EARTHQUAKE ANALYSIS OF A NUCLEAR POWER STATION TURBINE BUILDING

L.G. Jaeger and A.A. Mufti

SYNOPSIS

The paper discusses the response spectrum approach to the seismic analysis of structures. The application of the method to the dynamic analysis of a nuclear power station turbine building is described, the building being represented by a 3-D finite element model. A method developed by the authors to represent a complex structure by a much simpler one having the same dynamic characteristics of dominant modes is also presented, as an economical alternative for the seismic analysis of complex structures.

RESUME

Cette communication traite de la méthode spectrale pour l'évaluation sismique des bâtiments d'une centrale nucléaire. La structure est idéalisée par éléments finis en un modèle à trois dimensions. Une seconde méthode, développée par les auteurs, pour étudier le caractère dynamique des structures complexes est présentée comme alternative pour les analyses sismiques compliquées. Cette dernière méthode possède l'avantage de permettre une idéalisation plus simple de la structure sans pour autant modifier son caractère dynamique.

Dr. Leslie G. Jaeger obtained his Ph.D. from the University of London. He is currently Professor of Engineering and Vice-President, Academic, Acadia University, Wolfville, Nova Scotia, Canada.

Dr. Aftab A. Mufti obtained his Ph.D. from McGill University. He is currently Professor and Dean, School of Computer Science and Engineering at Acadia University, Wolfville, Nova Scotia, Canada.

Introduction

It is well known that the advent of the computer has caused dramatic changes in the methods and procedures used by consulting engineers in analysing important structures such as nuclear power plants and high-rise buildings. This change has obviously been beneficial to all concerned, leading for example to the high standards required by the National Building Code and Atomic Energy of Canada Limited (2, 7).

It is the main purpose of this paper to describe the application of the response spectrum method of earthquake analysis to complex structures.

In an earthquake analysis we may choose one of the following methods:

a) National Building Code-static analysis.

b) Natural frequency analysis (eigenvalue analysis) followed by response spectrum analysis.

c) Natural frequency analysis (eigenvalue analysis) followed by response history analysis.

d) Response history analysis by direct integration.

The choice of analysis from these four depends upon the importance of the structures and the code requirements. Many building codes around the world accept the procedure of a natural frequency analysis followed by response spectrum analysis. To illustrate the use of the response spectrum, a case study of the Point Lepreau nuclear power station turbine building is described. The analysis of the turbine building is performed on a complete 3-D finite element structural model of it. A method is developed for reducing the complete 3-D finite element structural model to the simplified "STICK" model in such a way that the latter can be used as an engineering approximation of the structure for dynamic analysis. The "STICK" model approach allows practising engineers an economical method to do dynamic analysis of a complex

3-D structure.

NATURAL FREQUENCY ANALYSIS (EIGENVALUE ANALYSIS)
FOLLOWED BY RESPONSE SPECTRUM ANALYSIS

The general equations of motion for base excitation are:

[1] $[M]\{\ddot{u}\} + [K]\{u\} + [C]\{\dot{u}\} = -\ddot{y}_g\{M\}$

where

$[M]$ = diagonal mass matrix

$[K]$ = symmetric stiffness matrix

$[C]$ = viscous damping matrix

$\{u\}$ = column vector of displacements

$\{\dot{u}\}$ = column vector of velocities

$\{\ddot{u}\}$ = column vector of accelerations

$\ddot{y}_g$ = ground accelerations

Prior to performing the actual dynamic analysis for large buildings it is highly desirable to eliminate the zero mass terms that might exist on the diagonal of the mass matrix [M]. This is the case when the masses are lumped at only a few nodes. The elimination procedure, usually known as "static condensation", is as follows:-

The complete set of equations is of the form

[2] $([K] - \omega_r^2[M])\{a^r\} = \{0\}$

The [K] matrix, the [M] matrix and the $\{a^r\}$ vector are partitioned as follows:

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} a_1^r \\ a_2^r \end{Bmatrix} - \omega_r^2 \begin{bmatrix} 0 & 0 \\ 0 & M_{22} \end{bmatrix} \begin{Bmatrix} a_1^r \\ a_2^r \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

whence

[3] $([K_{22}^*] - \omega_r^2\ [M_{22}])\ \{a_2^r\} = \{0\}$

where

$$[K_{22}] = [K_{22}^*] - [K_{21}][K_{11}]^{-1}[K_{12}]$$

and

$$\{a_1^r\} = -[K_{11}]^{-1}[K_{12}]\{a_2^r\}$$

$\{a_1^r\}$ = dependent degrees-of-freedom

$\{a_2^r\}$ = independent degrees-of-freedom

By this means one avoids the necessity of working with a full set of equations. This not only makes the computational effort much more efficient in terms of computer time but also improves the quality of the numerical results. The inverse $[K_{11}]^{-1}$ is obtained by the Choleski decomposition procedure (3).

For modal extraction the formulation is as follows:-

Let

$$\{a_2^r\} = [M_{22}]^{-\frac{1}{2}}\{q_2^r\}$$

Then

$$([K_{22}^*][M_{22}]^{-\frac{1}{2}} - \omega_r^2[M_{22}]^{\frac{1}{2}})\{q_2^r\} = \{0\}$$

Or

$$([\bar{K}_{22}] - \omega_r^2[I])\{q_2^r\} = \{0\}$$

where

$$[\bar{K}_{22}] = [M_{22}]^{-\frac{1}{2}}[K_{22}^*][M_{22}]^{-\frac{1}{2}}$$

In problems where in the dynamic or independent degrees-of-freedom $\{a_2^r\}$ in a model exceed the capability of routines employed to extract the eigenvalues and eigenvectors (such as Jacobi iteration, Household QR, etc.), and when extraction of modes from a narrow frequency range is desired, the "Inverse Iteration" procedure may prove to be more efficient. The Inverse Iteration method is also referred to as the Inverse Power Method with shifts. The method converges to the nearest eigenvalue to the shift point.

If there are no zero masses on the diagonal of the mass matrix [M] in equation [1], the displacement vectors {a} for the natural modes of vibration are found by solving the following eigenvalue problem:-

[4] $[K]\{a_n\} = \omega_n^2\ [M]\{a_n\}$

where

$\{a_n\} = n^{th}$ eigenvector

$\omega_n^2 = n^{th}$ eigenvalue

If there are p distinct mass locations and [K] is square, pxp, then there are p displacement vectors, one for each mode. Therefore we formulate the modal transformation matrix $[\Phi]$ whose columns are the displacement vectors $\{a\}$, i.e.

$$[\Phi] = [\{a_1\}\{a_2\}----\{a_p\}]$$

Having obtained $[\Phi]$ we now represent the vector of displacements $\{u\}$ by means of the following

[5] $\{u\} = [\Phi]\{q\}$

where $\{q\}$ is a generalized displacement vector.

