EFFECT OF THE COUPLING OF MOTION ON THE PERFORMANCE OF MACHINE FOUNDATION

A DISSERTATION

submitted in partial fulfilment of the
requirements for the award of the degree
of
MASTER OF ENGINEERING
in
EARTHQUAKE ENGINEERING
(With Specialization in Soil Dynamics)

By

KRISHNA NANDAN SHARMA

DEPARTMENT OF EARTHQUAKE ENGINEERING
UNIVERSITY OF ROORKEE
ROORKEE-247 667 (INDIA)

MARCH, 1999

CANDIDATE'S DECLARATION

I hereby declare that the work which is being presented in the dissertation entitled **"EFFECT OF THE COUPLING OF MOTION ON THE PERFORMANCE OF MACHINE FOUNDATION"** in partial fulfilment of the requirements for the award of the degree of **"MASTER OF ENGINEERING IN EARTHQUAKE ENGINEERING"** with specialization in **SOIL DYNAMICS,** submitted in the Department of Earthquake Engineering, University of Roorkee, Roorkee is an authentic record of my own work during a period of about 5 months from November, 1998 to March, 1999 under the guidance of **Dr. Ramakrishna, Scientist,** Central Building Research Institute, Roorkee, India and **Dr. S. Bandyopadhyay, Associate Professor,** Department of Earthquake Engineering, University of Roorkee, Roorkee, India.

The matter embodied in this dissertation has not been submitted by me for the award of any other degree or diploma.

PLACE : Roorkee
DATE : 30 March, 1999

Krishna Nandan Sharma
KRISHNA NANDAN SHARMA

This is to certify that the above statement made by the candidate is correct to the best of our knowledge.

S. Bandyopadhyay 30/3/99
Dr. S. Bandyopadhyay
Associate Professor
Department of Earthquake Engg.
Roorkee - 247 667

Ramakrishna
Dr. Ramakrishna
Scientist,
Central Building Research Institute
Roorkee - 247 667

ACKNOWLEDGEMENT

It is my great pleasure to express my indebtedness to my thesis guides Dr. Ramakrishna and Dr. S. Bandyopadhyay, whose encouragements and supports have made it possible to complete this work in its present form.

I wish to acknowledge the help given by Dr. S.Basu, Prof. & Head, Deptt.of Earthquake Engg., and Prof. D.K. Paul, O.C. computer for providing computer laboratory facilities and Mr. R.N. Dubey & Dr. M. Shrikhande for their helps.

I am grateful to the Director, CBRI, Roorkee for granting the kind permission to conduct the experimental work at CBRI, Roorkee. At the sametime, I express my deep gratitudes to Sri M.P. Jaisingh, Scientist Coordinator, Structural Engineering Division, CBRI, Roorkee for the permission to use the infrastructural facilities available with Building Dynamics Laboratory (BDL). I am obliged to Mr. Ajay Chaurasia, Scientist, BDL and Mr. Jalaj Parashar, STA, BDL, for their help during testing programme. I want to acknowledge Mr Dinesh Chandra, Scientist, BPPP Division and his team for helping in casting and curing of the block. The help rendered by GED, CBRI is highly acknowledged.

I, specially, wish to thank to my friends and all staff members of the department of Earthquake Engg. with Lab. Asstt. Sri N.K. Sharma who have been always helpful to me.

Krishna Nandan Sharma

KRISHNA NANDAN SHARMA

ABSTRACT

Machine foundations, subjected to dynamic loading vibrate in different modes. Sometimes, these different modes of vibration are coupled either due to an error in placing of the machine on the foundation or due to interactions with the other foundations lying nearby. This phenomenon of coupling of motions in machine foundations affects their responses, because of the interaction between the waves propagating through the soil-medium. The properties of waves in the soil-medium are functions of various parameters viz., type and properties of soil, layering of soil-medium etc.

The frequency and amplitude of vibration of machine foundation due to coupled motion is caused by an eccentric loading and interaction with other foundations. For the study of the coupling of motion between machine foundations ANSYS software has been used which employs the finite element procedure for analysis. The capabilities and the limitations of the computer package in analysing such problem have been discussed.

The effect of coupling of motion due to an eccentric loading has been studied by performing experiments also. The block vibration tests on a concrete block of size 2.25m x 0.75m x 0.70m has been done by placing the oscillator at the centre as well as at an eccentric point. The same block has also been analysed by ANSYS package. The comparison of results obtained from the two methods leads to some important concluding remarks.

LIST OF FIGURES

Fig. No.	Description of the figures
Fig. 1.1	First mode of bending of a cantilever beam
Fig. 1.2	Second mode of bending of a cantilever beam
Fig. 1.3	Third mode of bending of a cantilever beam
Fig. 1.4	First mode of axial compression of a cantilever beam
Fig. 1.5	Second mode of axial compression of a cantilever beam
Fig. 2.1	A model showing block foundation with surrounding soil
Fig. 2.2	Frequency versus amplitude of vibration of a machine foundation
Fig. 3	Uniform soil below the foundation
Fig. 4	Layered soil below the foundation
Fig. 5.1	Foundations located at a distance of 3B (= 7.32m)
Fig. 5.2	Meshing pattern of the machine foundation- soil system.
Fig. 5.3	Mode of vibration of machine foundation at on operating frequency (= 7.5 Hz).
Fig. 6.1	Foundations located at a distance of 2B (= 4.88 m)
Fig. 6.2	Meshing pattern of the machine foundation-soil system.
Fig. 6.3	Mode of vibration of machine foundation at an operating frequency (= 7.5Hz).
Fig. 7.1	Foundations located at a distance of B (= 2.44 m)
Fig. 7.2	Meshing pattern of the machine foundation – soil system.
Fig. 7.3	Mode of vibration of machine foundation at an operating frequency (= 7.5 Hz)

Fig. 8.1 Frequency versus amplitude in Y-direction of foundation A_1 (For D = 3B)

Fig. 8.2 Frequency versus amplitude in X-direction of foundation A_1 (For D= 3 B)

Fig. 8.3 Frequency versus amplitude in Y – direction of foundation A_2 (For D= 3 B)

Fig. 8.4 Frequency versus amplitude in X-direction of foundation A_2 (For D = 3B)

Fig. 9.1 Stress in X – direction (For D = 3B)

Fig. 9.2 Stress in Y-direction (For D = 3B)

Fig. 10.1 Strain in X – direction (For D = 3B)

Fig. 10.2 Strain in Y-direction (For D = 3B)

Fig 11.1 'LAZAN' type mechanical oscillator

Fig 11.2 Motor

Fig 11.3 Speed Control Unit

Fig 12.1 Duel Channel FFT Spectrum Analyser

Fig 12.2 Vibration meter with pick-up

Fig 12.3 An assembly of FFT spectrum analyser, vibration meter and pick-up

Fig 13 A concrete block of dimension of 2.25m x 0.75m x 0.7m

Fig. 14 A diagram of showing DC Motor, mechanical oscillator, flexible shaft, teco generator.

Fig. 15 A machine foundation of dimension 2.25m x 0.75 m x 0.7m with base plate

Fig. 16 A machine foundation of dimension 2.25m x 0.75m x 0.7m with oscillator.

Fig. 17 A vertical vibration test with oscillator at the centre of the block.

Fig. 18 A horizontal vibration test with oscillator at the centre of the block.

Fig. 19 A vertical vibration test with oscillator at the eccentric point

Fig. 20 A horizontal vibration test with oscillator at the eccentric point

Fig. 21 Frequency versus amplitude for central loading when the oscillator was placed for vertical mode of vibration.

Fig. 22 Frequency versus amplitude for eccentric loading when the oscillator was placed for vertical mode of vibration.

Fig. 23 Frequency versus amplitude for central loading when the oscillator was placed for horizontal mode of vibration.

Fig. 24 Frequency versus amplitude for eccentric loading when the oscillator was placed for horizontal mode of vibration.

Fig. 25 A model for FEM analysis for experimental block (dimension 2.25m x 0.75m x 0.7m)

Fig. 26 Element pattern of the machine foundation and soil system with boundary condition.

Fig. 27 Vertical mode of vibration of block at frequency (=7.5Hz).

Fig. 28 Horizontal and rocking mode of vibration of block at frequency (=6.426Hz).

Fig. 29 Operating frequency (Hz) versus force (KN) produced by the oscillator with ($e = 90^{\circ}$)

FIGURE 1 Typical steps and files in a frontal solution

FIGURE 2 Typical steps and files in a JCG solution.

LIST OF TABLES

Table No.	Description of the Tables	Page No.
Table 1	Required informations for analysis	78
Table 2	Comparison of results for cantilever beam	78
Table 3	Material properties	79
Table 4	Effect of different types of damping on the amplitude of vibration in case of sandy soil.	79
Table 5	Material properties of concrete and different soils	80
Table 6	Input data for ANSYS analysis	80
Table 7	Influence of soil properties on the performance of Machine foundation.	81
Table 8	Effect of layered soil on the performance of machine foundation.	81
Table 9	Effect of layered soil on the performance of machine foundation.	82
Table 10	Effect of eccentric loading on the amplitude of vibration of foundation at an operating frequency of 7.5 Hz.	82
Table 11	Effect of distance between foundations on the coupling of motion	83
Table 12	Effect of distance between foundations on the coupling of motion.	84
Table 13	Effect of phase-change of loading on the foundation interaction by coupling of motion for sandy soil.	85

Table 14 Amplitude of vibration and resonant frequency for both the concentric loading as well as eccentric loading from experiment. 86

Table 15 Amplitude of vibration and resonant frequency for both the concentric loading as well as eccentric loading obtained from ANSYS program. 86

TABLE 1 Structural Element types *APPENDIX*

TABLE 2 DOF constraints available in each discipline

TABLE 3 Commands for DOF constraints

TABLE 4 Primary and derived data for different discipline

TABLE 5 Load commands for a harmonic analysis

TABLE 6 Loads step options

NOTATIONS

1.	A	Cross-sectional area of a cantilever beam.
2.	A_b	Base area of a machine foundation
3.	B	Half of the length of the foundation
4.	C	Damping coefficient
5.	C_u	Coefficient of elastic uniform compression
6.	C_θ	Coefficient of elastic non uniform compression
7.	C_τ	Coefficient of elastic uniform shear
8.	C_ψ	Coefficient of elastic non-uniform shear
9.	D	Spacing between foundations
10.	D_m	Maximum largest difference occuring for all elements
11.	E	Young's modulus of elasticity
12.	e	Eccentricity of the rotating masses of an oscillator.
13.	e_x	Eccentricity of loading in X-direction
14.	$\{\phi_i\}$	Mode shape vector
15.	ϕ_y	Moment of inertia of the foundation about Y-axis.
16.	ϕ_{oy}	Total moment of inertia of the foundation about Y-axis.
17.	I_o	Moment of inertia of the foundation about vertical axis
18.	J_o	Polar moment of inertia of the foundation about vertical axis.
19.	$[K]$	Stiffness Matrix

20.	L	Length of a cantilever beam
21.	$[M]$	Mass matrix
22.	m	Mass per unit length
23.	m_e	Eccentric mass in an oscillator
24.	m	Mass of the machine and foundation
25.	P_0	Force generated by the oscillator
26.	$\{Q\}$	Nodal force Vector
27.	$\{q\}$	Displacement vector
28.	$\{R\}$	Load Vector
29.	$\{r\}$	Nodal displacement Vector
30.	ρ	Mass density
31.	ω_z	Natural frequency of foundation about Z-axis
32.	ω_i	Natural circular frequency of ith mode.
33.	ω_m	Operating frequency of the machine.
34.	θ	Angular displacement of footing.

