SEISMIC STRESSES OF PIPING SYSTEMS AND EQUIPMENT ON HEAT EXCHANGER SUPPORTING STRUCTURES

K.E. LEUNG

Bechtel Power Corporation, San Francisco, California 94119, U.S.A.

SUMMARY

A three-dimensional structural analysis program was used to analyze an unsymmetrical heat exchanger steel frame structure with an externally attached piping system and a laminated steel stack. The analysis indicated that the seismic stresses in the piping system considerably exceeded those from the conventional piping analysis in which coupling effect between the piping system and the structure was neglected. It appears that to calculate the stresses due to the differential seismic displacements of the pipe anchors according to ASME Section IV may not be sufficient in this case.

Local stresses such as those at the stack bracing connections, were computed by modeling the local system as finite elements and using the response spectra generated there. These local stresses were then combined with those obtained from the overall 3D structure response analysis.

Some equipment and pipings may have gaps. This was treated by using gap elements.

In conclusion, for nuclear piping systems with anticipated high temperature loads, it is necessary to consider the coupling effect between the piping and the building structure in the seismic stress computation.

1.0 Introduction

Piping systems and equipment in liquid metal fast breeder reactor (LMFBR) plant are designed to withstand severe seismic loads at elevated temperature in heat exchanger supporting structure. Application of finite element program for performing seismic and thermal stress analysis is considered as the only method can cope with the complete range of structural geometries and behavior mechanisms,linear and nonlinear, in a single computer program [1] . With the existing computational technique, it is demonstrated in this paper that a complete system dynamic analysis can be obtained analytically for three dimensional integrated structures of a general nature. A well knowngeneral purpose computer program ANSYS available in the United States is used for the analysis [2] . To improve accuracy of the piping stress calculation due to interaction response of the equipment and sub-systems of the heat exchanger supporting structure ,an integrated model approach is applied. The integrated model approach may be able to replace the conservative assumptions such as floor spectra computing without consideration of equipment interation of main system and using envelop of the maxima of the several support response spectra as piping system seismic input forcing function,or the not so conservative assumption of treating piping stress induced by differential anchor movements as secondary stress [3]. An additional advantage of using the integrated model arises because the same mathematical model may be defined identically with the thermal model. Time and cost can be saved by such transfer of data from one to the other[1].

The drawback of using the integrated model approach is the finite limitation of computer memory core size. But with increasing capacity of computer in the future and some large capacity computer with secondary storage available at present, reasonable detailed three dimensional mathematical integrated model can be performed with reasonable times. It is important that the user of large finite element computer programs should be careful not to use a finer model or excessive dynamic degrees of freedom than is necessary to obtain meaningful results. Programs have element library including the three dimensional stiffness element and sub-structure element are particular useful in the integrated system modelling. Verification,development of programs may be found in[5] [6] [7] [8].

In the next section the procedures and modelling techniques of the integrated system for seismic dynamic analysis are described. Two examples obtained from the analyses of heat exchanger supporting structures are present. The first example is response spectrum modal dynamic analysis of an integrated model with piping systems and equipment on heat exchanger supporting structures. The second example is mode frequency analysis by finite element method and by experimental method. The results from both methods are shown in good agreement.

2.0 Modelling techniques and procedures

Dynamic response of the integrated system is governed by the system's flexibility and mass distribution as well as dynamic degrees of freedom. The total system flexibility can be represented by sub-systems flexibility and individual structural members flexibilities , including lumped localized component shell flexibilities to connecting member. Many categories of the ANSYS element library are used to model the integrated system. Major structural members are represented by three dimensional beams,spars, and rigid beam elements. Concrete slab and wall are represented by quad flat shell element. Piping systems can be represented by elbow, pipe,and viscous damper elements. Nozzles supported by vessel and other interfaces may be represented by three dimensional stiffness element and sub-structure element.

3.1 Static model

A three dimensional mathematical static model is first developed according to design drawings from heat exchanger producer and supporting structure fabricator. Global geomtry and member stiffness properties are represented exactly as the drawings and specifications. Nodes are located at the intersection of the elastic lines with six static degrees of freedom. The static degree of freedom should be coupled or released depending on the detail connections condition of the joint. This is particulary important for piping supported by snubbers which must restrain the pipe in the plan normal to the axial axis of the pipe and free restraint of axial movement of the pipe. Three dimensional lumped mass elements and viscous damper elements are to be included in the static model . The post process of total forces combination between the dynamic run and static run require identical member numbering system of the two models. To reduce computer core requirement , the steel stack is represented by its bending stiffness only because breathing modes are usually in higher frequencies range which producing little significant interaction effect to the integrated system. The localized shell flexibility may be obtained by finite model of the shell with unit load input. The approximated flexibility of the localized shell is then combined with the connecting element to the shell. Final stresses of the shell are resulted from the sums of overall dynamic and static stresses in addition to local stresses restrain from connecting members.

