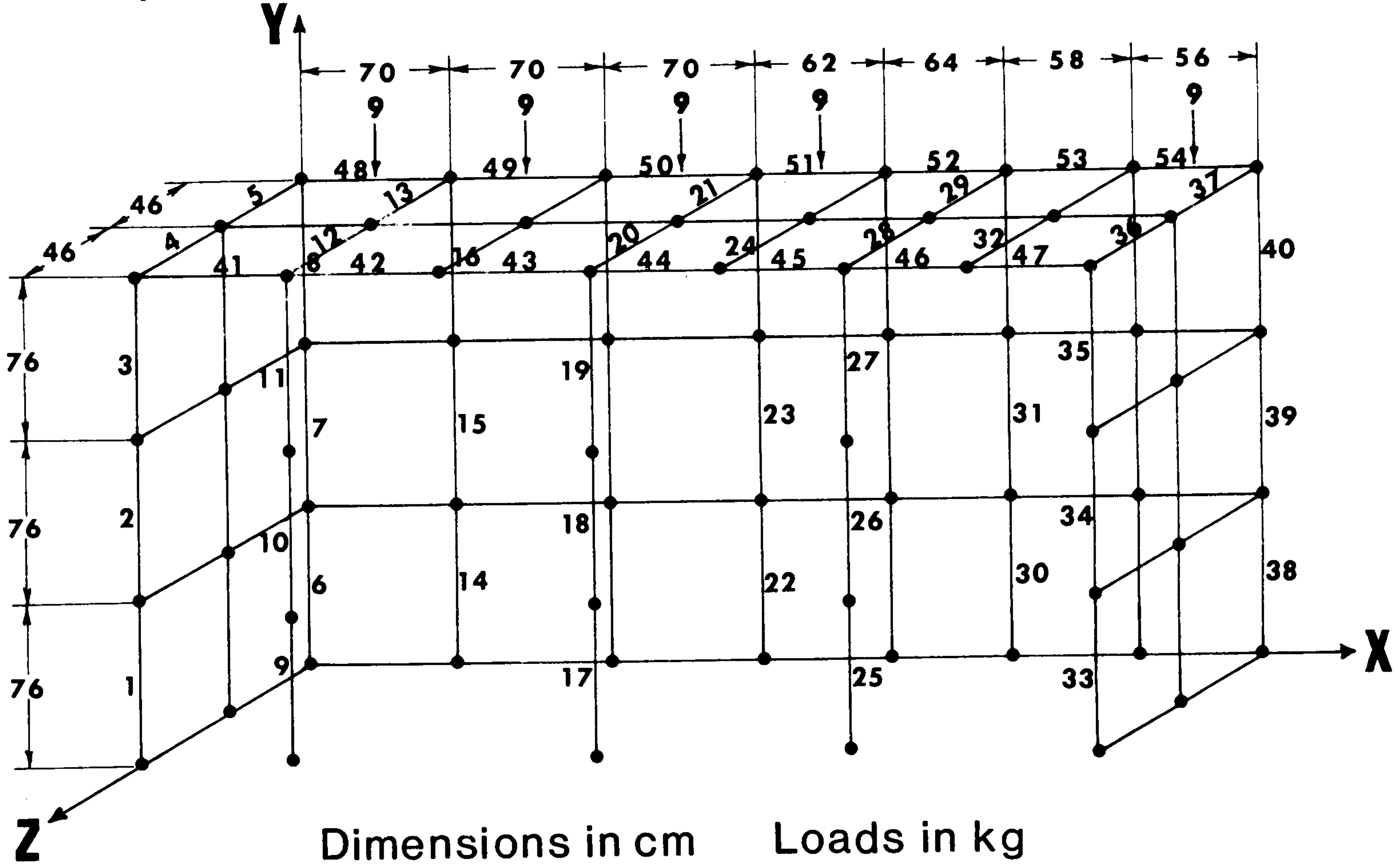
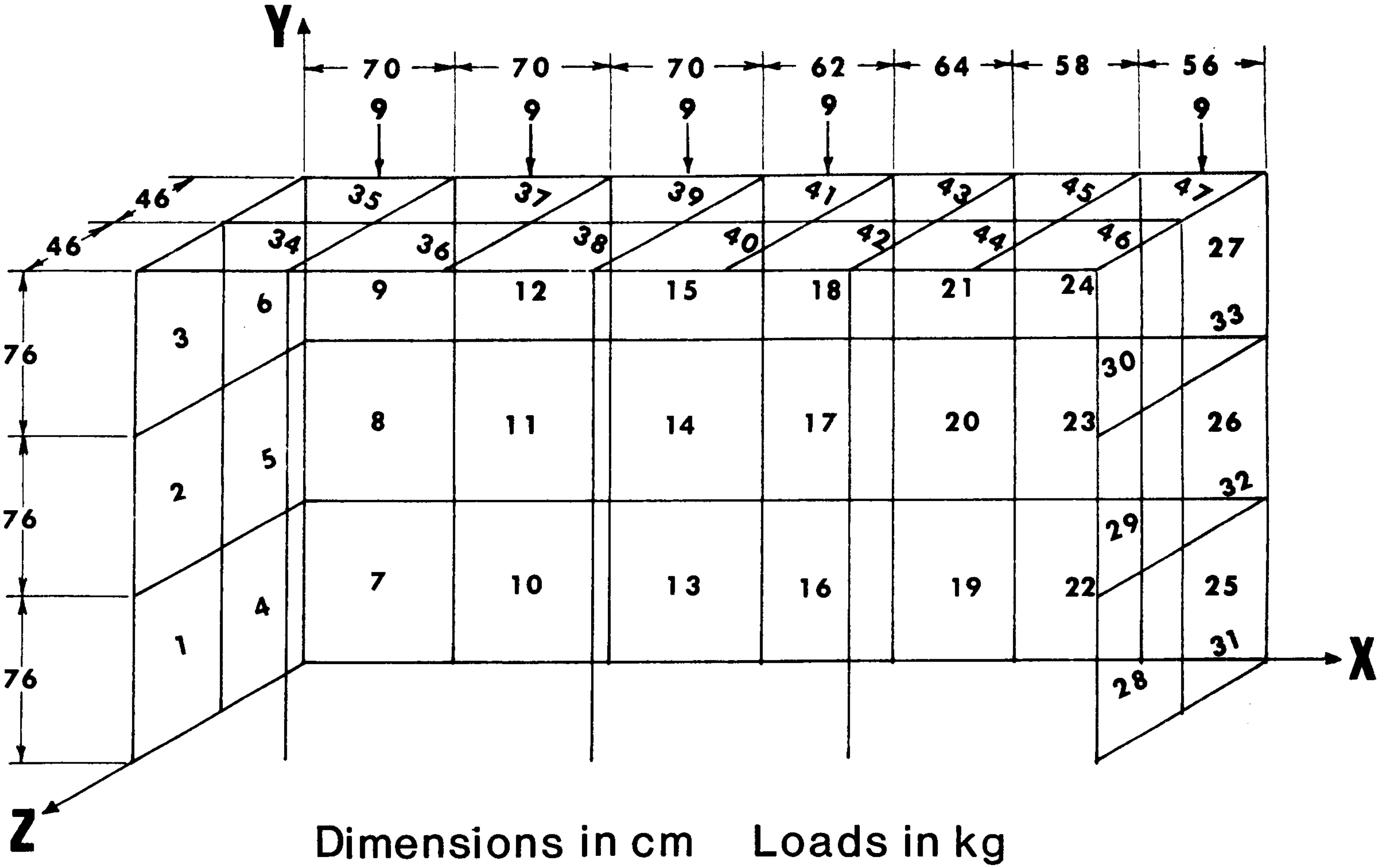


FIGURE 2

FINITE ELEMENT MODEL
(Numbers Designate Membrane Elements)



Dimensions in cm   Loads in kg

FIGURE 3