CONTENTS

CHAPTER PAGE NO.

CANDIDATE'S DECLARATION i

ACKNOWLEDGEMENT ii

ABSTRACT iii

LIST OF FIGURES iv

LIST OF TABLES vii

NOTATIONS ix

1. Introduction

1.1 General 1

1.2 Objectives of the study 2

1.3 Layout of the thesis 3

2. Literature Review

2.1 General 4

2.2 Eccentric Loading 4

2.3 Foundation Interaction 6

2.4 Response of the Foundation resting on Layered Soil 8

2.5 Effect of Shape on Torsional Vibration of Block Foundation 9

2.6 Design Criterion 10

3. ANSYS Software

3.1 General 12

3.2 Finite Element Method 12

3.2.1 Introduction 12

3.2.2 Steps in FEM Analysis 14

3.3 Basic Analysis Procedures 19

3.3.1 Steps of Analysis 19

3.3.2 Loading 25

3.3.3 Solution 27

3.3.4 An Overview of Postprocessing 31

3.4 Modelling and Meshing 32
  3.4.1 Overview of Model Generation 32
  3.4.2 Planning Approach for Solving Problems 34
  3.4.3 Meshing the Model 36
  3.4.4 Revising the Model 36
  3.4.5 Direct Generation 38
3.5 Structural Analysis by ANSYS 39
  3.5.1 Overview 39
    3.5.1.1 Elements Used in Structural Analysis 41
    3.5.1.2 Types of Solution Methods 41
  3.5.2 Modal Analysis 41
    3.5.2.1 Model Extraction Method 41
  3.5.3 Harmonic Response Analysis 44
3.6 Conclusion 48

4.0 Bench Mark Problem Solved by ANSYS

4.1 General 49
4.2 Analytical Solution of a cantilever beam 49
4.3 Analysis of the cantilever beam by ANSYS 50
4.4 Comparison of Results of ANSYS with Analytical Method 51
4.5 Problem of a Machine Foundation solved by ANSYS 51

5.0 Parametric Studies on Machine Foundation

5.1 General 53
5.2 Influence of Soil Properties on the Performance of Machine Foundation 53
5.3 Influence of Layered Soil Medium 55
5.4 Effect of Eccentric loading on the Amplitude of vibration. 57
5.5 Foundation Interaction by Coupling of Motion 58
  5.5.1 Effect of Spacing between Foundations 58
  5.5.2 Load Acting on Foundation A_1 only 59
  5.5.3 Loads Acting on both Foundations A_1 and A_2 60

5.5.4 Conclusion 62
5.5.5 Effect of Different Phases of Loading on the Foundations interaction 62

6. Experimental Work

6.1 Introduction 64
6.2 Experimental Setup (As per IS 5249 : 1969) 64
6.3 Present Experimental Work 66
6.3.1 Infrastructural Facilities available at CBRI, Roorkee 66
6.3.2 Testing Procedure 67
6.3.3 Testing Phases 68
6.4 Results and Discussion 71
6.5 Analysis of Experimental Work by ANSYS Program 72
6.5.1 Various steps of Analysis for forced vibration 73
6.5.2 Comparison of the results obtained from the experiment and the results obtained from ANSYS program 74
7.0 Conclusions 75
Tables 78
Figures
Appendix
References

Chapter-1

1. Introduction

1.1 General :

In the field of industrial development, machines are being set on the foundations for production of various useful and essential commodities. The foundations of these machines are subjected to dynamic loading which exhibit entirely different and complicated behaviour. Sometimes, the cost of foundations for heavy machines approaches to the cost of a medium-sized bridge. So, design and construction of a machine foundation becomes a challenging job for engineers. The concept behind designing a machine foundation is to know its amplitude and frequency of vibration. Usually, the amplitude of vibration is kept limited to certain permissible value so that there should not be any damaging effect on the parts of the machine and its foundation. At the same time, it should not cause any unwanted effects on the nearby structures. Similarly, the operating frequency of the machine is kept away from the natural frequency of the foundation, so that there should not be any the occurrence of resonance conditions.

Foundation for machine has different modes of vibration, e.g. vertical, horizontal, rocking, yawning and pitching. When the centroid of the machine, foundation and that of the base area are lying in the some vertical line, the mode of vibration is purely vertical for which simplified methods of analysis are available but as the modes of vibration combine together, the computation of the performance of machine foundation becomes more complicated. Sometimes, there are situations which may occur due to an error in placing the machine on the foundation. In such situations, the modes of vibrations are coupled. This type of problem is very difficult to be solved by conventional method of analysis. Similarly, the responses of one foundation changes due to interaction with another foundation lying nearby. For studying these problems, finite element method suits to be the best.

The finite element method is a numerical tool which requires descretization of a given structure into a network of finite elements and a digital computer to implement the analysis. It requires the formation and solution of the system of algebraic equations, likewise, done in numerical solution of practical problems in structural engineering. The special advantage of this method resides in its suitability for automation of equation formation process and in the ability to represent highly irregular and complex structures and loading conditions.

With the advent of high speed computing machines, the accuracy of computation has been tremendously improved and the computation time has been decreased. Such analysis is more sophisticated and scientific compared to the empirical methods which are now being obsolete. Hence, more and more commercially available packages, like ANSYS, NASTRAN, COSMOS, STAAD, SAP, NISA, ABAQUS etc. are used for analysis and design purposes.

1.2 Objective of the Study :

The objective of this study is to explore the capabilities of finite element method for analyzing the performance of machine foundation due to coupling of motion. Simultaneously, comparing these results with the results obtained from a block vibration test to arrive at final conclusions.

1.3 Layout of the thesis:

Chapter 1 introduces with the problems associated with the of coupling of motion of a machine foundation caused by various reasons like eccentric loading ,an interaction between foundations etc.,

Chapter 2 presents a review of literature on the theoretical and experimental works to find out the responses of the machine foundation due to various effects viz. the effect of eccentric moment, different type of pressure distribution on the base area in contact with the soil.

Chapter 3 represents the various features of the ANSYS program with brief introduction to the finite element method of analysis. Different steps and types of analysis have been dealt with in detail.

Chapter 4 contains some bench mark problems solved by ANSYS program to establish its capability so that it can be used for various types of studies.

Chapter 5 covers some important parametric studies done on the machine foundation, like the effect of different types of soil below the foundation along with the effect of layered soil medium. The effect of coupling of motion caused by an eccetric loading and the interaction between two foundations lying nearby have been studied. The effects of the change of phase of loading on the responses of the machine foundation have been also explored by ANSYS program.

Chapter 6 includes the experimental work that has been done to study the effect of coupling of motion on the performance of machine foundation due to an eccentric loading by performing block vibration tests. The routine tests have also been done.

Chapter 7 draws some important conclusions from the study and some important suggestions have been made for further study.

2. Literature Review

2.1 General:

Machine foundations subjected to different types of dynamic loadings such as harmonic type or impact type, vibrate in different modes. If the loading acts at the central point of the foundation i.e. through the combined centre of gravity of machine and foundation and the center of gravity of the base area of foundation in contact with the soil, lie on the same vertical line, the mode of vibration is simplified namely vertical. There are two approaches for the analysis of machine foundations, e.g. Barkan's spring–dashpot model and Richart's elastic half space theory. Among these two, Richart's elastic half space theory is accepted for design because it gives more compatible results due to consideration of radiation damping in the soil. But, in case of eccentric loading on the foundation, the coupling of different modes of vibration takes place which has been analyzed by Barkan using spring-dashpot model.

Some of the cases where the motion is represented by coupled modes of vibration are presented in the following sections.

2.2 Eccentric Loading:

In case of eccentric loading, vertical, horizontal and rocking vibration are inter coupled and the three coupled natural frequencies ω_1, ω_2 and ω_3 are possible. These are given by the roots of the following expression as suggested by Barkan (1962):

$$\omega_n^2 e_x^2 = \frac{\alpha(\omega_z^2 - \omega_n^2)(\omega_{n1}^2 - \omega_n^2)(\omega_{n2}^2 - \omega_n^2)}{\omega_z^2(\omega_x^2 - \omega_n^2)} \qquad \text{...........(2.1)}$$

where e_x is the eccentricity of the centroid of base area of the foundation measured along the x-axis from the centre of gravity of machine foundation.

$$\omega_z = \sqrt{\frac{K_z}{m}} = \sqrt{\frac{C_u A_f}{m}} \qquad \text{..........(2.2)}$$

$$\omega_{n1} = \sqrt{\frac{1}{2\alpha_y}\left[\omega_{\theta y}^{2} + \omega_x^{2} + \sqrt{\left(\omega_{\theta y}^{2} + \omega_x^{2}\right)^{2} - 4\alpha_y \omega_{\theta y}^{2}\omega_x^{2}}\right]} \qquad \text{............(2.3)}$$

$$\omega_{n2} = \sqrt{\frac{1}{2\alpha_y}\left[\omega_{\theta y}^{2} + \omega_x^{2} - \sqrt{\left(\omega_{\theta y}^{2} + \omega_x^{2}\right)^{2} - 4\alpha_y \omega_{\theta y}^{2}\omega_x^{2}}\right]} \qquad \text{............(2.4)}$$

and, $$\alpha_y = \frac{\phi_y}{\phi_{oy}} \qquad \text{...........(2.5)}$$

$$\omega_{\theta y}^{2} = \frac{(C_\theta I_y - WS)}{\phi_{oy}} \qquad \text{..........(2.6)}$$

$$\omega_x^{2} = \frac{C_\tau A_f}{m} \qquad \text{........... (2.7)}$$

$$\alpha = \frac{\phi y}{\phi 0y} \qquad \text{.......... (2.8)}$$

$$\phi_{0y} = \phi_y + m\bar{z} \qquad \text{........... (2.9)}$$

Cu = Coefficient of elastic uniform compression

C_θ = Coefficient of elastic non-uniform compression

C_τ = Coefficient of elastic uniform shear

S = Distance of centroid of the foundation above the soil.