3.2 Dynamic model

Development of the dynamic model is based on the static model. lumped weights may be redistributed to represent the actual seismic loading conditions, such as seismic stops for vertical suspension weights. Mass of liquid in tank or vessel may be represented by two masses with one rigidly fixed to the tank at an elevation above the bottom,and other mass attached through springs of stiffness K at an hgher elevation [4]. Dynamic degrees of freedom are to be included to represent the significant mode shapes and the seismically excitable frequencies. The integrated system are to be modelled to move in 3 orthogonal spatial direction simutaneously. Representative maximum values of the response of a given element of the integrated model is obtained by applying each of the three orthogonal spatial earthquake components independently to the base of the integrated model by method of spectrum dynamic analysis.

2.3 Mode-frequency analysis

The capability of solving for the natural frequencies of a large integrated model by the ANSYS program is based on a technique called [2] " matrix condensation " (Guyan Reduction). The mass and stiffness matrices are reduced to dynamic degrees of freedom as specified in the dynamic model developed in 2 .2 and the eigenvalue problem are solved for the N degrees of freedom . Since there are no technical guidelines available with respect to what constitutes a right set of N "master" degrees of freedom which one feels will characterize the natural frequencies of the large integrated model, one have reasonable expectations of the missing even the first mode frequency even with a great deal of professiona-l skill. Experimental analysis for mode frequency,therefore,is desired and can be used to confirm and refine the mathematical model if the major supporting structure is available. As the lowest natural frequency is a function of the lowest strain energy of the entire structural system, a simple 3-D model may be used to represent the major structure and perform full modal analysis [2] in order to looking for a right set of N "master" degrees of freedom for the large integrated model.

2.4 Time history dynamic analysis

This analysis is performed for the heat exchanger supporting structure to supply design response spectra for light equipment and to study location-s of minimum differential movements for piping supports. MRI/STARDYNE [9] program was used for this analysis because it is more efficient than other programs [2] [10] [11] [12] in our experience of 3-D model time history analysis. Also, using different programs may be considered as an independent checking for complex structural system. Fig. 1 and 2 show the mode shapes and generated floor spectrum from 3-D model by MRI/STARDYNE.

2.5 Spectrum modal dynamic analysis for integrated model

This analysis is performed for the following reasons;

(a) To supply seismic forces of the piping system for piping stress analysis.

(b) To perform final seismic stresses review for the heat exchanger supporting structure.

3.0 Example problems

In order to illustrate some of the characteristics of the integrated model dynamic analysis, two examples obtained from the analysis of heat exchanger supporting structure are presented.

3.1 Example No. 1

The integrated system as shown in Fig. 3 consists of three parts.

(a) Open space frame of the heat exchanger supporting structure.

(b) Piping systems.

(c) Major equipment and their sub-supporting structure .

The coupling effect of the stack to major structure can be seen from Fig. 1. Time history analysis shown that the accelerations are too large and varied too much in different elevations that modification of the supporting structure is necessary. Fig. 3 shows the front view and back view of the integrated model.

Table 1 shows the mode,frequency, and participation factor of the integrated model. Table 2 shows the piping , elbow, and major structural member forces and stresses.

3.2 Example No. 2

The integrated model as shown in Fig. 4 consists of open space frame, the laminated stack and major equipment. Accelerometers and vibrator are shown in locations indicated in Fig. 4. [13] .

Fig. 5 shows the analytical (finite element) solution of the mode shape and frequency and the experimental analysis[13] result of mode shape and frequency of the integrated model subjected to E-W motion. Fig. 6 and Fig. 7 are N-S motion and the torsional mode.

4.4 Disscussion

Although the seismic stresses can be obtained for the integrated model with existing computation techniques, the total stresses should include the local stress from gaps or local deformation in shells. Non-linear transient dynamic analysis can be performed [2] and detailed examples are available [14]. Future research and development are required in the areas of final stresses combination under conditions of loading history, dynamic yielding criteria and response displacement vector modal combination.

References

[1] GALLAGHER, R. H. "Computational methods in nuclear reactor structural design for high-temperature applications" ORNL-4765 (February,1973)

[2] SWANSON, J. A. "ANSYS Engineering analysis system user's manual" Swanson Analysis System,Inc., Elizabeth,Pennsylvania ,January 1973/

[3] ASME , ASME Boiler and Pressure Vessel Code Section 1V. USA

[4] NEWMARK, N. M. " Fundamentals of earthquake engineering" Prentice-Hall, Inc.N.J. USA 1971

[5] ASME, " Pressure vessel and piping 1972 computer programs verification" USA 1972

[6] ASME, " Pressure vessels and piping: Analysis and computers". Pressure vessels and piping conference,Miami Beach, Floria 1974

[7] SEVERUD. L, K. et al" Quarterly newsletter of FFTF project stress analysis" HEDL-SA-TN-2. Hanford Engineering Development Laboratory, USA October 1972

[8] SWANSON,J. A. " Status report on integrated analysis methods for nuclear reactor structural analysis", secondary international conf.on on structural mechanics in reactor technology,Berlin,Germany,Sep.1973

[9] STARDYNE: Users manual,Mechanics Research,lnc.1971 USA

[10] NASTRAM User's manual,level 15.5 ,The Macneal-Schwendler Corp. USA March 1974.

[11] MARC-CDC User information manual, Control Data Corp. USA

[12] SAP 4. " A structural analysis program " Report No. EERC 73-11. June 1973. University of California,Berkeley,California,USA

[13] Private communicantion with P. Ibanez and C.B. Smith of Applied Nucleonics Company,Inc.,Los Angeles,California,USA March 1975

[14] Engineering Analysis System Examples Manual .Swanson Analysis System Inc.