Use of the modal transformation relation [5] decouples the system of equations [1]. The n^{th} decoupled equation is similar to a one-degree of freedom system, being in fact

[6] $\ddot{q}_n + 2\beta\omega_n\dot{q}_n + \omega_n^2 q_n = -\gamma_n \ddot{y}_g$

where

$$\gamma_n = \sum_{r=1}^{j} M_r \Phi_{rn} / \sum_{r=1}^{j} M_r \Phi_{rn}^2$$

In equation [6] γ_n is the participation factor for the n^{th} mode, Φ_{rn} is the $(r,n)^{th}$ element of $[\Phi]$, M_r is the mass at the r^{th} position and β is the critical damping ratio.

Using the response spectrum shown in Figure 1 for a one-degree-of-freedom system (3, 4, 5) we have

[7] $|q_n| = |\gamma_n D_n|$

where D_n = relative displacement response spectrum value corresponding to a frequency of ω_n.

Equation [7] is available for n = 1,2,--p.

The response for each individual mode having been obtained we may now proceed to summation of the effects.

This is possible in the following ways:-

1. Absolute Sum

[8] $|f_i| = \sum_n |f_{in}|$

2. RSS (Square root of the sum of the squares)

[9] $|f_i| = (\sum_n f_{in}^2)^{\frac{1}{2}}$

3. Combination of Absolute and RSS summation

In equations [8] and [9]

f_i = i^{th} internal displacement, velocity, acceleration, forces or stresses.

f_{in} = i^{th} internal displacement, velocity, acceleration, forces or stresses corresponding to its response value in the n^{th} mode.

The combination of absolute and RSS summation often takes the form that if two or more natural frequencies are within 5% of one another the effects are added absolutely; therefore RSS summation is employed. In most codes however the requirement of absolute summation does not apply when the total number of modes is very large, as is the case for example when finite element methods of analysis are used.

CASE STUDY OF TURBINE BUILDING OF THE POINT LEPREAU NUCLEAR POWER PLANT

In this part of the paper a case study of the seismic analysis of the turbine building of the Point Lepreau Nuclear Power Plant complex is presented. The general outlay is shown in Fig. 2. Figures 3, 4 and 5 show respectively a typical elevation, side elevation and plan. There were 14 elevations, 8 side elevations, and up to nine floors.

General description of the turbine building

The Turbine Building consists of two areas; the auxiliary area and the turbine area. The areas are split into two bays above the 45 feet level by an expansion joint at lines 11 and 11a. The building is fully connected below the 45 feet level. Figures 2 and 5 illustrate the description.

(1) The auxiliary area--This area is bounded by columns L5, N5, L17 and N17, and is used for offices and utilities. The area is divided into two bays. The first bay is bounded by Column Lines 5 to 11, has floors at elevations 22', 35' 58'-6", 75'-3" and 98'-9". The roof slopes from elevation 123' at line N to 122' at line L. The floors comprise 7" slab (5½" concrete slab and 1½" metal decking) on steel beams. The roof is 1½" metal decking on steel beams. The steel beams at El. 22' and El. 35' at Line 5 are anchored to the foundation wall.

The second bay is bounded by Column Lines 11a to 17, has floors at elevations 22', 45', 75'-3", and 98'-9". The roof slopes from elevation 137'-10½" at Line N to 136'-10½" at Line L. Floors and roof construction of second bay are similar to the first bay. These floors and roofs act as diaphragms to transfer horizontal loads to bracing frames on Line L, N, 11a and 17.

(2) The turbine area--This area is bounded by columns N5, T5, N17 and T17 and houses the turbine installation. The area comprises a truss frame spanning 150' between column lines N and T, with floors at elevations 10', 17', 22' and 45', as shown in Figure 3. The truss is pin connected to the line N columns, which also support the auxiliary bay floors. The roof on the truss frame slopes from elevation 120' at Line T to elevation 123' at Line N, and comprises 1½" metal decking, fusion welded to purlins running between the trusses. The truss frame supports a travelling crane which runs from line 5 to 17. Two sets of horizontal bracing at the plane of the top chord of the truss transfer horizontal load to bracing frames 5, 11, 11a, 17 N and T.

The turbine area floors at elevations 10', 17', and 22' are 1½" steel grating spot welded to steel beams, whilst that at elevation 45' comprises a 5½" concrete slab on 1½" metal decking. Horizontal steel bracing at these four elevations transfers horizontal loading to vertical bracing.

Exterior enclosed walls are double tee concrete panels except on Line 17 which has steel siding.

Loads--Locations and intensities of dead and live loadings were specified. These typically comprised uniformly distributed loads due to self weight and concentrated loads due to equipment and piping.

Section properties--All section sizes were specified. It may be noted that the basic design approach was one of material saving, rather than standardization on relatively few sections; hence a great many section sizes were involved.

Material specifications--Structural steel was specified to conform to CSA G40.21 - 44W. Specified concrete strength (f'_c) was 3000 psi.

Connectivity assumptions--All beams were assumed to be pin connected to columns, and the columns themselves were assumed to be pin-footed.

Design damping ratio "β"--In selecting a design damping ratio, the 1975 National Building Code was followed; this recommends β=.03 for structural steel with welded connections and lightweight exterior and interior connection, and β=.05 for structural steel with welded or bolted connection with heavy exterior walls, normal interior partitions, reinforced concrete structures etc. The value β=.05 was selected as the damping ratio.

Earthquake input--Two references (1, 8) were followed. The response spectrum was provided by AECL.

The investigation concerned itself with the following aspects of the analysis:

(a) Examination of the structure as a three dimensional system composed of linear and plate members.

(b) Establishing a suitable three dimensional finite element model of the complete structure and obtaining its dynamic response behaviour.

(c) Establishing a suitable approximate model (the "STICK MODEL").

The finite element model of the building--As noted above, the building is split into two parts above the elevation 45' level. The totally connected nature of the building below the 45' level is such however that it proved impossible to treat the whole as two structures. Accordingly the entire building was idealized with finite elements.

The fourteen elevations, spanning on lines 5 to 11 and 11a to 17, were considered; a typical elevation is shown in Figure 3.

Eight side elevations, spanning on lines L, M, N, P, Q, R, S and T were considered; a typical side elevation is shown in Figure 4.

Nine floors were idealized for the building bounded by the lines 5 to 11 and eight floors for that bounded by lines 11a to 17; a typical floor plan is shown in Figure 5.

Three types of elements were used in the idealization of the building; these were the truss, beam and quad-plate or tri-plate elements as used in the "STARDYNE" programme. The corners or ends of the elements defined the nodes, and the nodes were numbered systematically in the planes of elevations 5 to 17 taken in sequence. It is fortunate that the STARDYNE programme has the following two aspects:-

(i) the numbering of the nodes can be discontinuous.

(ii) bandwidth optimization is carried out on the connectivity matrix of the final node numbering.

One needs these two features in order that new nodes (corresponding to redesign of parts of the building) can be introduced and numbered alongside the old nodes without the solution time becoming very lengthy and therefore costly. It is useful to recall that the solution time is approximately proportional to the square of the bandwidth; in the finite element model now being discussed the initial bandwidth was 5094 undeleted degrees of freedom, whilst the final bandwidth was 574.