W = weight of the foundation

I_y = Moment of inertia of the foundation about Y-axis

By solving the above equation (2.1) natural frequencies of vibration of machine foundation subjected to eccentric loading can be obtained.

Sometimes the responses of a foundation are affected by the presence of another foundation when both are subjected to dynamic loadings.

2.3 Foundation Interaction:

It has been recognized that dynamic interaction between structural foundations and the underlying soil plays an important role in their dynamic behaviour under the action of either external forces or seismic waves. For a group of foundation, the presence of adjacent foundation may affect the others further through the interaction effect of the sub-soil. In such a situation, each foundation which diffracts the incident wave field can be regarded as a disturbance producing a secondary wave field affecting the adjacent ones. (Ref.15)

Qian and Cheung (1998) studied the effect of incident seismic waves on the dynamic response of a single flexible foundation. A 8 x 8 mesh descretization was applied to both the soil foundation interface and the flexible foundation. After completion of analysis, it was found that there is no significant difference in the displacement in horizontal direction between a stiff and a soft foundation, while there is a marked decrease in the displacement in vertical direction, at high frequency range as the foundation stiffness increases. It is also observed that for a low frequency, say $\omega_0 = 1.0$ the foundation behaves as a rigid body and its response is virtually independent of the relative stiffness, while for a high frequency, say $\omega_0 = 4.0$ the effect of the foundation stiffness is significant.

In case of horizontally incident SH waves, propagating in the horizontal direction, the horizontal response at the center of the foundation is similar to that of foundations under a Rayleigh wave excitation. It indicates that there is a significant torsional motion of the foundation. In addition, a certain amount of rocking response has been generated and as a result there is considerable out-of- plane displacement.

Cases of two identical square foundations of certain thickness and sizes subjected to either to Rayleigh or SH waves was taken as examples for demonstrating the interaction between the foundations. After analysis, it has been found that for horizontally incident Rayleigh wave displacement both in horizontal direction and vertical direction at the center of foundation I are identical with those corresponding components of foundation II. Their values are very close to those obtained for a single foundation.

The behaviour of two foundations excited by an incident Rayleigh wave with $\theta_H = \pi/2$ was also studied. The responses for the center point of foundation I and foundation II exhibit strong fluctuation around the value corresponding to a single foundation. The intensity of fluctuation seems to be independent of the foundation stiffness. The results also reveal that as the spacing between the foundation decreases, the intensity of the fluctuation increases but the oscillating frequency decreases.

The response of foundation is also affected by the way the pressure is distributed on the soil lying below the foundation. This effect on the rocking vibration has been studied by A. Sridharan, Baidya and Raju et al. (1992) for three types of pressure distribution viz. Rigid, linearly varying and parabolic for weighted average displacement condition using elastic half space theory. To make use of this theory, the necessary parameter, viz. displacement functions, f_1 and f_2 are developed for different contact pressure distributions for rocking mode. In addition, other parameters, viz. static stiffness coefficient, k_ψ; modified displacement functions f_1 and f_2 and damping ratio, D_ψ for analog model are also developed.

From the results it has been found that there is a significant effect of contact pressure distribution on frequency and amplitude at resonance. So, special care should be taken to assume appropriate contact pressure distribution while designing the machine foundation subjected to rocking mode of vibration. Further it has been brought out that the displacement amplitude at resonance is significantly influenced by inertia ratio, b_ψ values. It is preferable to have as low a value of b_ψ (in other words as larger area as practicable) for safer design.

Also, Layering of soil medium below the foundation is an important factor which influence the responses of the foundation.

2.4 Response of Foundation resting on Layered Soil:

When poor soil is replaced by compacted fill then optimum effective fill depth (i.e. the depth of fill beyond which the increase in resonant frequencies become insignificant) determination become the most important parameter from its design consideration. (Ref.4)

The above study has been carried out by Biswas et. al.(1995)., by taking the model of a rectangular steel tank of dimension 122cm x 85 cm x 115 cm with 8 mm thick rubber. From analysis, it has been concluded that the effective mass of the soil participating in vibration can be expressed as $\rho L/3$ and if the width and equivalent soil surcharge of the foundation remains constant, then the effective depth of vibrating soil mass is maximum for footing having $(A/B) = 1.0$ and minimum for strip footing.

2.5 Effect of Shape on Torsional Vibration of Block Foundation:

The effect of shape of the footing on its behaviour under torsional vibration has been studied by Bharade, et. al., The model footing blocks were made of well seasoned teak wood and of various shapes. For application of torque in horizontal plane, a special arrangement was desired such that the horizontally applied force developed an eccentricity (of 18 cm) with respect to the center of gravity of footing.

It was observed that the coefficient of elastic nonuniform shear (C_θ), resonant frequency (f_{nr}) and damping factor (D) of a vibrating footing soil system are influenced by footing shape, being least for circular shape and maximum for rectangular shape. Resonant amplitude ($A_{\theta r}$) is also dependent upon footing shape, being maximum for circular shape and minimum for rectangular shape.

Torsional vibrations are induced in structural foundations where there exists eccentric forces in horizontal plane. The oscillation of a block footing subjected to a torsional moment (T), can be expressed by the following expression (Ref.2)

$$I_o\ddot{\theta} + C\dot{\theta} + C_\theta J_o\theta = T \qquad \ldots\ldots\ldots\ldots\ldots\ldots \qquad (2.10)$$

in which θ represents angular displacement of footing, $\dot{\theta}$ and $\ddot{\theta}$, its velocity and acceleration respectively, I_o represents its mass moment of inertia about vertical axis (i.e. axis of rotation), J_o represents second moment of areas of its base about vertical axis, C_θ represents coefficients of elastic non-uniform shear and c represents damping coefficients.(Ref.3)

2.6 Design Criterion:

Foundation supporting the machines may vibrate at any of the modes (e.g. vertical, horizontal, rocking and torsional) depending on the type of the machines. For the design of foundation – soil systems subjected to any mode of vibration, the primary, criterion is limiting the displacement amplitude at the operating frequency. Another equally important criterion is the resonance condition. To avoid, resonance condition, the operating frequency should be kept away from the natural frequency of the foundation soil system keeping in mind these two criteria, the design can be carried out either by using elastic half space theory or mass-spring dashpot model or by empirical method out of these, elastic half space theory is most popular and rational.[Ref.13(ii)]

As per Sridharan et al. (1992) for vertical and horizontal modes of vibration, the amplitude for frequency ratios other than 1.0 is very near to resonance amplitude for lower values of mass ratio for constant force system. With increase in mass ratio, the amplitude decreases significantly. But the behaviour is almost same for both sides of resonance. Hence, one can design as overtuned or undertuned keeping the mass ratio as large as practicable. For rotating mass system, the behaviour is not same in both sides of resonance. Amplitude decreases very rapidly with increase in mass ratio when ω/ω_m is less than one. Hence, it is desirable to design as overtuned keeping mass ratio as large as for practicable for a rotating mass system.

For rocking and torsional modes, the amplitude at frequency ratios other than 1.0 is very much less with increase in inertia ratio for a constant force system and the behaviours are almost same on both sides of resonance. Hence, one can design for a constant force system as overtuned or undertuned keeping the inertia ratio as large as practicable. Amplitude is decreased very rapidly with decrease in the frequency when

$\omega / \omega_m < 1.0$ for a rotating mass system. Hence, it is desirable to design as overtuned case keeping inertia ratio as large as practicable.

From all the previous studies, it is sound that a number of factors influence the performance of a machine foundation in different modes of vibration. Many of these studies are done by experiments or by analytical method which have some limitations and approximations. So the finite element method is supposed to be the best one for studying the dynamic responses of a machine foundation.

3. ANSYS Software

3.1 General:

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex non-linear, transient dynamic analysis. The ANSYS, linc. offers a variety of derived a products with features tailored for specific finite element analysis disciplines. Features of the multi-purpose ANSYS program include finite element analysis capabilities for all engineering disciplines-structural, mechanical, electrical, electromagnetic, electronic, thermal fluid and biomedical The basic ANSYS program is available in two versions with different names

(i) ANSYS/ Multi physics is the "full" ANSYS program plus the electromagnetic and FLOTRAN add-on options.

(ii) ANSYS /Mechanical has the linear stress, structural, dynamic analysis, buckling substructuring, heat transfer, thermal acoustics and piezoelectric capabilities of ANSYS /Multiphysics but excludes the three add on options: electromagnetic, LS-DYNA explicit dynamics and FLOTRAN computational fluid dynamic (CFD). Before going into the details of ANSYS a brief introduction to the finite element method and its application to the dynamics is presented in the following section.

3.2 Finite Element Method:

3.2.1. Introduction :

With the invention of high speed electronic digital computers, finite element method also was developed by the engineers for solving various complicated problems.

Initially, this method was developed for structural analysis but the basic concept of using this method made it versatile in use for various other fields of engineering.

Sometimes, it becomes impossible to obtain analytical mathematical solutions for many engineering problems. Actually, an analytical solution is a mathematical expression that gives the values of the desired unknown quantity at any location in a body, and consequently, it is valid for an infinite number of points in the body. Sometimes, numerical methods yield approximate results at some discrete points in the body. The process of selecting only a certain number of discrete points in the body can be termed as descretization. One of the ways to descretize a body or a structure is to divide it into an equivalent system of smaller bodies, or units. So, the solution for whole body can be obtained by assembling the solution for discrete points. So, this approach is known as going from part to whole (Ref. 6).

Thus, it can be said that the FEM is the representation of a body or a structure by an assembly of subdivision called finite elements. These elements are considered interconnected at joints which are called nodes or nodal points. Simple functions are chosen to make approximation of the distribution or variation of the actual displacements over such finite element. Such assumed functions are called displacement functions or displacement models. The unknown magnitudes or amplitudes of the displacement function are the displacements (or the derivatives of the displacements) at the nodal points. Hence, the final solution will yield the approximate displacements at discrete locations in the body, the nodal points. A displacement model can be expressed in various simple forms, such as polynomials and trigonometric function. Since polynomials offer ease in mathematical manipulation, they have been employed commonly in finite element applications.