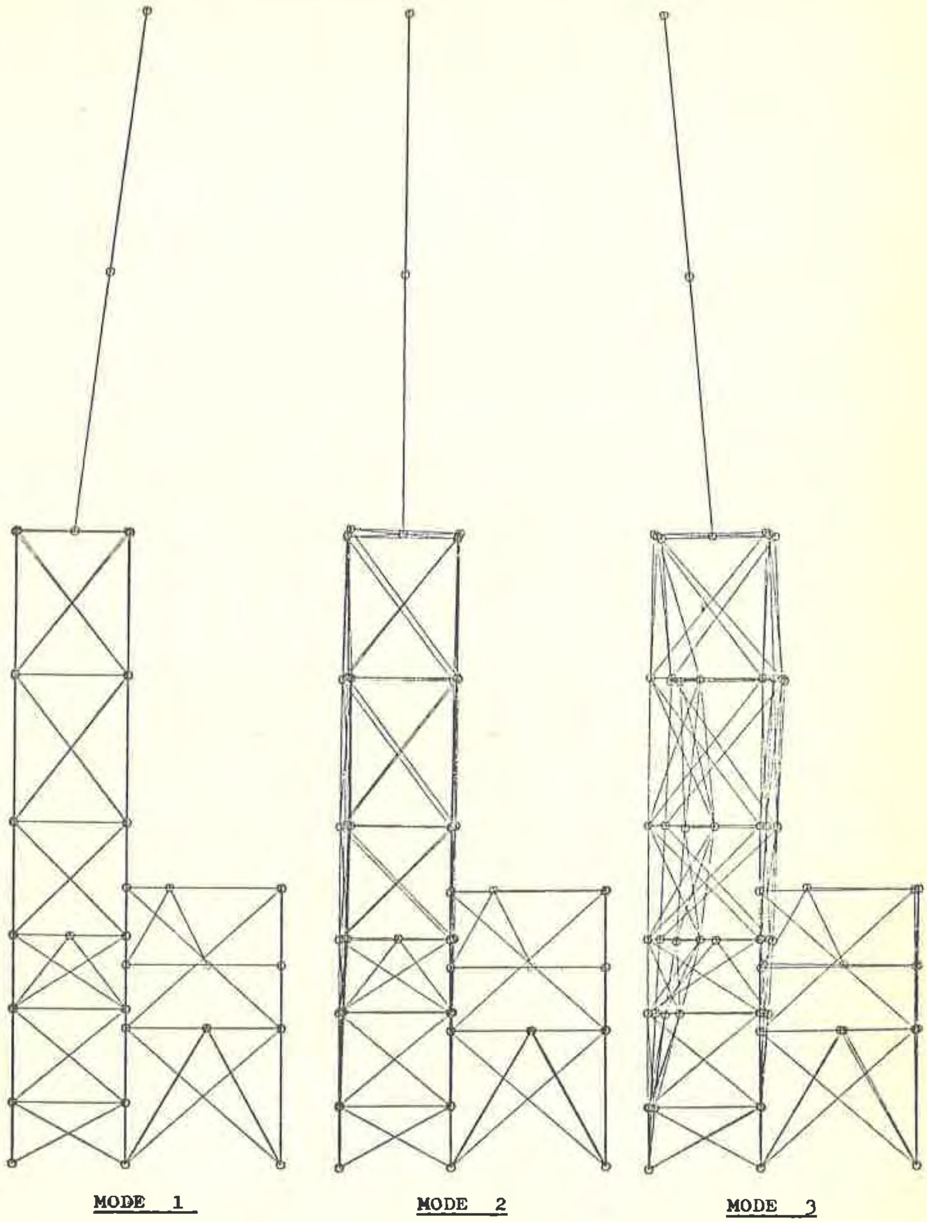


Fig. 1--- Mode frequency analysis by STARDYNE [9]