The typical truss shown in Figure 3 could not be modelled exactly; a pin jointed equivalent truss was used for the modelling, the properties of the equivalent truss being chosen so as to give the same strain energy function of bending moment and shear force as the original. This saved about 25 nodes per truss, or approximately 300 nodes for the whole building; the saving enabled the finite element model to stay within the circuits of capability of the STARDYNE program, which were 1000 nodes and 6000 degrees-of-freedom.

Clearly, not all of the beams of the building lay on the lines joining the node positions of the finite element model. Beams that were not on nodal lines were conceptually moved on to such lines, sometimes with a division of the beam properties between two parallel nodal lines. No such conceptual movement took place for columns however, and the relevant nodes were placed on the column lines in all cases.

The floors were for the most part idealized as quad-plate or tri-plate elements and several different types of properties were developed to suit them. Since both quad-plate and tri-plate elements are based upon a continuum of material and material properties areas of steel grating floor were represented as equivalent continua having the same properties per unit width as the grating.

Masses were calculated automatically by the programme for the uniformly distributed elements whilst concentrated masses were applied individually at the appropriate nodes. Some of the modelling techniques are shown in Figures 3, 4 and 5.

Data Preparation--Nodal data required included nodal co-ordinates and restraints, referred to global axes x_1, x_2 and x_3, as shown in Figure 2. Initially six degrees-of-freedom were assigned to each node; subsequently, degrees-of-freedom were suppressed as appropriate.

Element data provided to the programme included

(a) geometric definition of the element by type and nodal co-ordinates.

(b) material properties of the elements.

(c) section properties of the elements. For beam and truss elements the number of sections involved was very large, and therefore a separate programme was written to develop a section property table for the STARDYNE programme from part of the Canadian Institute of Steel Construction manual.

Data for response spectrum analysis was taken from the spectrum for the reactor site developed by AECL. The overall data is shown in Table 1.

Data checking--This was perhaps the most difficult step in the entire analysis procedure. Initially a small "stand alone" programme was used to check the format compatibility between the data on the cards and the STARDYNE programme; once this was achieved the data was processed through the STARDYNE geometry phase.

Subsequent to the geometry phase two types of output were checked. The first of these was graphical; computer plots describing Figures 3, 4 and 5. The second was numerical in nature, providing such information as length, depth, thickness, pin codes etc.; the STARDYNE programme has several intelligent "error message" features which were very helpful at this point. A few iterations of the geometry phase and associated checks were sufficient to remove the errors.

Finite element analysis of the structure--

The structure was analysed with the STARDYNE Program using a CDC 6600 computer.

The analysis consisted of five distinct phases:

1. Geometry Phase
2. Static Run Phase
3. Frequency Analysis and Modal Extraction Phase
4. Post-Processing Phase
5. Response Spectrum Analysis Phase

Geometry phase--All the input is printed out as card images followed by a printout of data in a convenient format.

The bandwidth optimization is done by the program. This gives flexibility in numbering the nodes. The comparison of the time estimates between HQR and Inverse Iteration was both interesting and very important from a cost point of view. It was clear that Inverse Iteration would be a very costly method, even for a few modes, being about three times as expensive as HQR. Moreover, for a very large structure many of the modes generated are purely local in nature; because of this an entire inverse iteration run could be useless, wasting about $6,000 to $10,000 of computer time without any useful information coming out of the analysis.

At this point it was decided to lump the masses of the building at only 45 nodes, strategically chosen, usually above columns, in such positions that the overall effect was consistent with the actual masses; this was necessary in order to use HQR. The authors feel that this is a better approach than taking the actual mass distribution and using inverse iteration; as mentioned above any mistake in the latter could be very costly. The point here is to ensure that only "overall" modes of vibration are included.

Static run phase--The static run was done to calculate deflections, for basically two reasons:

1. To establish the "STICK MODEL". A "stick model" is an assembly of line elements, usually vertical or horizontal, with masses at some or all of the nodes. The structural properties of the line

elements and the size and distribution of the masses are so chosen that the dynamic behaviour of the stick model is a good approximation to that of the actual building so far as the lower natural frequencies are concerned.

2. To investigate the integrity of the 3-D finite element model.

The lateral loads were calculated in the x_1, and x_2 direction by applying 0.2g in those two directions with the masses lumped at the 45 nodes.

Frequency analysis and modal extraction (HQR) phase--All 135 frequencies were extracted, these frequencies corresponding to 135 dynamic degrees-of-freedom (45 nodes with three dynamic degrees-of-freedom per node).

Post processing phase--This is a static run to calculate deflections and forces corresponding to the eigenvectors calculated by HQR.

Spectrum analysis phase--In this phase four runs were done in the x_1, x_2 and x_3 directions respectively, and the fourth was a combination of x_1, x_2 and two-thirds of x_3. Input to this part was in accordance with the AECL response spectrum in terms of frequencies and corresponding accelerations. This phase is very costly per run, as can be seen in Figure 6. In fact the large computer came to a standstill when 60 modes were tried. We found that 18 modes were sufficient for our purpose, and that no trouble then ensued.

On several occasions a job was cancelled because of the "Thrashing Phenomenon". By "thrashing" is meant the situation in which two or more large programs are competing with one another for use of the computer resources; it can happen that an available time slice to one program is entirely taken up in rolling information from secondary storage to main storage, at the end of which the demands of a second program take over and require the information to be put back into secondary storage so that the main storage may be available to the second program. The same sequence can be repeated in relation to the second program, so that in effect no computational work is done by the processor.

Discussion of the results--The importance of the static run phase is paramount in establishing a low cost analysis. A static run costs only a very small fraction of that of a response spectrum analysis of the entire building. The overall transverse stiffness with respect to horizontal loading which are obtained from the static run enable one to make large savings in the subsequent dynamic analysis. A suggested sequence of steps is given later in this paper.

As noted above the response spectrum analysis was carried out for a structure having forty-five lumped mass locations with three degrees-of-freedom at each. In the Householder method all one hundred and thirty-five eigenvectors and their associated natural frequencies are calculated, without regard to the user's wish only to extract information about a much smaller number. All of these modes were plotted and scrutinized by the authors, and the purely local modes were rejected, as were those corresponding to the higher frequencies.

Eventually the response spectrum analysis was carried out with respect to the first eighteen "overall" modes, a step which resulted in substantial savings. The output was in simple and readable form, comprising RSS ("root sum square") deflections for all nodes in the x_1, x_2 and x_3 directions, and RSS forces and stresses for all elements. The results are shown in Figures 8 and 9; these are discussed below in comparison with those of the stick model.

STICK MODEL FOR THE DYNAMIC ANALYSIS OF POINT LEPREAU NUCLEAR POWER GENERATING STATION BUILDING

The stick model for the dynamic analysis consists of six sticks and is shown in Figure 7. It is made up of 31 nodes and 35 members. THe proposed model is based on the following assumptions:

(i) The flexural rigidities of various members are proportioned to give the same static deflections in x_1 and x_2 directions as are obtained from the three dimensional finite element static analysis of the actual structure, see Fig. 8. Assumed values of areas and moments of inertia for members are given in Table 3. The static deflections for 0.2g loadings are shown in Figure 8. The proportioning of members of the stick model was carried out iteratively and three iterations were sufficient; the cost was very low, being less than $10 per iteration.