3.2.2 Steps in FEM Analysis:

The sequence of steps explained below, present the actual solution process that is followed in setting up and solving the equilibrium problem (Ref. 6).

(i) Descritization of the continuum: The continuum means the whole physical body, structure, or solid to be analysed. By descretization, we means that the whole body is divided into small equivalent parts called finite elements, which has been explained earlier. The finite elements may be triangles, group of triangles or quadrilaterals for a two-dimensional continuum. For three dimensional body, the finite elements may be tetrahedra, rectangular prism, or hexahedra, the generation of mesh of elements should be properly judged and it should represent effectively the given continuum for the particular problem considered. Some of the criteria that influence the mesh generation are as below:

(a) The locations for nodes or subdivision lines and planes are places where abrupt changes in geometry, loading and material properties occur. A node must occur at the point of application of concentrated load because all loads are converted into equivalent nodal point loads. Even for distributed loads we compute equivalent nodal loads. To facilitate this process, it is logical to select as nodal points any location of which there is an abrupt change of distributed loads. Similar consideration apply for discontinuities in geometry, such as the discontinuity in plate thickness. Problems involving non-homogeneous materials also have natural locations for nodal lines, such as the interface between materials of different properties, which represents a layered soil medium. In a stress analysis, the effect of the presence of a crack can be determined by choosing the mesh in such a way that each side of the crack is considered an external boundary, and therefore nodes should be provided along both sides of the crack.

(b) The mesh should be refined where the stress concentration is expected to be more.

(c) If straight-sided elements are used, curved boundaries are approximated as piecewise linear by the sides of the elements adjacent to the boundary. Then, it should be ascertained that the curved boundary is approximated as closely as possible, probably by refinement of the mesh in the vicinity of the most pronounced boundary curvature.

(d) For facilitating manipulation in a computer program or code, we must have a systematic method of labelling elements and nodes. Thus designation of a numbering system is an essential part of descretization.

(e) Band width should be minimized for minimizing the solution time and the storage requirements for the overall stiffness matrix. It can be achieved by changing the stiffness matrix of the individual elements and the system of notation for the nodes. Minimization of band width can be achieved by avoiding the use of many secondary, external nodes in case of higher order models. The can be done by choosing derivatives of the displacements as additional degree of freedom at the primary external nodes. Second, we can perform a systematic subdivision and adopt an appropriate numbering system for the nodes. If the node numbers are used as the basis for numbering the nodal displacements, the bandwidth of the overall stiffness matrix depends upon the largest difference between any two external node numbers for a single element. Let D_m be the maximum largest difference occurring for all elements of the assemblage. The semiband width, B , is then given by

$$B = (D_m + 1)\, f \qquad \text{.......(3.1)}$$

Where f is the number of degrees of freedom at each node. Hence, to minimise the bandwidth, the nodal numbering should be selected to minimize D_m.

(f) Aspect ratio plays an important role in finding out the good results. For two dimensional elements, aspect ratio is defined as the ratio of largest dimension of the element to the smallest dimension. The optimum aspect ratio at any location within the grid depends largely upon the difference in rate of change of displacements in different directions. If the displacement vary at about the same rate in each direction, the closer the aspect ratio to unity, the better the quality of the solution.

(ii) Selection of the displacement models : The simplest displacement model that is commonly employed is a linear polynomial. There are three interrelated factors which influence the selection of a displacement model.

- The type and the degree of the displacement model must be chosen, at first.
- The particular displacement magnitudes that describe the model must be selected. There are usually the displacement of the nodal points, but they may also include derivatives of the displacements at some or all of the nodes.
- Third, certain requirements should be satisfied which ensure that the numerical results approach the correct solution.

(iii) Derivation of the element stiffness using a variational principle: The stiffness matrix contains the coefficients of the equilibrium equations desired from the material and geometric properties of an element and obtained by applying the principle of minimum potential energy, the stiffness relates the displacement and the force at the nodal points which is given by the expression:

$$[K]\{q\} = \{Q\} \qquad \text{..........} \qquad (3.2)$$

where,

[K] = stiffness matrix

{q} = displacement vector

{Q} = nodal force vector

(iv) Assembly of the algebraic equation for the overall discretized continuum: By this process, the global stiffness matrix for whole body is formed from element stiffness matrix. The overall equilibrium relation is given by

$$[K]\{r\} = \{R\} \qquad \text{.........} \qquad (3.3)$$

where,

[K] = stiffness matrix

{r} = nodal displacement vector

{R} = load vector

(v) The algebraic equations assembled as above are solved for unknown displacement

(vi) The other quantities like stress, stains etc. are derived from the basic quantity i.e. displacement.

3.2.3 FEM Used in Dynamics:

There are four different types of problems used in dynamics free vibration, steady- state response, transient response to known excitation, and response to random excitation. We have mass matrix along with stiffness matrix while dealing with dynamics problems. There are two basic types of mass matrix, the consistent mass matrix and the lumped mass matrix. The former may be derived by Hamilton's principle, which is the variational theorem associated with the displacement finite element method of dynamic analysis.

(i) Consistent Mass Matrix – For all problems in dynamics which are time dependent, we have equation of motion for the element in the form of :

$$[m]\{\ddot{q}\} + [k]\{q\} = \{Q\} \qquad \text{..........} (3.4)$$

The [m] is the consistent mass matrix given by

$$[m] = \iiint_v \rho [N]^T [N] dv \qquad \text{...........}(3.5)$$

(ii) Lumped Mass Matrix : In the direct method of formulation, the mass of length, area, or volume tributary to a particular node is considered to be concentrated at the node. The resulting lumped mass matrix which is a diagonal matrix is given by :

$$[m] = \iiint_v \rho [\psi]^T [\psi] dv \qquad \text{.....................}(3.6)$$

where $[\psi]$ is the matrix of function ψ_i which have unit value over the region tributary to node i and also value elsewhere.

(iii) Consistent Masses versus Lumped Masses :

Lumped masses provide economies compared to consistent masses. The diagonal lumped mass matrix for the assemblage requires less storage space than the banded consistent mass matrix. Diagonal lumped form greatly facilitates matrix calculations. Moreover, for few problems on beam, plate and shell analyses, lumped masses permit a marked reduction of the number of equations occurring in the dynamic problems.

3.3 Basic Analysis Procedures:

3.3.1 Steps of Analysis :

A typical ANSYS analysis has three distinct steps: (Ref. 20)

(i) The model is built at first

(ii) The load should be applied and the solution is obtained.

(iii) The results are reviewed.

(i) Building a Model:

Building a finite element model requires more of an ANSYS user's time than any part of the analysis. At first, a job name and analysis title is specified. Than PREP7 preprocessor in used to define the element types, element real constants, material properties and the model geometry.

(a) Defining the job name :

The job name is a name that identifies the ANSYS job. When a jobname for an analysis is defined the jobname becomes the first part of the name of all files the analysis creates.

(b) Defining an Analysis Title:

GUI (Graphical user interface) path, **utility Menu >File> change title,** or the **/TITLE** common, defines a title for the analysis ANSYS includes the title on all graphics displays and on the solution output.

(c) Defining Units :

The ANSYS program does not assume a system of units for analysis. Units must be consistent for all input data.

(d) Defining Element Types:

The ANSYS element library contains more than 100 different types, Each element type has a unique number and a prefix that identifies the element category: **BEAM4, PLANE 77, SOLID 96,** etc. The element categories available are shown in TABLE 1.

(e) Defining Element Real Constants:

Element real constants are properties that depend on the element type, such as cross sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA) moment of inertia (IZZ), height (HEIGHT) shear deflection constant (SHEARZ), initial strain (ISTRN) and added mass per unit length (ADDMAS). Not all element types require real constants and different elements of the same type may have different real constant values.

(f) Defining an Material Properties :

Most element types require material properties. Depending on the application, material properties may be:

* Linear or nonlinear

* Isotropic, orthotropic or anisotropic

- Constant temperature or temperature - dependent.

As with element types and real constants, each set of material properties has a material reference number. The table of material reference number versus material property sets is called the material table.

(g) Creating the Model Geometry:

Once the material properties are defined, the next step in an analysis is to generate a finite element model nodes and elements - that adequately describes the model geometry. There are two methods to create the finite element model.

- Solid Modelling
- Direct Generation

With solid modelling, the geometric shape of the model is described then ANSYS is instructed to automatically mesh the geometry with the nodes and elements. With direct generation, the location of each node and connectivity of each element are manually defined.

(ii) Applying Loads and obtaining the Solution:

In this step, the SOLUTION processor is used to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite elements solution. The loads can be applied by using the PREP7 preprocessor also.

(a) Defining the Analysis type and Analysis options :

The analysis type based on the loading conditions and the response wished to obtain is chosen. For example, if natural frequencies and mode shapes are to be calculated, then modal analysis is chosen. The following type analysis can be selected in ANSYS program.

- Static (or steady - state)
- Transient
- Harmonic
- Modal
- Spectrum
- Buckling
- Substructuring

Not all analysis types are valid for all disciplines. Modal analysis for example is not valid for a thermal model.

(b) Applying Loads:

The word loads are used in this program includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads in the ANSYS program are divided into six categories.

- DOF Constraints
- Forces
- Surface loads
- Body loads
- Inertia loads
- Coupled field loads

These loads can be applied either on the solid model (keypoints, lines and areas) or the finite element model (nodes and elements).

Two important load-related terms are load step and substep. A load step is simply a configuration of loads for which user obtain a solution. In a structural analysis, for example, wind loads can be applied in one step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

Substeps are incremental steps taken within a load step. It is used mainly for accuracy and convergence purposes in transient and nonlinear analysis. Substeps are also known as time steps - steps taken over a period of time.

(c) Specifying Load Step Option:

Load step options are options that can be changed from load step to load step, such as number of substeps, time at the end of a load step, and output controls. Depending upon the type of analysis, load step options may or may not be required.

(d) Initiating the Solution:

Once the SOLVE command is issued or equivalent GUI path is chosen, the ANSYS program takes model and loading information from the data base and calculates the results. Results are written to the results file and also to the database. The only difference is that only one set of result can reside in the database at one time, while all sets of results (for all substeps) can be written to the results files.

(iii) Reviewing the Results:

Once the solution has been calculated, the ANSYS postprocessors can be used to review the results. Two postprocessors are available : POST 1 and POST 26. In POST 1 post processor, contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis. POST 1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and path operations.