Fig. 2 Floor spectrum from 3-D model by STARDYNE

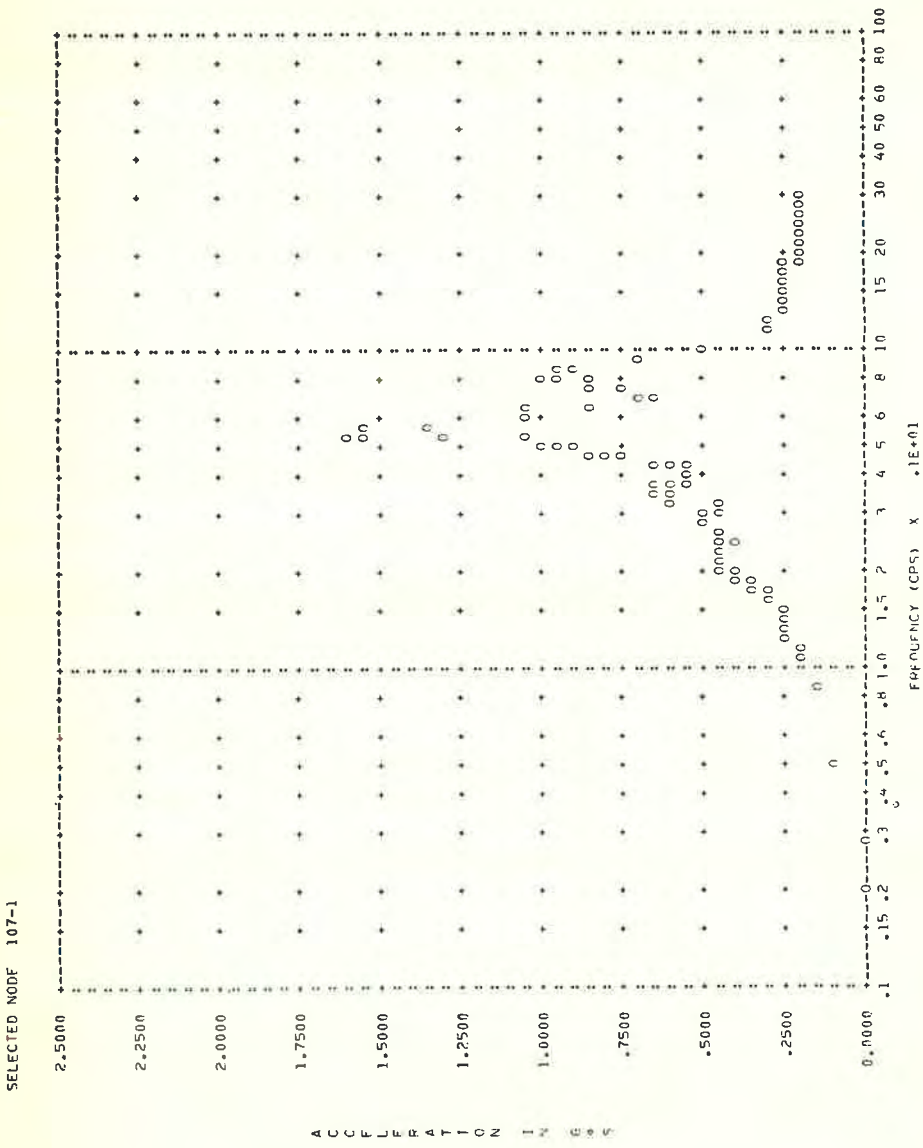
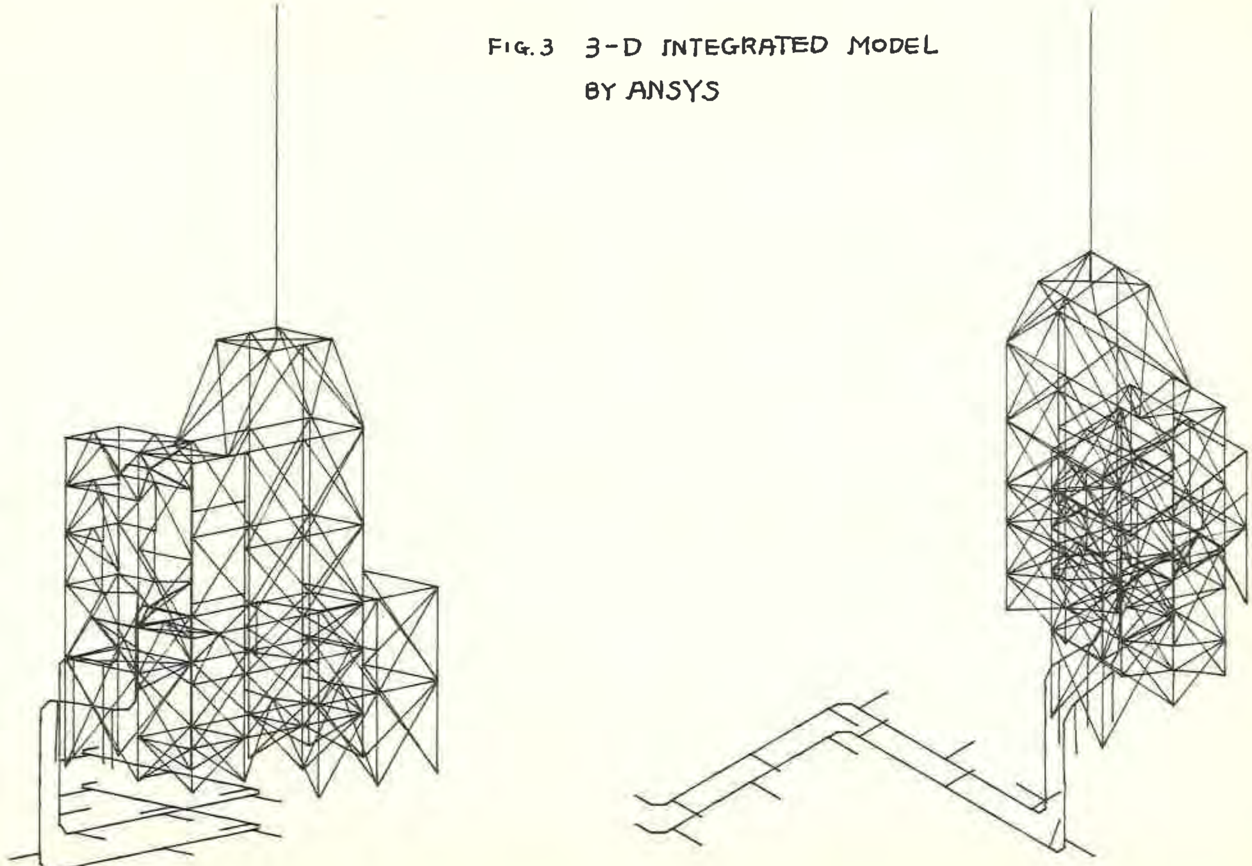