(ii) The floor masses are applied as lumped masses to their respective nodes, the details of which are given in Table 4.

(iii) Nodes 1 to 8 are clamped to the ground. All other nodes are allowed to have only the three translational degrees-of-freedom.

Comparison of Results Between the Building and the Stick Model

Table 2 gives the frequencies of the first eighteen vibration modes of the building, whilst table 5 gives the same information for the stick model. The close correspondence for the first ten modes is noted, after which (as may be expected) there is dispersion.

Figure 8 shows a comparison of deflections by both analysis; the comparison is good, indicating that the two have similar response to static load.

Figure 9 gives RSS accelerations by both analyses; once again the results are close. This indicates that one could use the accelerations from the stick model for the design of mechanical and electrical equipment. It is noted that the stick model cannot give directly stresses and forces in the actual building. However one can use the RSS accelerations of the stick model as input for an analysis of the complete building. An important result from the work now being reported is the confirmation of the validity of this procedure. Further work in verification of it is still going on.

The preferred sequence of steps in holding down costs is therefore as follows:-

(i) represent the building by a finite element model

(ii) run the geometry phase

(iii) subject the finite element model to static horizontal loading and carry out a static analysis

(iv) from the results of the static analysis design a stick model

(v) carry out dynamic analysis on the stick model

(vi) use the accelerations obtained from the dynamic analysis of the stick model as input loading for analysis of the complete building, thereby obtaining forces and stresses in the members.

Clearly the reasoning behind steps (i) to (vi) above is the avoidance of doing a dynamic analysis of the entire building.

CONCLUSIONS

In this paper the dynamic analysis of a very complex building, using a finite element model, has been described. The points of difficulty, and the ways in which large amounts of computer time may be wasted, have been identified. A sequence of analysis using both the full finite element representation of the building and a much simpler stick model has also been presented and recommended.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge permission given by the New Brunswick Electric Power Commission to reproduce results of consulting work done for them. They are also happy to acknowledge the co-operation which has always been forthcoming from Mr. John Pader, Senior Structural Engineer with the New Brunswick Power Commission and his colleagues Mr. Robert Fairweather, Mr. Robert Listen and Mr. Bijoy Saha. This co-operation has greatly facilitated the progress of the work now being reported. The computer work was carried out on a CDC 6600 Computer at Multiple Access Ltd.

REFERENCES

1. Atomic Energy of Canada Limited, Earthquake Design Requirements for CANDU Nuclear Power Plants, DESIGN GUIDE, Nov. 1974.

2. National Building Code of Canada 1975, issued by the Associate Committee on the National Building Code, National Research Council of Canada, Ottawa.

3. Rosen, R., Rubenstein, M.F., "Dynamic Analysis by Matrix Decomposition", Journal of the Engineering Mechanics Division, ASCE, Vol. 94, No. EM2, Proceedings Paper 5879, April 1968, pp. 385-395.

4. Biggs, J.M., Introduction to Structural Dynamics, McGraw-Hill, New York, 1964.

5. Clough, R.W. and Penzien, J., Dynamics of Structures, Prentice-Hall, Inc., New York, 1975.

6. Hurty, W.C. and Rubenstein, M.F., Dynamics of Structures, Prentice-Hall, Inc., New York.

7. Alexander, C.M., Smith, E.C., "Lepreau Design Basis Earthquake Spectra". Atomic Energy of Canada Ltd., Publication No. DGA-87-01041-1, June 1975.

8. Smith, E.C. et al "Design Guide", Atomic Energy of Canada Ltd., Publication No. DG-00-01040-1.

TABLE

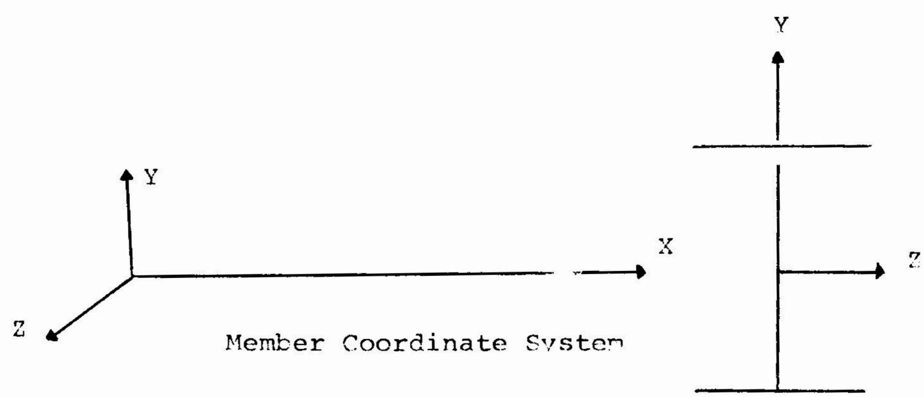
Number of Nodes	=	1000
Number of Elastic Bar Members	=	3774
Number of Nodes With Restraints	=	1000
Number of Tri-Plates	=	32
Number of Quad-Plates	=	302
Number of Nodes with Weights	=	45

Table 1 . Finite Element Idealization's Over-all Data for Turbine Building

MODE NUMBER	FREQUENCY (F)
1	.86
2	1.38
3	1.54
4	1.68
5	1.85
6	1.92
7	2.00
8	2.52
9	2.59
10	2.75
11	2.84
12	2.90
13	2.91
14	2.96
15	2.99
16	3.11
17	3.28
18	3.32

Table 2. Eigen-Values and Frequencies

Member	Area (Sq. ft.)	I_x (ft^4)	I_y (ft^4)	I_z (ft^4)
1 to 6	50.0	0.0	15.6	3.2
7 to 8	60.0	0.0	20.0	5.0
9	16.0	0.0	5.0	5.0
10 to 14	25.0	0.0	34.5	13.5
15 to 16	18.0	0.0	18.0	5.0
17	3.0	0.0	2.5	5.0
18,34,35	16.0	0.0	3.75	1.1
19,32,33	16.0	0.0	2.15	7.2
20 to 25	0.5	0.0	0.0	1.5
26 to 27	18.0	0.0	0.0	0.0
28 to 31	0.5	0.0	0.0	0.0



Member Coordinate System

Table 3. Sectional Properties

Node No.	Mass (in 1000000 Slugs)
1 to 8	0.0
9	0.044
10	0.077
11	0.1194
12	0.068
13	0.0094
14	0.0675
15	0.123
16	0.0087
17	0.03
18	0.029
19	0.067
20	0.1594
21	0.1267
22	0.0289
23	0.102
24	0.0935
25	0.0275
26	0.0165
27	0.013
28 to 31	0.0