In POST 26, the time history postprocessor, is used to review results at specific points in the model over all time steps. The graph plots of result data versus time (or frequency) and tabular listings can be obtained by POST 26. Other POST 26 capabilities include arithmetic calculations and complex algebra.

Substeps are points within a load step at which solution are calculated. It can be used for different reasons:

- In a nonlinear static or steady state analysis, Substeps are used to apply the loads gradually so that an accurate solution can be obtained.
- In a linear or nonlinear transient analysis, substeps are used to satisfy transient time integration rules (which usually dictate a minimum time step for an accurate solution).
- In a harmonic response analysis, substeps are used to obtain solutions at several frequencies within the harmonic frequency range.

Equilibrium iterations are additional solutions calculated at a given substep for convergence purpose. They are iterative corrections used only in nonlinear analysis (static or transient), where convergence plays in important role.

So, a load step is a set of loads applied over a given time span. Substeps are time points within a load step at which intermediate solutions are calculated. The difference in time between two successive substeps can be called a time step or time increment. Equilibrium iterations are iterative solutions calculated at a given time point purely for convergence purpose.

3.3.2 Loading:

The main aim of a finite element analysis is to examine how a structure or component responds to certain loading conditions. Specifying the proper loading condition is, therefore, a key step is the analysis. One can apply loads on the model in a variety of ways in the ANSYS program. Also, with the help of load step options, one can control how the loads are actually used during solution. Examples of loads in different disciplines are:

Structural : displacement, forces, pressures, temperature (for thermal strain), gravity.

Thermal : Temperatures heat flow rates, convection, internal heat generation, infinite surface.

Magnetic : magnetic potentials, magnetic flux, magnetic current segments, source current density, infinite surface

Electric : Electric potentials (voltage) electric current electric charges, Charge densities, infinite surface.

Fluid : Velocities, pressures

Loads are divided into six categories DOF constraints, forces (concentrated loads), surface loads, body loads, inertia loads and coupled – field loads.

3.3.2.1 Load steps, Substeps and Equilibrium Iterations:

A load step is simply a configuration of loads for which a solution in obtained. In a linear static or steady – state analysis, one can use different load steps to apply different set of loads – winds load in the first load step, gravity load in the second load step, both loads and a different support condition in the third load step, and so on.

Sub steps are points within a load step at which solutions are calculated. It can be used for different reasons:

- In a nonlinear static or steady state analysis, substeps are used to apply the loads gradually so that an accurate solution can be obtained.
- In a linear or nonlinear transient analysis, substeps are used to satisfy transient time integration rules (which usually dictate a minimum time step for an accurate solution).
- In a harmonic response analysis, substeps are used to obtain solutions at several frequencies within the harmonic frequency range.

Equilibrium iterations are additional solution calculated at a given substep for convergence purposes. They are iterative corrections used only in nonlinear analyses (static or transient), where convergence plays an important role.

So, a load step is a set of loads applied over a given time span. Substeps are time points within a load step at which intermediate solutions are calculated. The difference in time between two successive substeps can be called a time step or time increment. Equilibrium iterations are iterative solutions calculated at a given time point purely for convergence purposes.

3.3.2.2 Stepped versus Ramped loads.

When one specify more than one substep in a load step, the question of whether the loads should be stepped or ramped arises:

- If a load is stepped, then its full value is applied at the first substep and stays constant for the rest of the load step.

- If a load is ramped, then its value increases gradually at each substep, with the full value occurring at the end of the load step.

Load step options is a collective name given to options that control load application, such as time, number of substeps, the time step and stepping and damping of loads. Other types of load step options include convergence tolerances (used in nonlinear analysis), damping specifications in a structural analysis, and output controls.

3.3.2.3 Load Applications:

Loads can be applied either on the solid model (on key points, lines and areas) or on the finite element model (on nodes and elements). For example, one can specify forces at a keypoint or a node. Similarly one can specify connections (and other surface loads) on lines and areas or on nodes and element faces. No matter how one specify the loads, the solver expects all loads to be in terms of the finite element model. Therefore, if one specify load on the solid model, the program automatically transfers them to the nodes and elements at the beginning of the solution. TABLE 2 and TABLE 3 shows the DOF constraints available in each discipline and commands for DOF constraints respectively.

3.3.3 Solution:

In the solution phase of the analysis, the computer takes over and solves the simultaneous equations that the finite element method generates. The results of the solution are:

(a) Nodal degree of freedom values, which from the primary solution, and (b) derived values; which from the element solution. The element solution is usually calculated at the elements integration points. The ANSYS program writes the results to the database as well as to the result files.

Several methods of solving the simultaneous equations are available in the ANSYS program:

(i) Frontal Solver.

(ii) Jacobi Conjugate Gradient (JCG) solution.

(iii) Incomplete Cholesky Conjugate Gradient (ICCG) solution.

(iv) Preconditioned Conjugate Gradient (PCG) solution.

(v) An automatic iterative solver option (ITER).

(i) The Frontal Solver:

The frontal solver does not assemble the complete global matrix. Instead, ANSYS performs the assembly and solution steps simultaneously as the solver processes each element. The method works as follows:

- After the individual element matrices are calculated, the solver read in the degrees of freedom for the first element.
- The program eliminates any degree of freedom that can be expressed in terms of the other DOF by writing an equation to the .TRI file.The process repeats for all elements until all degrees of freedom have been eliminated and a complete traingularised matrix is left on the .TRI file.
- The program then calculates the model DOF solution by back substitution, and uses the individual element matrices to calculate the element solution. FIGURE 1 shows the main steps in a frontal solution and the files produced at each step.

A term frequently mentioned in the context of a frontal solver is 'wavefront'. The wavefront is the number of degrees of freedom retained by the solver during the traingularization on because they cannot yet be eliminated. As the solver processes each element and its degrees of freedom, the wavefront swells and shrinks, and finally becomes zero when all degrees of freedom have been processed. The highest value of the wavefront is called the maximum wavefront, and the averaged, root mean square value is called the RMS wavefront.

(ii) Jacobi Conjugate Gradient (JCG) Solver:

The JCG solver also starts with element matrix formulation, but the remaining steps are different. Instead of, traingularizing the global matrix, the JCG solver assembles the full global matrix. The solver then calculates the DOF solution by iterating to convergence (starting with an assumed zero value for all DOF). FIGURE 2 summarizes these steps. The JCG solver is best suited for 3D scalar field analysis that involves large, sparse matrices, for example a 3D magnetic field analysis.

(iii) Incomplete Cholesky Conjugate Gradient (ICCG) Solver:

The ICCG solver operates similarly to the JCG solver with the following exception:

- The ICCG solver is more robust then the JCG solver for matrices that are not well conditioned. Performance will vary with matrix conditioning, but in general ICCG performance compares to that of the JCG solver.
- The ICCG solver uses a more sophisticated preconditioner then the JCG solver. Therefore, the ICCG solver requires approximately twice as much memory as the JCG solver.

The ICCG solver is available only for static analysis, full harmonic analysis, or full transient analysis. The ICCG solver is useful for models that have sparse matrices, and is valid for elements with symmetric and unsymmetric matrices. The solver runs faster then the frontal solver for most problem sizes.

(iv) Preconditioned Conjugate Gradient (PCG) solver:

The PCG solver operates similarly to the JCG solver, with following exceptions:

1. The PCG solver is usually about 4 to 10 times faster then the JCG solver for structural solid elements and about 10 times faster for shell elements. Savings increase with the problem size.
2. The JCG solver uses the diagonal of the global assembled matrix as a preconditioner.
3. It requires approximately twice as much memory as the JCG solver.

(v) Automatic Iterative Solver Option:

The automatic iterative solver option chooses on appropriate iterative solver (JCG, PCG etc.) based on the physics of the problem. When one use the automatic iterative solver option, an accuracy level must choose to be used for convergence checking.

- A linear static structural analysis with an accuracy level of 1 would use the PCG solver with a convergence tolerance of $1*10^{-4}$.
- A transient thermal analysis with a selected accuracy level of 1 would use the JCG solver with a convergence tolerance of $1*10^{-6}$. This solver option is available only

for linear static/full transient structural analysis and steady state transient thermal analysis.

3.3.4 An Overview of Post processing:

After building the model and obtaining the solution, the results can be viewed by postprocessing. So postprocessing means reviewing the results of an analysis. It is probably the most important steps in the analysis, because by using this one can understand how the applied loads affect the design, how good finite element mesh is, and so on.

Two postprocessors are available to review the results: POST1, the general postprocessor, and POST 26, the time- history postprocessor. POST1 allows to review the results over the entire model at specific load steps and substeps (or at specific time-points or frequencies).

POST26 allows to review the variation of a particular result item at specific points in the model with respect to time, frequency or some other result item. In a transient magnetic analysis, for instance, one can graph the eddy current in a particular element versus time, or, in a nonlinear structural analysis, one can graph the force at a particular node versus its deflection.

The ANSYS solver unites results of an analysis to the results file during solution. The name of the result file depend on the analysis discipline.

Jobname. RST for a structural analysis

Jobname. RTH for a thermal analysis

jobname. RMG for a magnetic field analysis

Jobname. RFL for a FLOTRAN analysis

Types of Data Available for Postprocessing:

The solution phase calculates two types of results data: (TABLE 4)

- Primary data consists of the degree of freedom solution calculated at each node; displacements in a structural analysis, temperature in thermal analysis, magnetic potentials in a magnetic analysis, and so on. These are also known as nodal solution data.
- Derived data are those results calculated from the primary data, such as stresses and strain in a structural analysis, thermal gradients and fluxes in a thermal analysis, magnetic fluxes in a magnetic analysis, and the like.

3.4 Modelling And Meshing:

3.4.1 Overview of Model Generation:

The ultimate purpose of a finite element analysis is to re-create mathematically the behaviour of an actual engineering problem. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions and other features that are used to represent the physical system.

In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and elements that represent the spatial volume and connectivity of the actual system. (Ref. 21)

3.4.1.1 Typical steps involved in Model generation:

A common modelling session might follow this general outline. Planning the approach should be done by determining the objectives, basic form of model appropriate element types and mesh density. All these considerations are finalised before initiating ANSYS session.