FIG. 3 3-D INTEGRATED MODEL BY ANSYS



BACK VIEW

FRONT VIEW

TABLE 1 EXAMPLE NO. 1(ANSYS) K7/6

MODAL PARTICIPATION FACTORS

MODE	FREQUENCY	P.FACTOR	MODE COEF.	EQUIV. MASS
1	5.586	6.632	3.565	8.098
2	10.13	5.966	.3657	2.965
3	10.15	-.7641	.4659E-01	.6938
4	11.12	.3054	.1383E-01	.5503
5	11.64	6.450	.2439	4.217
6	12.61	-.5029E-01	.1459E-02	.3990
7	13.31	.4512E-01	.1113E-02	.3475
8	13.89	.6903	.1480E-01	.4408
9	14.03	.2771	.5729E-02	.4050
10	14.32	-2.664	.5108E-01	4.988
11	14.94	-.7148E-01	.1156E-02	.4781
12	15.94	-.1917	.2690E-02	.4128
13	16.82	-2.581	.3219E-01	4.213
14	16.85	.3063E-01	.3806E-03	.3702
15	17.43	.2813	.3228E-02	.3865
16	17.69	-.9521	.1054E-01	3.451
17	19.15	-.5244E-01	.4727E-03	.3745
18	19.24	1.366	.1215E-01	4.336
19	19.78	1.483	.1219E-01	5.486
20	20.77	.9667E-01	.7238E-03	.3975
21	21.00	-.6054	.4451E-02	2.982
22	21.57	-.2349	.1651E-02	.5136
23	21.88	-.3386E-01	.2324E-03	.3898
24	21.91	.1342	.9190E-03	.3980
25	22.62	-.6491E-01	.4207E-03	.5641
26	23.10	.3433E-01	.2145E-03	.4927
27	23.97	.1194	.6984E-03	.4965
28	25.80	.1364E-01	.6952E-04	3.095
29	25.83	.7538	.3833E-02	.3584
30	27.13	.7573	.3493E-02	.3471
31	27.13	-.9916E-02	.4573E-04	.4829
32	28.09	.3331	.1429E-02	.3698
33	28.72	-2.123	.8689E-02	3.462
34	30.03	.4197	.1555E-02	3.405
35	30.62	-.1966E-01	.6963E-04	.5691
36	31.57	.8686	.2859E-02	5.576
37	33.56	-.6138	.1727E-02	.5178
38	34.44	-.7428	.1946E-02	.5854
39	34.55	-2.134	.5538E-02	1.687
40	38.11	-.6606E-02	.1260E-04	.2553
41	38.12	.7615E-04	.1452E-06	.2553
42	38.32	.4108	.7685E-03	3.091
43	38.92	1.587	.2805E-02	4.431
44	41.81	.6326E-01	.9632E-04	.3532
45	42.20	-.4310E-01	.6502E-04	.3834
46	44.55	.1853	.2648E-03	.2649
47	46.02	-.7351E-01	.1017E-03	.2134
48	46.30	-.4513E-03	.6206E-06	.2119
49	46.30	.1957E-02	.2690E-05	.2119
50	47.26	.6540	.8809E-03	1.746
51	51.84	-.2334E-01	.2866E-04	.3755
52	52.25	1.253	.1527E-02	3.936
53	53.93	-.3000E-02	.3542E-05	.3426
54	55.01	-.9998	.1157E-02	4.095
55	58.21	-.2829	.3094E-03	2.581
56	58.49	.9130	.9937E-03	3.372
57	59.39	.4941	.5296E-03	3.733
58	61.19	.2719E-01	.2829E-04	.2692
59	61.20	-.1227E-01	.1276E-04	.2690
60	62.05	.9619	.9869E-03	.6716