Table 4. Distribution of Masses to Various Nodes

Mode No.	Frequency (Hertz)
1	0.86
2	1.40
3	1.48
4	1.88
5	1.90
6	2.02
7	2.37
8	2.77
9	2.80
10	3.54
11	3.69
12	4.20
13	4.44
14	4.93
15	5.94
16	7.03
17	7.43
18	7.45
19	7.89
20	9.00
21	9.66
22	11.04
23	11.74
24	12.98
25	13.92
26	15.63
27	18.88
28	22.64
29	23.86
30	29.82

Table 5. Frequencies in Stick Model

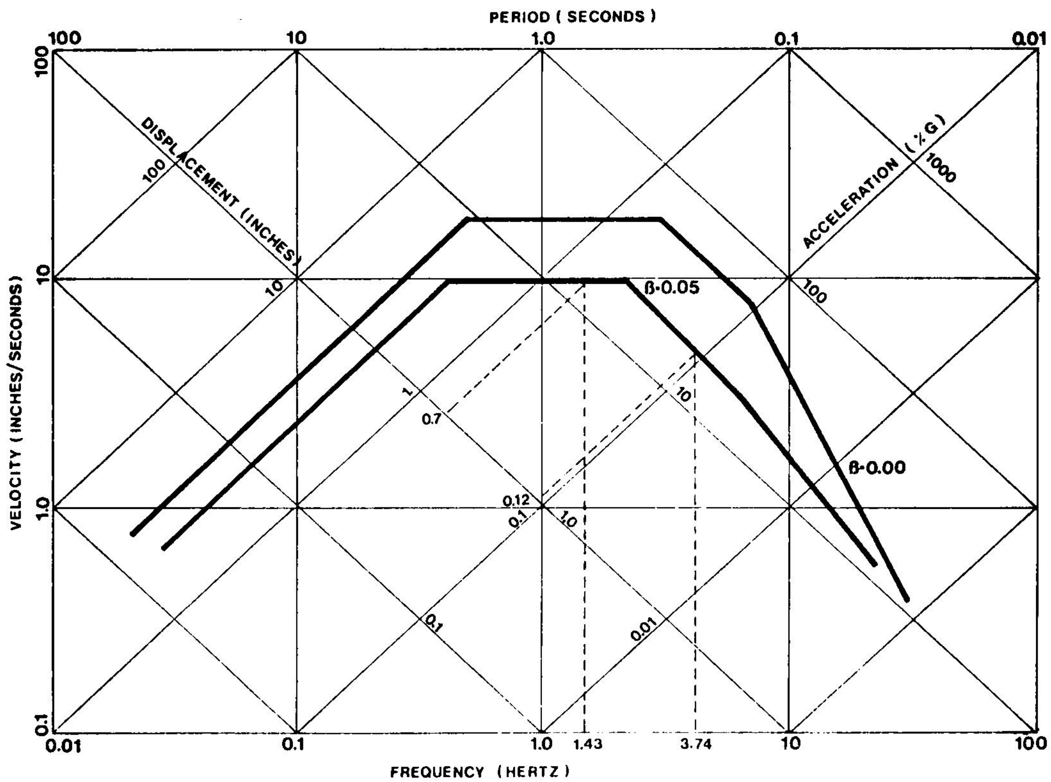


Fig. 1 Design Seismic Response Spectra