- To initiate model building session, the preprocessor should be entered.
- A working plane should be established.
- Basic geometric features should be generated using geometric primitives and Boolean operators.
- An appropriate coordinate system should be activated.
- Other solid model features should be generated from the bottom up. That is, keypoints should be created and their lines, areas and volumes should be defined.
- More Boolean operators or number controls should be used to join separate solid model regions together as appropriate.
- Tables of element attributes (element types, real constants, material properties and element coordinate system) should be created.
- Element attribute pointers should be set.
- Meshing control should be set to establish desired mesh density. This step is not always required because default element sizes exist when one enter into program. If one wishes to refine the mesh automatically, one should exit the preprocessor at this point and activate adaptive meshing.
- Nodes and elements should be created by meshing the solid model
- After generating the nodes and elements, features such as surface-to-surface contact elements, coupled degree of freedom, and constraint equations should be added.
- Model data should be saved to Jobname.DB.
- Lastly, after completing the task one may exit the preprocessor.

3.4.2 Planning Approach for Solving Problems:

The first step of one's analysis relies not on the capabilities of the ANSYS program but relies instead on own education experience and professional judgement. One user can determine what the objectives of the analysis must be. The objectives one establish at the start will influence the remainder of one's choices as the model is generated.

3.4.2.1 Choosing a Model type (2-D,3-D, etc.)

Finite element model may be categorized as being 2 - dimensional or 3-dimensional, and as being composed of point elements, line elements, area elements or solid elements. Of course one can interring different kinds of elements as required (taking care to maintain the appropriate compatibility among degrees of freedom).

For example one might model a stiffened shell structure using 3-D shell elements to represent the skin and 3-D beam elements to represent the ribs one's choice of model generation will be most practical for the problem.

LINE models can represent 2-D or 3-D beam or pipe structures as well as 2-D models of 3-D axisymmetric shell structure solid modelling usually doesn't offer much benefit for generating line models; They are most often created by direct generation methods.

2-D solid analysis models are new for thin planer structures (Plane stress), "infinitely long" structures having a constant cross section (plane strain), or axisynnetric solid structures. Although many 2-D analysis models are relatively easier to create by direct generation on methods, they are usually easier to create with solid modelling.

3-D SHELL MODELS are used for thin structures in 3-D space. Although some 3-D shell analysis models are relatively easy to create by direct generation methods, they are usually easier to create with solid modelling.

3-D SOLID analysis models are used for thick structures in 3-D space that have neither a constant cross-section nor an axis of symmetry.

3.4.2.2 Choice between Linear and Higher order Elements:

The ANSYS programs element library includes two basic types of area and volume elements linear (with or without extra shapes) and quadratic.

(i) Linear Element (No midside Nodes)

For structural analyses these corner model element with extra shape function will often yield an accurate solution in a reasonable amount of computer time. When using these elements, it is important to avoid their degenerate forms in critical regions. One should avoid using excessively distorted linear elements. In nonlinear structural analyses, one will usually obtain better accuracy at less expense if one use a fine mesh of these linear elements rather than a comparable coarse mesh of quadratic elements.

(ii) Quadratic Element (Mid side Nodes):

For linear structural analyses with degenerate element shapes (that is, triangular 2-D elements and wedge or tetrahedral 3D-elements), the quadratic elements will usually yield better result at less expense than will the linear elements.

3.4.3 Meshing the model:

The procedure of generating a mesh of nodes and elements consists of three main steps.

(i) The element attributes should be set.

(ii) Meshing controls (optional) should be set.

(iii) The mesh should be generated.

The second step, setting mesh controls is not always necessary because the default mesh controls are appropriate for many models. If no controls are specified the program will use the default setting to produce a free mesh. As an alternative, one can use the Smart Size feature to produce a better quality free mesh.

3.4.3.1 Free or Mapped mesh:

Before meshing the model and even before building the model it is important to think about whether a free or mapped mesh is appropriate for the analysis A free mesh is one that has no restriction in term of element shapes and has no specified pattern applied to it.

Compared to a free mesh, a mapped mesh is terms of the element shape it contains and the pattern of the mesh A mapped mesh containing only quadrilaterals (area) or only hexahedron (volume) elements. In addition a mapped mesh typically has a regular patterns with obvious rows of elements. If this type of mesh is desired the user must build the geometry as a series of fairly regular volumes and / or areas that can accept a mapped meshed.

3.4.3.2 Setting element Attributes:

Before a mesh of nodes and elements are generated one must first define the appropriate element attributes that is, one must specify the following:

- Element type (i.e., BEAM 3, PLANE 82 etc.).
- Real constant set (usually comprising the element's geometric properties such as thickness or cross-sectional area).
- Material properties set (such or Young's modulus, thermal conductivity, etc.).
- Elements coordinate system.

3.4.3.3 Smart Elements Sizing for free meshing:

Smart element sizing (Smart Sizing) is a meshing feature that creates initial elements sizes for free meshing operation. Smart Sizing gives the mesher a better chance of creating reasonably shaped elements during automatic mesh generation.

The smart sizing algorithm first computes estimated element edge lengths for all lines in the areas or volumes being meshed. The edge length on these lines are then refined for curvature and other small features in the geometry. Since all lines and areas are sized before meshing begins, the quality of the generated mesh is not dependent on the order in which the areas or volumes are meshed.

If quadrilateral elements are being used for area meshing. Smart Sizing tries to set an even number or line divisions around each area so that an all quadrilateral mesh is possible. Triangles will be included in the mesh only if forcing all quadrilaterals could create poorly shaped elements.

There are two categories of smart sizing controls: basic and advanced. To use the basic controls, a mesh size level from 1 (fine mesh) to 10 (coarse mesh) should be specified. The program automatically sets a series of individual control values that are used to produce the requested size level.

One may prefer to use the advanced method, which involves setting the individual control quantities manually. This allows to 'tweak' the mesh to better fit the needs.

3.4.3.4 Default Element sizes for mapped meshing:

The DESIZE command allows to modify such defaults as the minimum and maximum number of elements that will be attached to an unmeshed line maximum spanned angle per element and minimum and maximum edge lengths. This command is always used to control element sizing for mapped meshing DESIZE setting are also used by default for free meshing. However it is recommended that smart sizing should be used instead for free meshing operations.

3.4.4 Revising the Model:

There are generally two situations in which a user may want to refine a mesh in a local region (i) one has meshed a model and would tike a finer mesh in specific regions of the model or (ii) user has completed the analysis and based on the results, would like a more detailed solution in a region of interest. For area meshes, the ANSYS program allows to refine the mesh locally around specified nodes, elements, key points or lines.

More detailed solution in a region of interest. For area meshes, the ANSYS program allows to refine the mesh locally around specified nodes, elements, keypoints or lines.

3.4.5 Direct Generation:

Direct generation is the approach in which one defines the nodes and elements of a model directly. Despite the many convenience commands that allow user to copy, reflect, scale etc. a given pattern of nodes or elements, direct generation can commonly require about ten times as many data entries to define a model as compared to solid modelling.

A model that is assembled by direct generation is defined strictly in terms of nodes and elements. Even though node and element generation operations can be interspersed, no one element can be defined until after all its nodes have been created.

3.5 Structural Analysis by ANSYS:

3.5.1 Overview

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings; but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies and machine housings as well as mechanical components such as pistons, machine parts, and tools.

The seven types of structural analysis available in the ANSYS family of products are as below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements, other quantities, such as strains, stresses and reaction forces are then derived from the nodal displacements. (Ref. 22)

One can perform the following types of structural analysis:

(i) **Static analysis:** It is used to determine displacements, stresses etc. under static loading conditions. Both linear and nonlinear static analysis. Nonliearities can include plasticity, stress stiffening, large deflection, large strain, hyperelasticity, contact surfaces and creep.

(ii) **Modal analysis:** It is used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

(iii) **Harmonic analysis:** It is used to determine the response of a structure to harmonically time-varying loads.

(iv) **Transient Dynamic analysis:** It is used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned above under static analysis are allowed.

(v) **Spectrum analysis:** It is an extension of the nodal analysis and used to calculate stresses and strains due to a response spectrum or a PSD input (random variations).

(vi) **Buckling analysis:** It is used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling and nonlinear buckling analyses are possible.

(vii) **Explicit Dynamic analysis:** ANSYS 5.3 provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

In addition to the above analysis types, several special purpose features are available:

- Fracture mechanics
- Composites
- Fatigue
- P-method

(ii) Block Lanczos Method:

The Block Lanczos eigenvalue solver uses the Lanczos algorithm where the Lanczos recursion is performed with a block of vectors. This method is as accurate as the subspace method, but faster.

The Block Lanczos method is especially powerful when searching for eigen frequencies in a given part of the eigenvalue spectrum of a given system. The convergence rate of the eigenfrequencies will be about the same when extracting modes in the midrange and higher end of the spectrum as when extracting the lowest modes.

(iii) Power Dynamics Method:

The power Dynamics method internally uses the subspace iterations, but uses the PCG iterative solver. This method is significantly faster than either the subspace or the Block Lanczos methods, but may not converge if the model contains poorly shaped elements. This method is especially useful in very large models (1,00,000 + DOFs) to obtains a solution for the first few modes.

The power Dynamics method does not perform a strum sequence check (that is, does not check for missing modes), which should effect only problems with multiple repeated frequencies. This method always uses lumped mass approximations.

(iv) Reduced Method:

The reduced method uses the HBI algorithm (House holder – Bisection – inverse iteration) to calculate the eigenvectors. It is relatively fast because it works with a small subset of degrees of freedom called master DOF. Using master DOF leads to an exact [K] matrix but an approximate [M} matrix (usually with some loss in mass). The accuracy of the results, therefore, depends on how well [M] is approximated, which is turn depends on the number and location of masters DOFs.

(v) Unsymmetrical Method:

The unsymmetric methed, which also uses the full [K] and [M] matrix, is meant for problems where the stiffness and mass matrix are unsymmetric (for example, acoustic fluid –structure interaction problem). It uses the Lanczos algorithm which calculates complex eigenvalues and eigenvectors if the system is non-conservative (for example, a shaft mounted) on bearings). The real part of the eigenvalue represents the natural frequency and the imaginary part is a measure of the stability of the system –a negative value means the system is stable, whereas a positive value means the system is unstable. strum sequence checking in not available for this method. Therefore, missed modes are a possibility at the higher end of the frequencies extracted.

(vi) Damped Method:

The damped method is meant for problem where damping cannot be ignored, such as rotor dynamics applications. It uses full matrices ([K], [M], and the damping matrix [C]). It uses the Lanczos algorithm and calculates complex eigenvalues and eigenvectors. In this case the imaginary part of the eigenvalue represents the natural frequency and the real part is a measure of the stability of the system. Strum sequence checking is not available for this method. Therefore, missed modes are a possibility at the higher end of the frequencies extracted.