TABLE 2 EXAMPLE NO.1 (ANSYS)

```
OBE HORI. RESPONSE SPECTRUM AT EL 530 MTS E-W -1 FROM ODT B-3225


***** ELEMENT STRESSES *****    MODE =   1    FREQUENCY =   5.56137


3-D PIPE     1   NODES  400  401   MATERIAL  2 A.TEMP=    70.  O.TEMP=  70.0  I.TEMP=  70.0
1 DIR=    -3. BEND=      0. ST=      0. SP=      0. SF=     16. TH=      0. SMAX=      0. SMIN=    -3. 2MXSR=    3.
2 DIR=    -3. BEND=     95. ST=      0. SP=      0. SF=     16. TH=      0. SMAX=     93. SMIN=   -98. 2MXSR=   98.

FORCES ON NODE  400  -.182423E+02  -.592421E+01   .189424E+00   .595080E-13   .284217E-13   .227374E-12
FORCES ON NODE  401   .182423E+02   .592421E+01  -.189424E+00   .170482E+01  -.284217E-13   .164180E+03

3-D BEAM     2   NODES  302  301   MATERIAL  1   TEMP=    70.   TTOPZ=  70.0   TBOTY=  70.0   DIR.STR.=    -5.
BZ1=    -0.  BY1=     -0.  MAX1=     -5.  MIN1=     -5.    BZ2=     0.  BY2=      0.  MAX2=     -5.  MIN2=    -5.
FORCES ON NODE  302   .344598E+02  0.            0.            0.            0.            0.
FORCES ON NODE  301  -.344598E+02  0.            0.            0.            0.            0.

3-D PIPE     3   NODES  800  801   MATERIAL  2 A.TEMP=    70.  O.TEMP=  70.0  I.TEMP=  70.0
1 DIR=    -3. BEND=      0. ST=      0. SP=      0. SF=     18. TH=      0. SMAX=     -0. SMIN=    -3. 2MXSR=    3.
2 DIR=    -3. BEND=    104. ST=      0. SP=      0. SF=     18. TH=      0. SMAX=    101. SMIN=  -107. 2MXSR=  107.

FORCES ON NODE  800  -.199035E+02  -.645647E+01   .119522E+00   .341061E-12  0.            .454747E-12
FORCES ON NODE  801   .199035E+02   .645647E+01  -.119522E+00   .107570E+01  0.            .179132E+03

3-D PIPE     4   NODES  801  802   MATERIAL  2 A.TEMP=    70.  O.TEMP=  70.0  I.TEMP=  70.0
1 DIR=    -3. BEND=    104. ST=     -0. SP=      0. SF=     16. TH=      0. SMAX=    101. SMIN=  -107. 2MXSR=  107.
2 DIR=    -3. BEND=     12. ST=     -0. SP=      0. SF=     16. TH=      0. SMAX=      9. SMIN=   -15. 2MXSR=   15.

FORCES ON NODE  801   .176929E+02  -.645647E+01   .119522E+00  -.107570E+01  -.170530E-12  -.179132E+03
FORCES ON NODE  802  -.176929E+02   .645647E+01  -.119522E+00   .215140E+01   .170530E-12   .198955E+02

3-D BEAM     5   NODES  702  701   MATERIAL  1   TEMP=    70.   TTOPZ=  70.0   TBOTY=  70.0   DIR.STR.=    -5.
BZ1=    -0.  BY1=     -0.  MAX1=     -5.  MIN1=     -5.    BZ2=     0.  BY2=      0.  MAX2=     -5.  MIN2=    -5.
FORCES ON NODE  702   .375964E+02  0.            0.            0.            0.            0.
FORCES ON NODE  701  -.375964E+02  0.            0.            0.            0.            0.

3-D PIPE     6   NODES  401  402   MATERIAL  2 A.TEMP=    70.  O.TEMP=  70.0  I.TEMP=  70.0
1 DIR=    -3. BEND=     95. ST=      0. SP=      0. SF=     15. TH=      0. SMAX=     93. SMIN=   -98. 2MXSR=   98.
2 DIR=    -3. BEND=     11. ST=      0. SP=      0. SF=     15. TH=      0. SMAX=      8. SMIN=   -13. 2MXSR=   13.

FORCES ON NODE  401   .162175E+02  -.592421E+01   .189424E+00  -.170482E+01   .568434E-13  -.164180E+03
FORCES ON NODE  402  -.162175E+02   .592421E+01  -.189424E+00   .340963E+01  -.568434E-13   .182229E+02

  ELBOW      7   NODES  402  403   MATERIAL  2 A.TEMP=    70.  O.TEMP=  70.0  I.TEMP=  70.0
1 DIR=    -3. BEND=     12. ST=      0. SP=      0. SF=     15. TH=      0. SMAX=      9. SMIN=   -15. 2MXSR=   15.
M DIR=    -7. BEND=     45. ST=      1. SP=      0. SF=      7. TH=      0. SMAX=     38. SMIN=   -52. 2MXSR=   52.
2 DIR=    -7. BEND=     48. ST=      1. SP=      0. SF=      5. TH=      0. SMAX=     41. SMIN=   -56. 2MXSR=   56.

FORCES ON NODE  402   .162175E+02  -.592421E+01   .189424E+00  -.340963E+01   .125104E-12  -.182229E+02
FORCES ON NODE  403  -.162175E+02   .592421E+01  -.189424E+00   .511445E+01   .170482E+01  -.744167E+02
```

