The Vibration Analysis of a Large Structural Model Using Different Methods of Analysis

P. Varpasuo

Helsinki University of Technology, Department of Civil Engineering, Rakentajanaukio 4A, SF-02150 Espoo 15, Finland

Abstract

This report is a continuation for the analysis of a reactor containment building for aircraft crash load. In the original analysis the computation was carried out using solely direct integration method. Later we obtained a possibility to investigate this problem by using CRAY-1 super computer and modesuperposition method. In the first place the influence of the mesh density of the finite element and finite difference meshes was considered. Two different mesh densities were used. A model of approximately one thousand degrees of freedom was analysed using both finite difference and finite element methods. The finite element analysis of the 1000 DOF model was made by three different programs, namely, STARDYNE, SAPIV and ANSYS. The ANSYS program did not appear efficient in this type of problem and it was dropped in further computations. The finite difference analysis was made by STAGS program and the displacement history for load application point in the model was in reasonable agreement with the results calculated by finite element models. At this phase of the analysis it was found out that the data preparation for STAGS program was quite complicated for this type of complex three dimensional model and it was also dropped in further computations. Next a final structural model of the containment building with 4200 degrees of freedom was prepared. It consisted of beam and flat shell elements and was analysed for aircraft crash load using Wilson theta and Newmark direct integration methods and SAPIV and STARDYNE programs on Honeywell H66/17 and CDC Cyber 175 computers. The same model was analysed by modesuperposition method with 25 modeshapes using subspace iteration mode extraction method of SAPIV program on CRAY-1 computer. In points near the level of the foundation slab the displacement histories are in reasonable agreement with each other. Higher up, the displacements calculated by direct integration and by modesuperposition differ strongly. Some kind of agreement in displacements could be achieved, if the number of modeshapes in modesuperposition analysis would be more than three hundred.

1. Introduction

The opinions about the effectiveness and accuracy of different dynamic analysis methods and geometric discretation methods applied to large structural models loaded by complicated loadings vary widely.

In this study we put the aim to clarify this point using the problem of an aircraft crash to the containment building as a test case. This problem seems to be suitable for the purpose because the nature of loading is complicated and it has a high frequency content which requires the inclusion of frequencies up to 100 hertzes in to the analysis. Further, in order to discretize the model appropriately the density of the mesh for the analysis has to be carefully calibrated.

The mesh density calibration and also the comparison of effectiveness of different analysis programs and finite element method versus finite difference method was done using a test model with one thousand degrees of freedom.

On the basis of this analysis the final model was formed and the amount of degrees of freedom was fixed to 4200. Also finite element analysis passed difference analysis and SAPIV program was selected as a main tool because it was an in-house program and STARDYNE program was used to some verification purposes. The bulk of dynamic response calculations were done using direct integration techniques both in the case of the test model and for the final analysis model. The five lowest frequencies were determined for the test model. For the final model the 25 lowest frequencies and modeshapes were determined and also the response calculations for the displacements of some degrees of freedom were carried out using modesuperposition method.

2. The Investigation of the Test Model

The test model consisted of three concentric cylinders at the radiuses R = 4.9, 23.25 and 38.5 m and between the levels -5.70 and +27.00 m. These cylinders were joined with three annular plates at levels -5.7, +9.5 and +27.0. The middle cylinder at radius 23.25 continued above the level +27.00 until the level +45.5 and was topped with semispherical cupola, the apex of which was at the level +68.75. All wall thicknesses in the model were 1.5 m and only one half of the containment building between directions 0° and 180° was modelled and symmetry conditions were applied at the boundary. The discretation interval in circumferential direction was 15°. All nodes at the foundation level -5.70 were fixed against translations and rotations.

The material properties of concrete were taken to be as follows: Young's modulus equaled to 30 000 MN/m^2: Poisson's constant equaled to 0.15 and the volume weight to 0.023 MN/m^3. The Rayleigh damping constants were taken to be 2.99 and 0.000151 which correspond to 5 % damping at 5 and 100 Hz. The time step in direct integration was taken to be 1 ms. The concentrated horisontal impact load at the level +45.5 and in the symmetry plane had following characteristics: from zero to 5 ms: linear rise to 26.25 MN; from 5 to 26 ms: constant value of 26.5 MN; from 26 to 40 ms: linear rise to 52.5 MN; from 40 to 53 ms: constant value of 52.5 MN; from 53 to 72 ms: linear fall to zero. The results of the analysis of the test model can be summarized in the table where the horizontal displacement of the load application point is tabulated for five first steps in the response history as function of used program, discretation method, dynamic analysis type and direct integration scheme.