3.5.3 Harmonic Response Analysis :

Harmonic response analysis is a technique used to determine the steady state response of a linear structure to loads that vary sinusoidally (harmonically) with time.

The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies.

This analysis technique calculates only the steady state, forced vibrations of a structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis.

Harmonic response analysis is a linear analysis. Any nonlinearties, such as plasticity and contact (gap) elements, will be ignored, even if they are defined. User can, however, have unsymmetric system matrices such as those encountered in a fluid structure interaction problem. Harmonic analysis can also be performed on a prestressed structure, such as a violin string (assuming the harmonic stresses are much smaller than the pre-tension stress). Load commands for harmonic responses analysis and load step options have been shown in TABLE 5 and TABLE 6 respectively.

Three harmonic response analysis methods are available:

(i) **The full method**

(ii) **The reduced method**

(iii) **The mode superposition method**

(i) The full method:

This method is the easiest of the three methods. It uses the full system matrices to calculate the harmonic response (no matrix reduction). The matrices may be symmetric or unsymmetric. The advantages of the full method are :

(ii) **Modal analysis:** It is used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

(iii) **Harmonic analysis:** It is used to determine the response of a structure to harmonically time-varying loads.

(iv) **Transient Dynamic analysis:** It is used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned above under static analysis are allowed.

(v) **Spectrum analysis:** It is an extension of the nodal analysis and used to calculate stresses and strains due to a response spectrum or a PSD input (random variations).

(vi) **Buckling analysis:** It is used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling and nonlinear buckling analyses are possible.

(vii) **Explicit Dynamic analysis:** ANSYS 5.3 provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

In addition to the above analysis types, several special purpose features are available:

- Fracture mechanics
- Composites
- Fatigue
- P-method

3.5.1.1 Elements used in structural analysis:

Most ANSYS element types are structural elements, ranging from simple spars and beams to more complex layered shells and large strain solids.

3.5.1.2 Types of Solutions Methods:

Two solution methods are available for solving structural problems in the ANSYS family of products: the h-method and the p-method. The h-method can be used for any type of analysis, but the p-method can be used only for linear structural static analysis.

Depending on the problem to be solved, the h-method usually requires a finer mesh than the p-method. The p-method provides an excellent way to solve a problem to a desired level of accuracy while using a coarse mesh.

3.5.2 Modal analysis:

Modal analysis is used to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required for spectrum analysis or a mode superposition harmonic or transient analysis.

Modal analysis in the ANSYS family of products in a linear analysis. Any nonlinearities, such as plasticity and contact (gap) elements, will be ignored even if they are defined.

3.5.2.1 Mode Extraction Methods:

The basic equation solved in a typical undamped modal analysis is the classical eigenvalue problem:

$$[K]\ \{\phi\} = \omega_i^2\ [M]\ \{\phi i\} \qquad (3.7)$$

Where [K] = stiffness matrix;

{ϕi}= mode shape vector (eigenvector of mode;)

ω_i = natural circular frequency of mode i (ω_i^2 is the eigenvalue)

[M] = mass matrix

Many numerical methods are available to solve the above equation. ANSYS offers six methods:

(i) Subspace method

(ii) Block Lanczos method

(iii) Power Dynamics

(iv) Reduced (House holder) method

(v) Unsymmetric method

(vi) Damped method

The first four, the subspace, the Block Lanczos, the Power Dynamics and the reduced methods are the most commonly used.

(i) Subspace Method:

The subspace method uses the subspace iteration technique, which internally uses the generalized Jacobi iteration algorithm. It is highly accurate because it uses the full [k] and [M] matrices. For the same reason, however, the subspace method is slower than the reduced method. This method is typically used in cases where high accuracy is required or where selecting master DOF is not practical.

(ii) Block Lanczos Method:

The Block Lanczos eigenvalue solver uses the Lanczos algorithm where the Lanczos recursion is performed with a block of vectors. This method is as accurate as the subspace method, but faster.

The Block Lanczos method is especially powerful when searching for eigen frequencies in a given part of the eigenvalue spectrum of a given system. The convergence rate of the eigenfrequencies will be about the same when extracting modes in the midrange and higher end of the spectrum as when extracting the lowest modes.

(iii) Power Dynamics Method:

The power Dynamics method internally uses the subspace iterations, but uses the PCG iterative solver. This method is significantly faster than either the subspace or the Block Lanczos methods, but may not converge if the model contains poorly shaped elements. This method is especially useful in very large models (1,00,000 + DOFs) to obtains a solution for the first few modes.

The power Dynamics method does not perform a strum sequence check (that is, does not check for missing modes), which should effect only problems with multiple repeated frequencies. This method always uses lumped mass approximations.

(iv) Reduced Method:

The reduced method uses the HBI algorithm (House holder – Bisection – inverse iteration) to calculate the eigenvectors. It is relatively fast because it works with a small subset of degrees of freedom called master DOF. Using master DOF leads to an exact [K] matrix but an approximate [M} matrix (usually with some loss in mass). The accuracy of the results, therefore, depends on how well [M] is approximated, which is turn depends on the number and location of masters DOFs.

(v) Unsymmetrical Method:

The unsymmetric methed, which also uses the full [K] and [M] matrix, is meant for problems where the stiffness and mass matrix are unsymmetric (for example, acoustic fluid –structure interaction problem). It uses the Lanczos algorithm which calculates complex eigenvalues and eigenvectors if the system is non-conservative (for example, a shaft mounted) on bearings). The real part of the eigenvalue represents the natural frequency and the imaginary part is a measure of the stability of the system –a negative value means the system is stable, whereas a positive value means the system is unstable. strum sequence checking in not available for this method. Therefore, missed modes are a possibility at the higher end of the frequencies extracted.

(vi) Damped Method:

The damped method is meant for problem where damping cannot be ignored, such as rotor dynamics applications. It uses full matrices ([K], [M], and the damping matrix [C]). It uses the Lanczos algorithm and calculates complex eigenvalues and eigenvectors. In this case the imaginary part of the eigenvalue represents the natural frequency and the real part is a measure of the stability of the system. Strum sequence checking is not available for this method. Therefore, missed modes are a possibility at the higher end of the frequencies extracted.

3.5.3 Harmonic Response Analysis :

Harmonic response analysis is a technique used to determine the steady state response of a linear structure to loads that vary sinusoidally (harmonically) with time.

The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies.

This analysis technique calculates only the steady state, forced vibrations of a structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis.

Harmonic response analysis is a linear analysis. Any nonlinearties, such as plasticity and contact (gap) elements, will be ignored, even if they are defined. User can, however, have unsymmetric system matrices such as those encountered in a fluid structure interaction problem. Harmonic analysis can also be performed on a prestressed structure, such as a violin string (assuming the harmonic stresses are much smaller than the pre-tension stress). Load commands for harmonic responses analysis and load step options have been shown in TABLE 5 and TABLE 6 respectively.

Three harmonic response analysis methods are available:

(i) **The full method**

(ii) **The reduced method**

(iii) **The mode superposition method**

(i) The full method:

This method is the easiest of the three methods. It uses the full system matrices to calculate the harmonic response (no matrix reduction). The matrices may be symmetric or unsymmetric. The advantages of the full method are :

(a) It is easy to use, because one does not have to worry about choosing master degrees of freedom or mode shapes.

(b) It uses full matrices, so no mass matrix approximation is involved.

(c) It allows unsymmetric matrices, which are typical of such applications as acoustics and bearing problem.

(d) It calculates all displacements and stresses in a single pass.

(e) It accepts all type of loads: nodal forces, imposed (non-zero) displacements, and element loads (pressures and temperatures).

(f) It allows effective use of solid – model loads

A disadvantage of the full method is that no prestressed option is available. Another disadvantages is that this method usually is more expensive than either of the other methods when one use the frontal solver. However, when one uses the ICG solver or the ICCG solver the full method can be very efficient.

(ii) The Reduced Method:

The reduced method enables to condense the problem size by using master degrees of freedom and reduced matrices. After the displacements at the master DOF have been calculated, the solution can be expanded to the original full DOF set. The advantages of this method are:

(a) It is faster and less expensive compared to the full method when using the frontal solver.

(b) Prestressing effect can be included.

The disadvantages of the reduced method are:

(a) The initial solution calculates only the displacements at the master DOF. A second step known as the expansion pass, is required for a complete displacement, stress and force solution. (However, the expansion pass might be optional for some applications).

(b) Element loads (pressures, temperatures etc. cannot be applied)

(c) All loads must be applied at user defined master degrees of freedom (this limits the use of solid model loads)

(iii) The Mode Superposition Method:

This method sums factored mode shapes (eigenvectors) from a modal analysis to calculate the structures response. Its advantages are :

(a) It is faster and less expensive than either the reduced or the full method for many problem.

(b) Element loads applied in the preceding modal analysis can be applied in the harmonic response analysis via the LVSCALE command.

(c) It allows solutions to be clustered about the structures natural frequencies. This results in a smoother, more accurate tracing of the response curve.

(d) Prestressing effects can be included.

(e) It accepts modal damping (damping ratio as a function of frequency)

Disadvantages of the mode super position method are:

(a) Imposed (non-zero) displacements cannot be applied.

(b) When Power Dynamics in used for the modal analysis, initial conditions cannot have previously applied loads.

3.6 Conclusion :

ANSYS program is a versatile computer package which can be used for any type of engineering problems coming in our day-to-day life. The exact simulation of problems for analysis is a difficult work in ANSYS but not impossible. It can be done by knowing the appropriate and adequate data required for analysis. Above all, own engineering judgement for simulation is an important part for using this computer package. Along with linear analysis one can do even nonlinear analysis with ANSYS program. But this package has some limitations which cannot be ignored.

Lastly, it can be concluded that ANSYS program has very wide range of types of analysis, which can be used with full confidence.

4. Bench Mark Problem Solved by ANSYS

4.1 General:

As discussed earlier, that there is significant influence of coupling of motion on the performance of machine foundation. To study this, the ANSYS package has been explored. Before analyzing the real problem of our concern, the capability of ANSYS program for dynamic analysis has been established by solving two bench mark problems. First problem is to find out the natural frequency of a cantilever beam and second one is a solved problem on machine foundation from Richart et.al (1970).