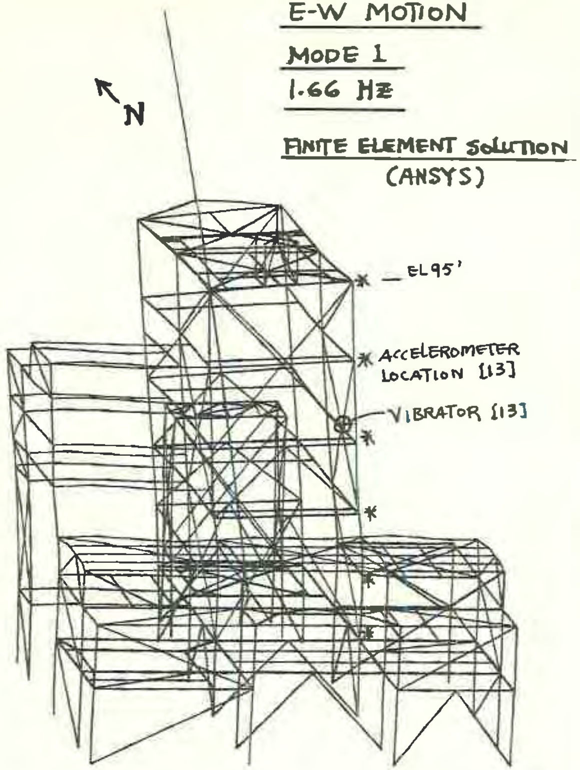


FIG. 4 & FIG. 5a

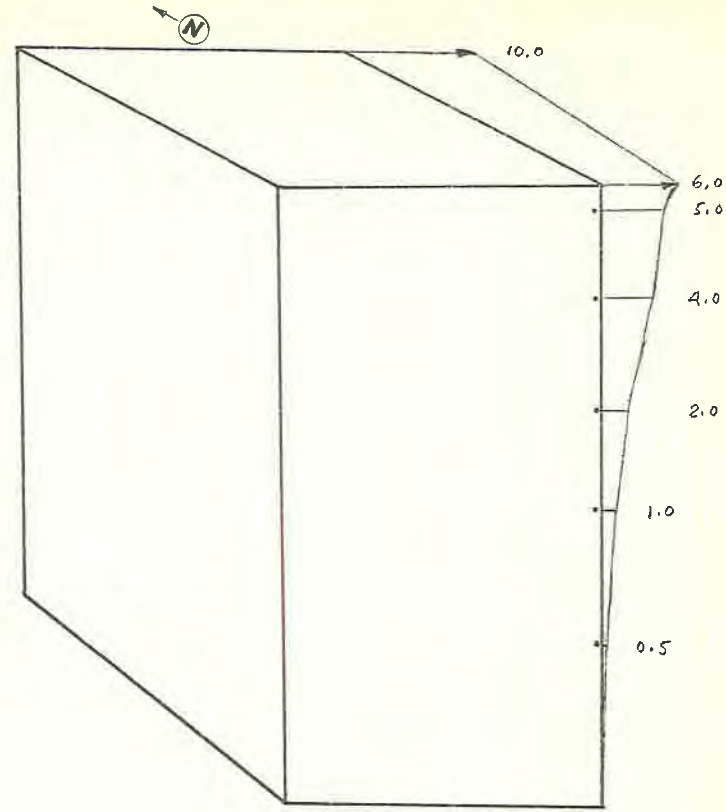


FIGURE 5b

FIRST DOMINANT EW MODE (**EXPERIMENTAL** [13])
1.80 Hz @ 3.5%

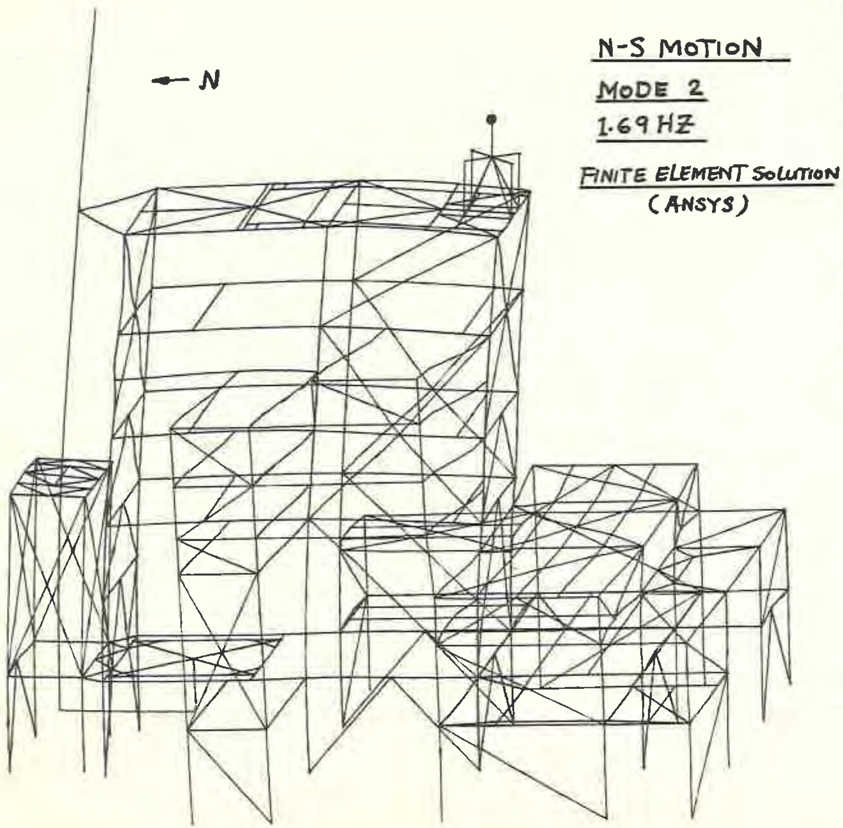


FIG 6a

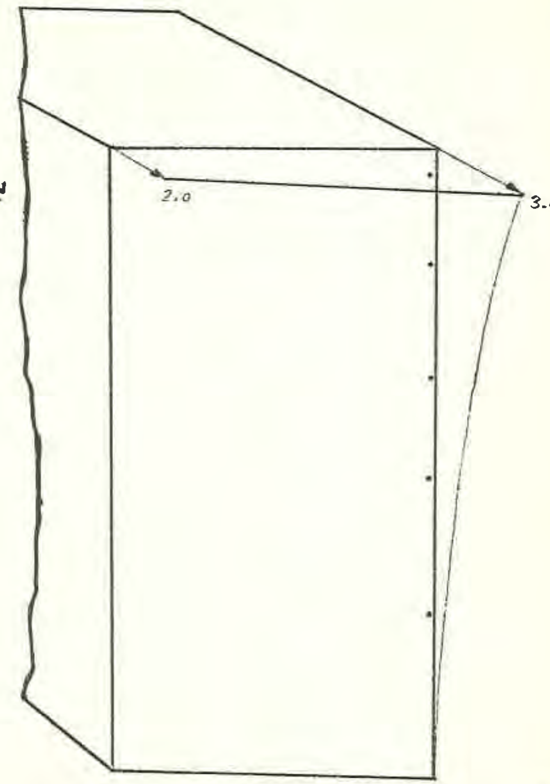