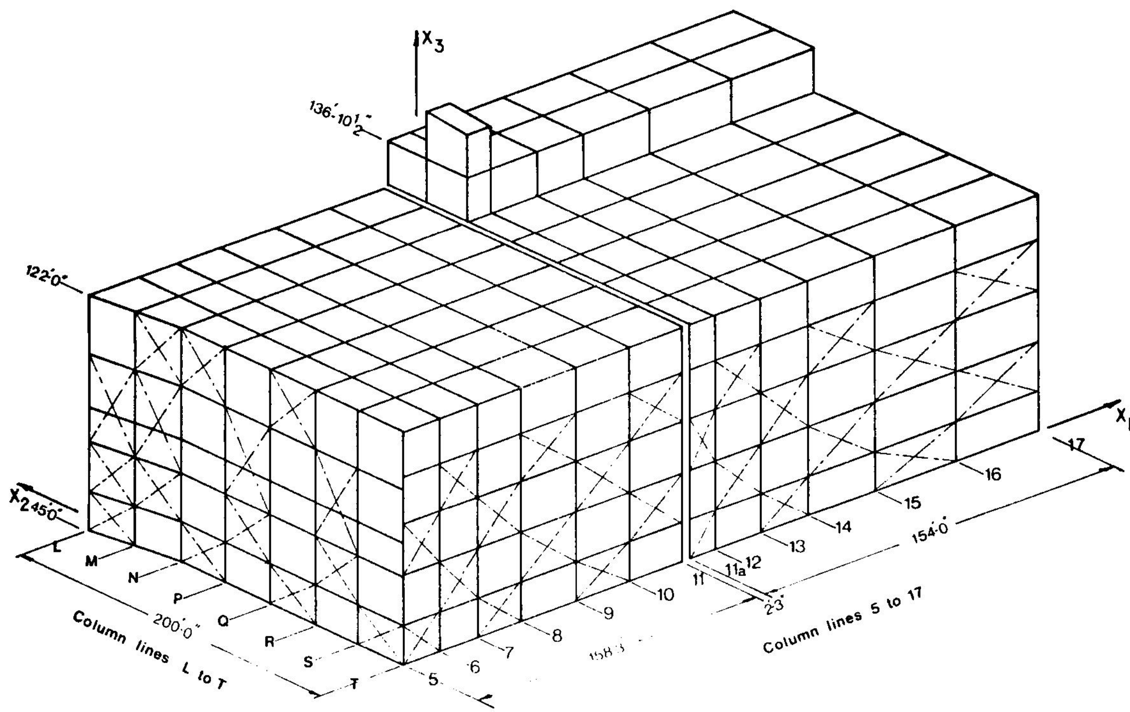
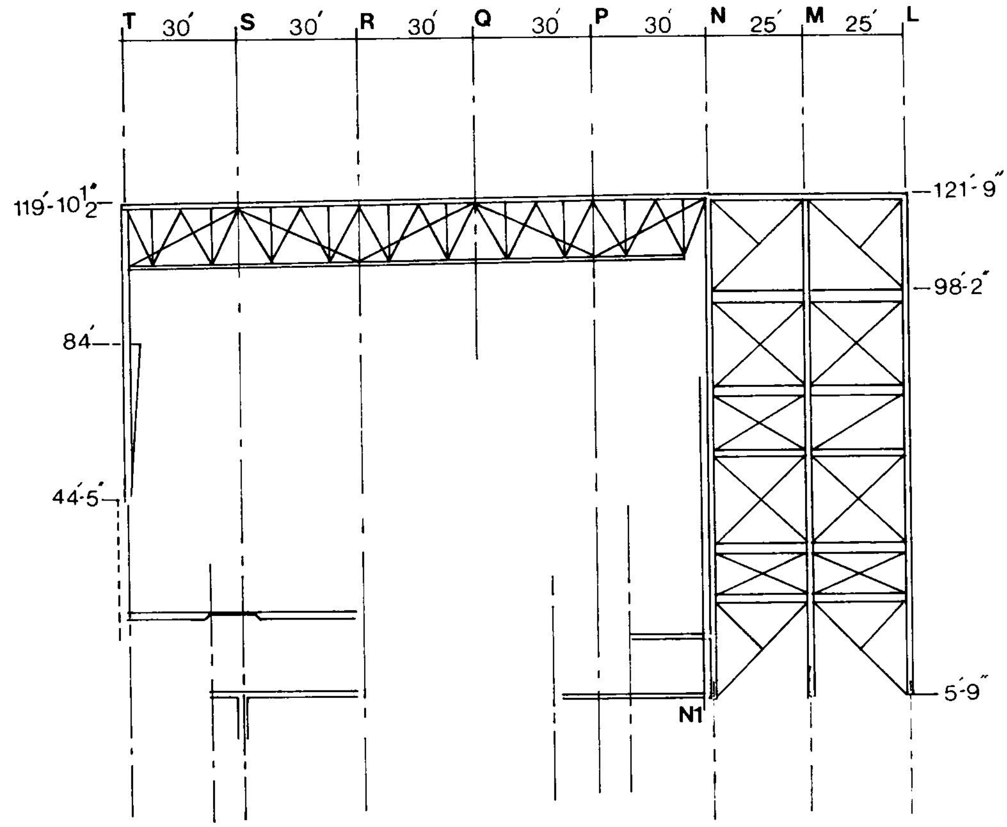
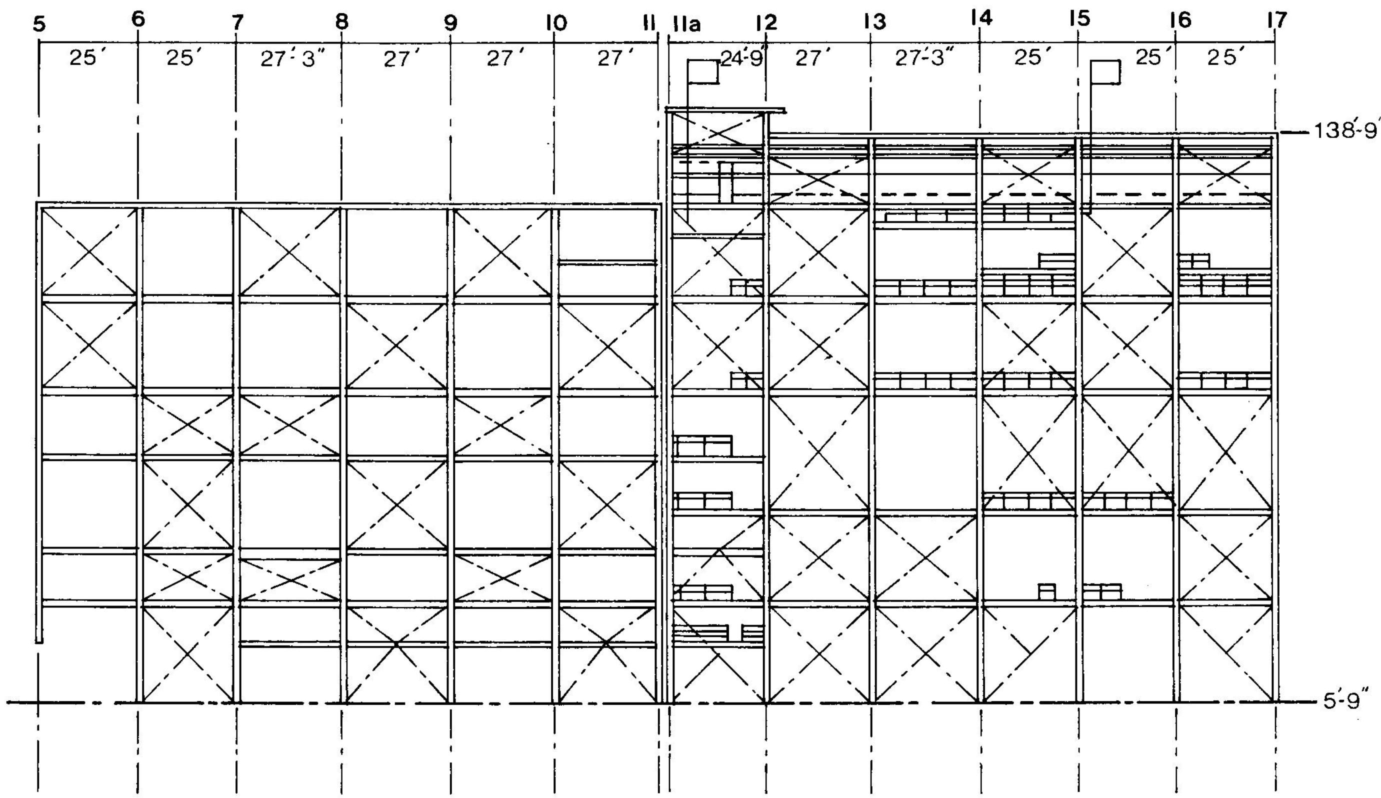


Fig. 2 3-D Building: Nuclear Powered Turbine Building



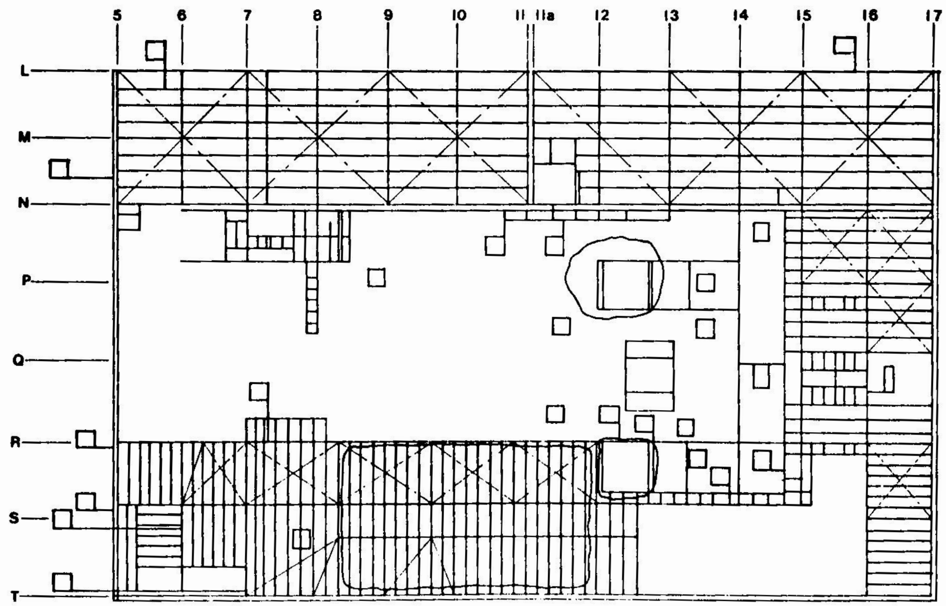
Elevation at Column Line 11

Fig. 3 Elevation at Column Line 11



Elevation at Column Line N

Fig. 4 Elevation at Column Line N



Structural Steel Floor Plan at Elevation 17'0" & 22'0"