Time ms	ANSYS Fem Houbolt Nodes 157 El. 168 DOF 585 (66 Master DOFs) 145 r.m.s. wavefront (m)	SAPIV Fem Wilson Theta Nodes 157,170,196 El. 168,180,204 DOFs 651,723,867 1/2 band width 228,375,228 (m) (m) (m)	STARDYNE Fem Newmark Nodes 157 El. 168 DOFs 651 1/2 band width 105 (m)	STAGS Fdm Trapez. Nodes 440 Seg. 280 DOFs 1239 avg.band width 139 (m)
1	.246E-S	.054E-4,.075E-4,.150E-4	.079E-4	.134E-4
2	.110E-4	.427E-4,.059E-3,.117E-3	.471E-4	.080E-3
3	.300E-4	.141E-3,.197E-3,.379E-3	.147E-3	.252E-3
4	.063E-3	.329E-3,.458E-3,.085E-2	.337E-3	.057E-2
5	.111E-3	.063E-2,.087E-2,.156E-2	.064E-2	.109E-2

The costs in comparable units for the analyses were ANSYS: 4500 units; SAPIV: 1450, 1830, 1415; STARDYNE: 950; STAGS: 1650 units. Also the five lowest eigenfrequencies of the test model were calculated by SAPIV and they were 4.8, 7.3, 8.3, 9.8 and 10.6 Hz. The cost of the mode extraction run was 5950 units.

On the basis of above facts the SAPIV program was selected for analysis because it gave comparable results with STARDYNE and STAGS and could be made as cost effective as STARDYNE with the user band with optimization. STAGS seemed to be comparable to STARDYNE and SAPIV but it was dropped out because of modelling difficulties when the structure had to be discretized more in detail. ANSYS required a lot of core memory and its model seemed to be far too stiff (because of its master degrees of freedom system, only 66 MDOFs could be included in analysis) and the analysis cost was a way out of the area of the other programs. Finally, because of the cost of mode extraction seemed to be higher than that of direct integration, the direct integration method was selected for dynamic analysis. In figure 1 the horizontal displacement history of the load application point from zero to 280 milliseconds is presented for the test model and calculated with the aid of STAGS, SAPIV and STARDYNE programs.

3. The Investigation of the Final Analysis Model

The mesh density for the final model was determined from the available computation budget, from one side, and from the minimum mesh density for cylinders and annular plates in order to plot meaningful stress resultant contour plots, from the other side. The mesh density in circumferential direction was taken to be the same as for the test model and the amount of nodes in all radial sections was selected to be 60 so that the total amount of modes was 780 and total amount of degrees of freedom was 4188. The half band width in the stiffness matrix was 391. The plot of the final analysis model is presented in /1/. Only the translational degrees of freedom in the model did have a specified mass. The total weight of the final model was 202 000 tons. The foundation soil in the model was an elastic half space which was supposed to begin from below the pile foundation of ten meters height. The Young's modulus for the soil was 4000 MN/m^2; the volume weight 0.018 MN/m^3 and Poisson's ratio 0.3. The Rayleigh type damping was assumed to be 6 % at 5 Hz and 20 % at 100 Hz for direct integration method and 4 % for all modeshapes in modesuperposition method. The thicknesses for concrete walls varied from half meter to one and half meter. The dynamic Young's modulus for concrete was 45 000 MN/m^2; the volume weight 0.023 MN/m^3 and Poisson's ratio 0.15. The impact load for the final model was the same as for the test model. The direct integration for final model was carried out by Wilson theta method and 2 ms step length. In modesuperposition method the 25 first frequencies and modeshapes were determined and the frequencies varied between 1.2 Hz and 16.4 Hz. The estimate for highest frequency in the model was 519 Hz. The displacement time histories in free perpendicular directions for a point at level +45.5 and 90^o from the load application point in circumferential direction have been presented in fig. 2. It can be concluded from the plots that the agreement of the results calculated by direct integration method and modesuperposition method vary strongly depending on the displacement direction.

Some data about the computation times of different parts of dynamic analyses by some known mainframe computer has been given in the following table:

Computer	Analysis Type	CPU Time	Program
H66/17	Mode Extraction (3 modes)	15 hours	SAPIV
H66/17	Stiff.Matr.Decomposition	7.5 hours	SAPIV
H66/17	Direct Integration (75 steps)	4.5 hours	SAPIV
CYBER 175	Mode Extraction (25 modes)	80 minutes	STARDYNE
CYBER 205	Mode Ext.& Response (3 modes)	450 seconds	SAPIV
CRAY-1	Mode Extraction (3 modes)	167 seconds	SAPIV
CRAY-1	Mode Ext.&Response (25 modes)	908 seconds	SAPIV

4. Conclusion

The displacement response in load application direction seems to be reasonably accurately calculable by few modeshapes in modesuperposition method for all levels in reactor building outside of the load application point itself. The accuracy of the modesuperposition deteriorates rapidly in vertical displacements and even more rapidly in displacements perpendicular to load application direction. It might be estimated that the sufficient amount of modeshapes for reasonably accurate determination of response in vertical direction could be one hundred and maybe from 300-500 for horizontal displacements perpendicular to load application direction for the structural model in the investigated case.