FIGURE 6b

FIRST DOMINANT NS MODE (EXPERIMENTAL)
1.55 Hz @ 4.0% [13]

K7/6

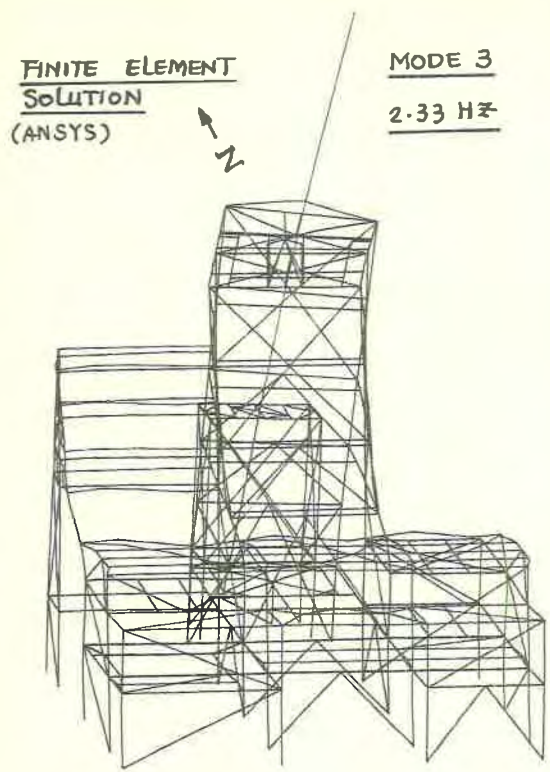


FIG 7a

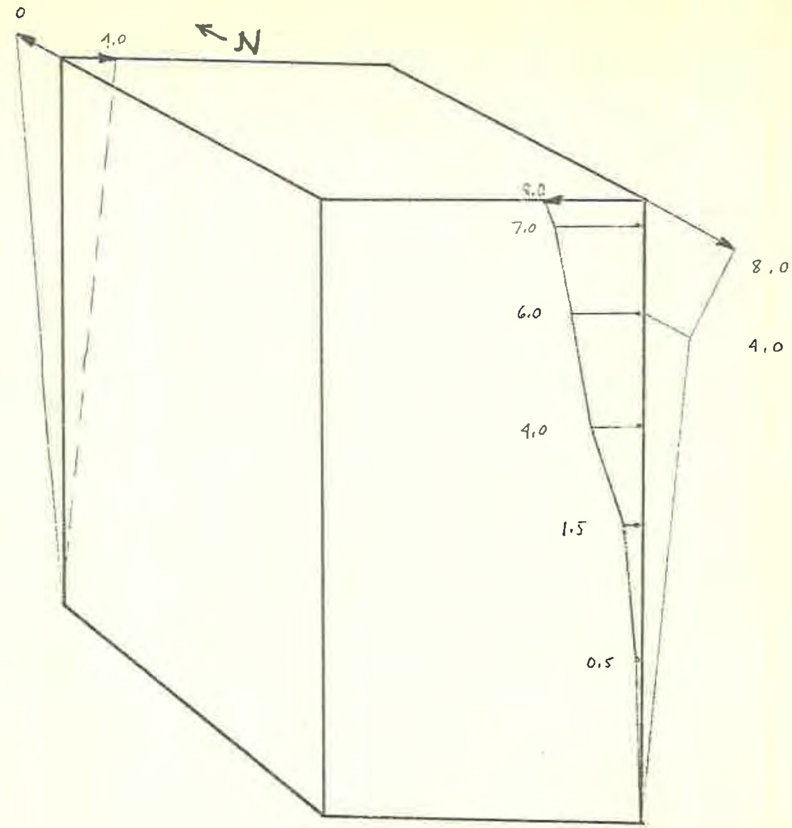


FIGURE 7b

FIRST DOMINANT TORSIONAL MODE (EXPERIMENTAL)
2.90 Hz @ 2.5% [13]

k7/6