Fig. 5 Floor Plan at Elevation 17'-0" & 22'-0"

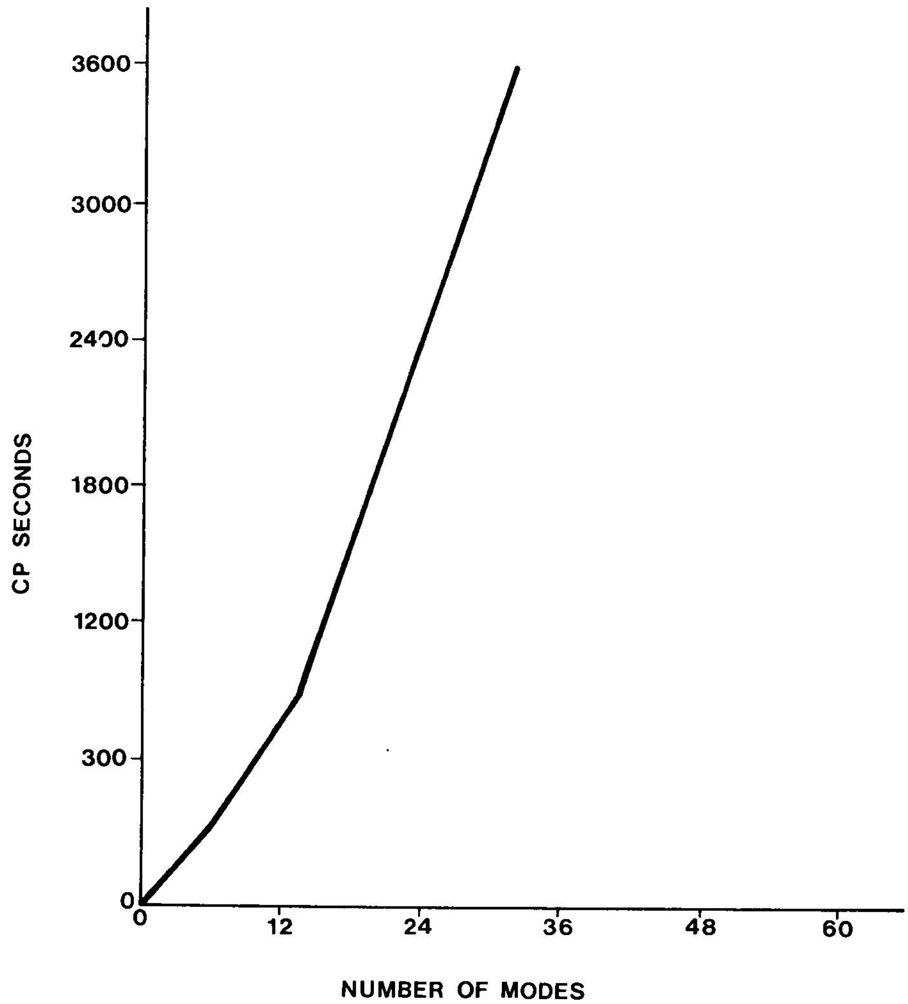


Fig. 6 Response Spectrum Analysis; Number of Modes VS CPU Seconds

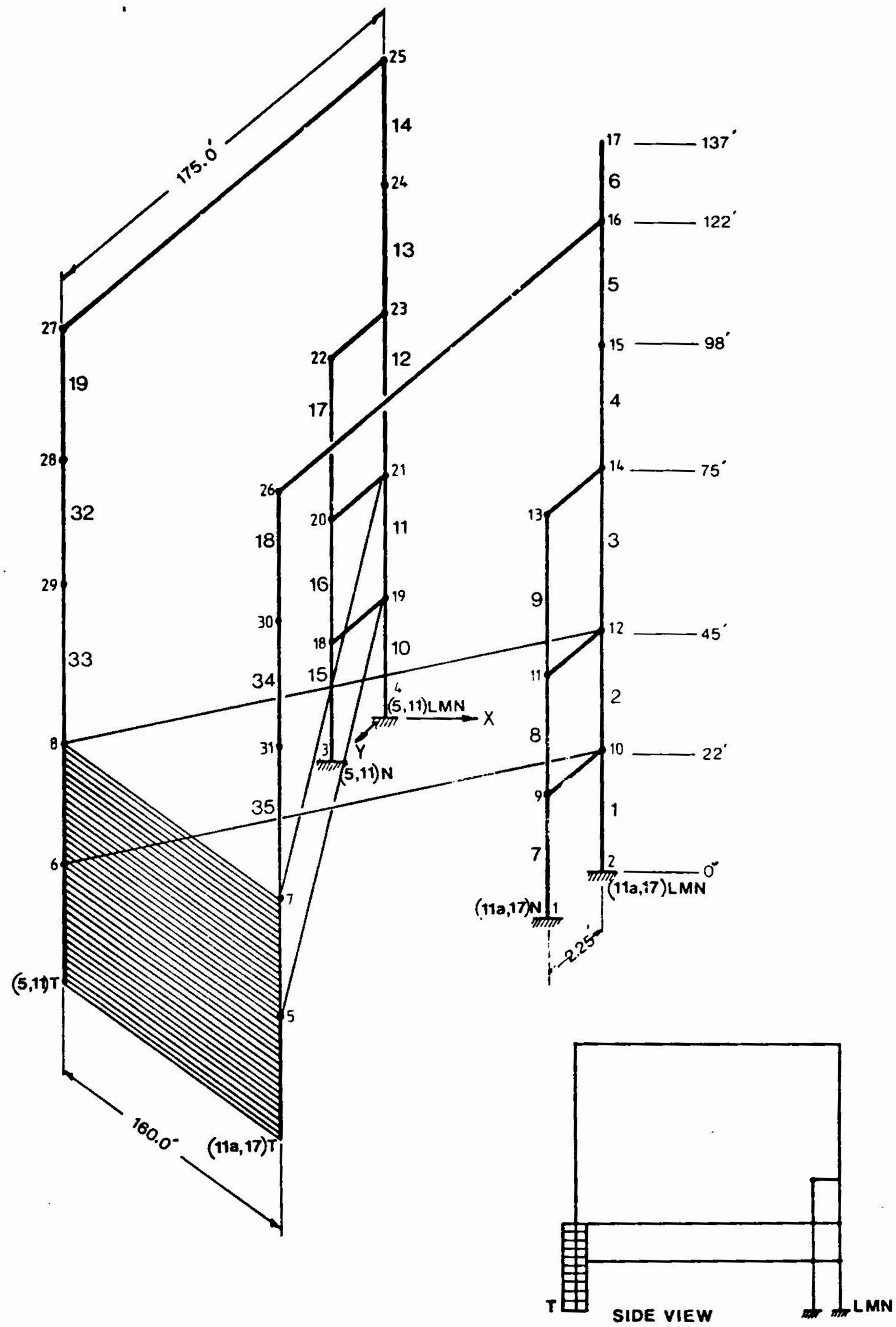
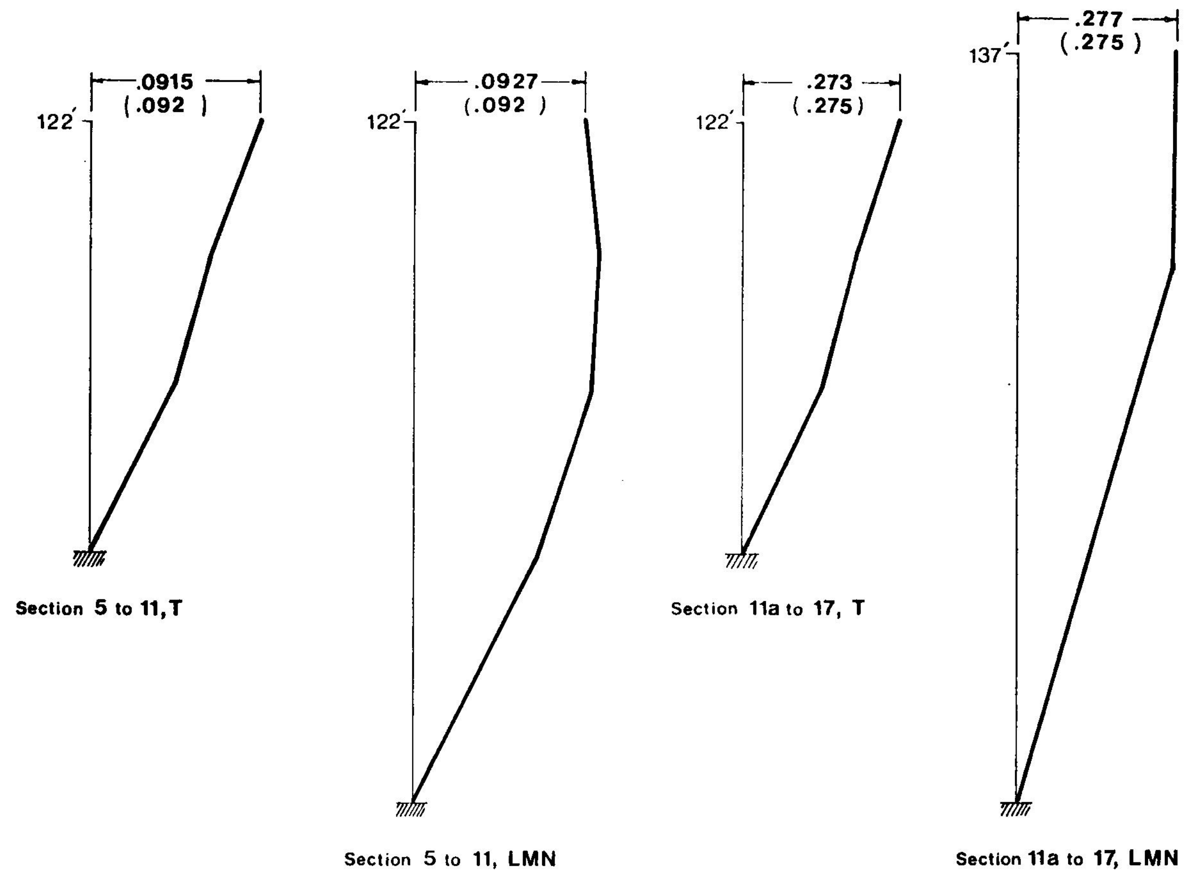
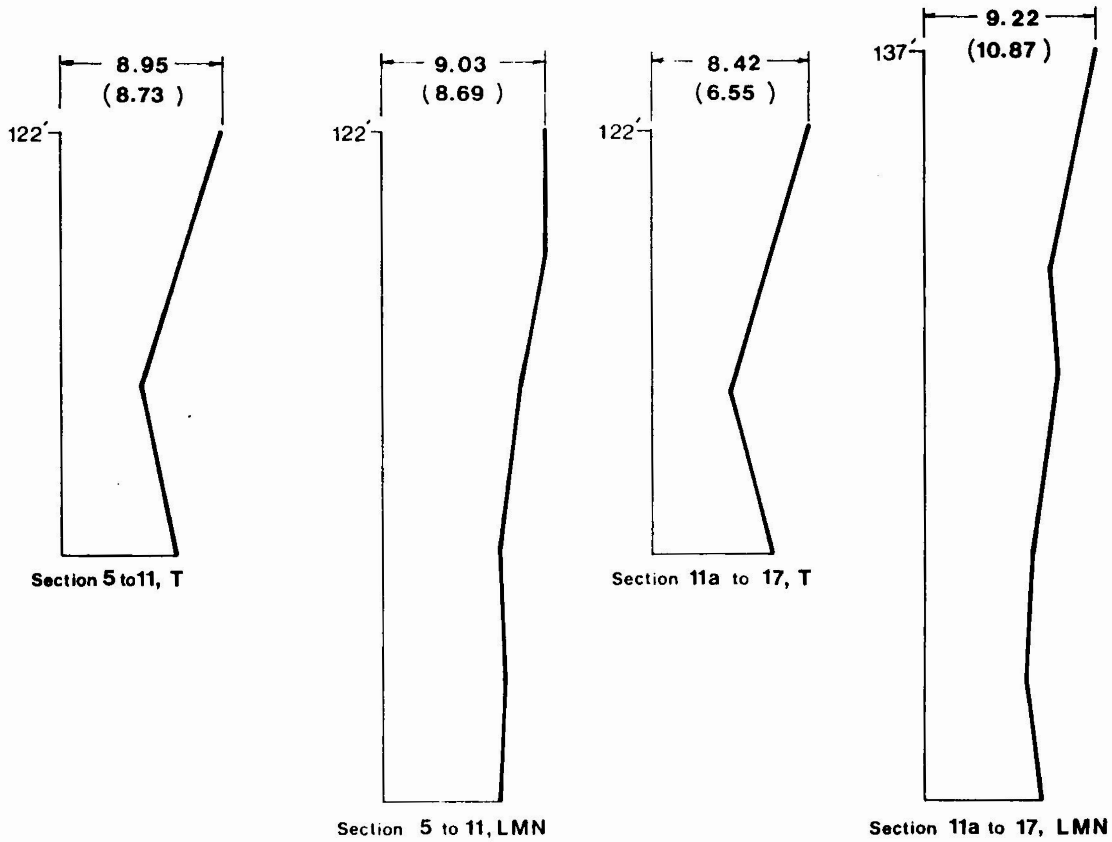


Fig. 7 Stick Model



NOTE: Values within brackets are those which have been obtained from three dimensional analysis.
Values are in feet.

Fig. 8 Static Deflections of Complete Building and Stick Model in x_2 Direction



NOTE: Values within brackets are those which have been obtained from three dimensional analysis
Values are in feet/sec.^{2}

Fig. 9 R.M.S. Acceleration of Complete Building and Stick Model in x_2 Direction