5. References

/1/ P. Varpasuo, J. Kenttälä: "The Analysis of Global Effects of an Aircraft Crash to the Containment Building of a Nuclear Power Plant", Proc. of SMIRT-6, Paris, 1981, Paper J 9/10.

/2/ SAP4 Structural Analysis Program, Users Manual, Earthquake Engineering Research Center, Berkeley, 1973.

/3/ MRI STARDYNE, Users Manual, Mechanics Research Inc., Minneapolis, 1980.

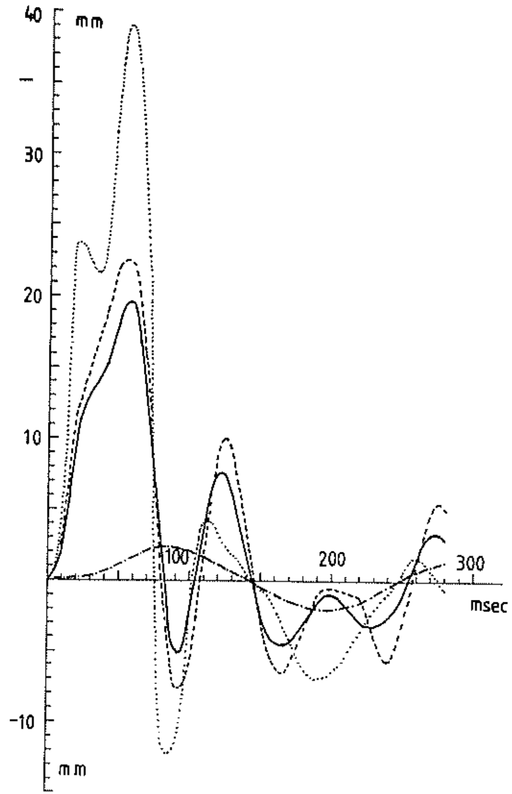

Figure 1. The Displacement Response of the Test Model in Impact Point to the Load Application Direction

Legend:	STAGS	....
	STARDYNE	---------
	SAPIV (651 DOFs)	————
	SAPIV (5 modes)	-.-.-.-.-

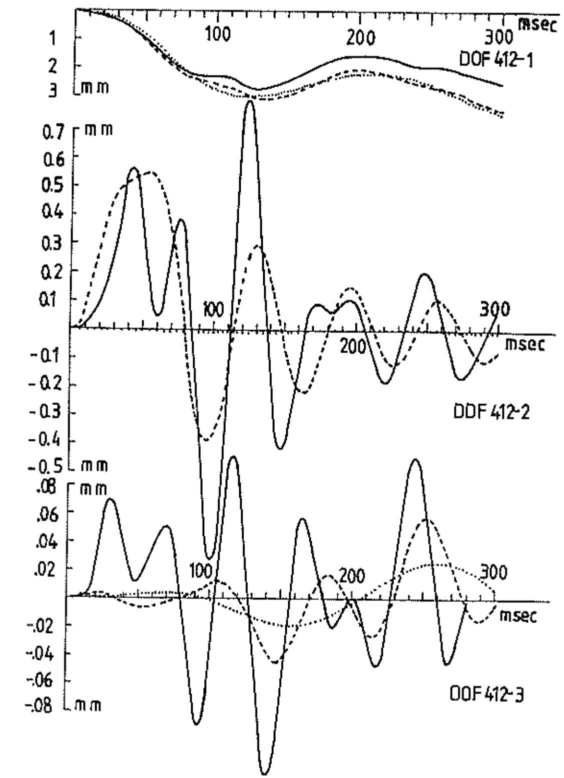

Figure 2. The Displacement Responses of the Final Model at the Level +45.5 and 90^{o} Degrees from the Load Application Point in the Circumferential Direction

Legend:	Direct Integration	————
	Mode Superposition (25 modes)	--------
	Mode Superposition (3 modes)	....

DOF 412-1 = Load Application Direction

DOF 412-2 = Perpendicular Direction

DOF 412-3 = Vertical Direction