4.2 Analytical Solution of a cantilever beam:

The first three natural frequencies in bending of a cantilevers beam of length L, width B and depth D, having cross-sectional area A(=BxD) are given by the following expressions.(Ref.8)

(a) For bending :-

$$\omega_1 = (1.875)^2 \sqrt{\frac{\mathrm{EI}}{\rho \mathrm{AL}^4}} \text{ rad /sec.} \qquad (4.1)$$

$$\omega_2 = (4.694)^2 \sqrt{\frac{\mathrm{EI}}{\rho \mathrm{AL}^4}} \text{ rad /sec} \qquad (4.2)$$

$$\omega_3 = (7.855)^2 \sqrt{\frac{\mathrm{EI}}{\rho \mathrm{AL}^4}} \text{ rad /sec} \qquad (4.3)$$

(b) For axial compression :

$$\omega_1 = \frac{\pi}{2}\sqrt{\frac{EA}{mL^2}} \text{ rad/sec.} \qquad (4.4)$$

$$\omega_2 = \frac{3\pi}{2}\sqrt{\frac{EA}{mL^2}} \text{ rad/sec.} \tag{4.5}$$

$$\omega_3 = \frac{5\pi}{2}\sqrt{\frac{EA}{mL^2}} \text{ rad/sec.} \tag{4.6}$$

where,

E = Young's modules of elasticity

I = Moment of inertia about z-axis

ρ = mass density

A = Cross-sectional area

$\overline{m} = \rho \times A$ = mass per unit length

L = Length of the cantilever beam

4.2.1 Analysis of the Cantilever Beam by ANSYS :

A concrete cantilever beam of length 1.0 m and depth 0.30 m has been selected for the study through ANSYS program. The width for the cantilever is provided by giving the element thickness equal to 0.10m. The element types chosen are PLANE 42 and PLANE 82, which are solid 4-noded and solid 8-noded respectively. The material properties assigned to concrete have been taken as : Young's modulus of elasticity = 2.1 x 10^9 Kg/m^2 which is obtained from IS:456:1978 by using the expression $E_c = 5700\sqrt{fck}$ N/mm^2. The mass density for reinforced concrete has been taken as per IS:456:1978 equals to 25KN/m^3 the Poisson ratio assigned to concrete for dynamic analysis is taken as 0.24 (Table1). The type of analysis chosen for finding out the natural frequencies of vibration of the cantilever is modal analysis with analysis option as Block Lanczos method.

4.2.2 Comparison of Results of ANSYS with Analytical Method:

The natural frequencies obtained from ANSYS match, almost with those obtained from analytical method. The first mode of vibration has been animated in ANSYS program and found that it is first mode of vibration in bending at frequency 14.37Hz which is nearly equal to I5.31Hz, obtained from the equation (4.1) Similarly, it has been found that the second mode of vibration shows the bending property and has its value of natural frequency of 89.93Hz from PLANE 42, elements and 89.70Hz from PLANE 82 elements. In the same way, the first three modes of bending have been determined and their value of natural frequencies have been compared with those results obtained from the analytical method. The first two modes of axial vibration have been also observed and their values are compared with the values obtained from formulae. It has been observed that all these values of natural frequencies, either in bending or axial vibration obtained from ANSYS program and those obtained from analytical are nearly equal to one another. Comparison between the two methods is reported in Table 2. The various modes of vibration of the cantilever beam (modal analysis) has been shown in Figs.1.1 to 1.5.

4.3 Problem of A Machine Foundation solved by ANSYS:

One example problem has been taken from Richart et.al (1970) on machine foundation. The block size has been taken as 4.88m x 2.1336 m x 0.91m fully embedded in the soil. The underlying soil is of sandy type having Young's modulus of elasticity = 2.61×10^{7} kg/m^{2}, mass density = 245.0 kg –sec/m^{4} and Poisson's ratio = 0.33. For finding out the solution by ANSYS, the model used is of two-dimensional, plane stress with thickness type. The thickness provided to the element is equal to the width of the foundation. The soil portion has been taken as four times the length of the foundation on both sides as well as the depth having same dimension. The details of the block and the surrounding soil idealized is shown in Fig. 2.1.

Initially, the problem geometry has been generated through preprocessing. Different properties mentioned in the Table 3, have been assigned to the corresponding areas. The basic size of 8.0 has been chosen for meshing the model. The end conditions applied are all fixed at the boundary.

A modal analysis has been performed to find out the natural frequency of vibration of this foundation. ANSYS offered 30.5 Hz as the vertical mode of vibration which is near to 31.1 Hz (Fig.2.2) obtained by Richart et.al (1970). A harmonic analysis has been done by applying the load at mid nodal point of the block of magnitude of 5164.2 kg. The frequency range for analysis has been kept as 7.0 Hz to 8.0 Hz in ten steps to capture the amplitude of vibration at the operating frequency of 7.5 Hz of the machine foundation.

A small parametric study has been conducted on the effect of the combination of stiffness matrix multiplier and constant damping ratio. The results have been shown in Table 4. The result matches with the analytical solution when the stiffness matrix multiplier is between 0.01 to 0.05 and at the constant damping ratio of 0.10 because at value of stiffness matrix multiplier as 0.01 and constant damping ratio of 0.1 the amplitude of vibration is obtained as 0.071 m and at stiffness matrix multiplier of value 0.05 and constant damping ratio of 0.1 the amplitude of vibration is 0.043 mm which is near to 0.05 as obtained by analytical method.

Thus it can be concluded that the results obtained by ANSYS are compatible and can be used for various analyses.

5. Parametric Studies on Machine Foundations

5.1 General :

Performance of machine foundation depend on various parameters like properties of soil lying below the foundation (Young's modulus of elasticity, mass density of soil, Poisson's ratio etc.), different layers of soils and their respective depths from the bottom of foundation, types of loading (like harmonic, transient), points of application of loading etc. In this chapter, the effects of different parameters as mentioned above, have been studied using ANSYS program.

5.2 Influence of Soil Properties on the Performance of Machine Foundation:

For the study of the effect of soil properties on the amplitude of vibration of foundation, three types of soil have been considered for analysis. In the first case uniform soil below the foundation has been considered. The model for analysis contains soil of uniform layer (Fig.3). The foundation is fully embedded in the soil. The size of the foundation taken is 4.88 m x 2.1336 m x 0.91 m, a typical size of block as taken for the bench mark problem from Richart et.al. (1970). The soil mass is taken as 4 B to both the sides of the foundation and depth also equal to 4B, where B is the half of the length of the foundation. The material properties of concrete and soils have been shown in Table 5. The input data is shown in Table 6. The element chosen is of plane stress with thickness of 2.1336 m. The mesh has been generated by choosing free mesh with basic Smart Size of value 8. Then the elements in concrete block have been refined by an amount of half. Boundary conditions have been applied as all fixed at the nodal points at the soil boundary. A load of value 5164.2 kg is applied at nodal point at the centre of the block.

The type of analysis type taken is of harmonic with analysis option as Full method. The stiffness-matrix multiplier and constant damping ratio have been taken as 0.005 and 0.01 respectively. The results have been viewed by using general postprocessor (POST26) at the nodal point of loading.

5.2.1 Soil Type: Sandy Soil

The amplitude of vibration in vertical direction is found to be equal to 0.094 mm at operating frequency of 7.5 Hz. But in horizontal direction the value obtained is less than vertical direction which is equal to 0.00525mm (Table 7).

5.2.2 Soil Type: Plastic Clay

In the second case the analysis is done for plastic clay having Young's modulus = 310×10^7 kg/m^2 and mass density = 180.0 kg-sec/m^4 and Poisson's ratio = 0.35. In this case of analysis, the amplitude of vibration in vertical direction is obtained as 0.303mm. In this case, the amplitude of vibration is almost 3.5 times of the value obtained in case of sandy soil. The value of amplitude of vibration in horizontal-direction is equal to 0.00375mm (Table 7). The value in horizontal-direction is less than that value in case of sandy soil. It may be due to the various properties of the soil like mass-density and Poisson's ratio.

5.2.3 Soil Type: Fine Sand

The type of soil taken is of fine sand which has less value of Poisson's ratio (i.e. $\mu = 0.20$). In this case, the amplitude of vibration in Y-direction at operating frequency of 7.5Hz. is equal to 0.205mm. The value in horizontal direction is obtained as 0.0041mm. In case of fine sand the amplitude of vibration in vertical direction is less than that of plastic clay but in horizontal direction value is more.

It is found that the effect of uniform soil below and surrounding the foundation are different and depend on type and properties of soil.

5.3 Influence of Layered Soil medium:

Amplitude and frequency of machine foundation depend upon the layering of soil below the foundation. It has different response at different depths for a particular type of soil. To study this effect, a model having the same foundation size 4.88 m x 2.1336 m x 0.91 m has been taken along with the soil to the both sides of the foundation equal to the four times of the half of the length of the foundation and depth of soil having seven layers (Fig. 4). Foundation is taken as fully embedded in the soil. The first layer of soil is the surrounding soil of the foundation and then four layers each having depth equal to half of the length of the foundation and two layers at the bottom having depth equal to the two times that of upper layers. In ANSYS program, at first, type of soil taken for analysis is sandy soil in upper layer and plastic clay for lower layers. Before meshing the model for finite element analysis the areas having same type of soil i.e. in first case, six layers from the top have been added by applying Boolean add command. Then, the areas are glued. The size of mesh has been taken as Smart Size of value 8.0. After completing the free meshing, the boundary conditions are provided with all fixed. The force equal to 5164.2 kg is applied at the mid node of the foundation and frequency range given for analysis is from 7 to 8, so that the response at operating frequency equal to 7.5 Hz can be read from the general postprocessor (POST26).

The value of amplitude of vibration in the first case has been obtained as 0.0930mm which is same as the value when all the layers (total depth 21.96 m) have been of the same type of soil as in the previous section. So, it can be concluded that the influence of lower layer on the amplitude of vibration is in significant. In the second case, the first five layers from the top (depth = 12.2 m) have been taken as sandy soil and the last two layers (depth = 9.76 m) are of plastic clay. The value obtained in this case is equal to 0.0835mm which is less than the previous value. So the influence of two lower layers affects the result (depth = 9.70 m). In the next step, the top four layers (depth = 9.70 m) are taken as sandy soil and last three layers as plastic clay. The amplitude of vibration is obtained as 0.0850mm which is greater than the just previous case which shows that the influence of the underlying soil. In further sets of analysis, the value goes on increasing from 0.0890mm to 0.1460mm. So, in the last set when only one layer below the foundation is of sandy soil and the remaining soil below is of plastic clay type, the amplitude of vibration is more than when the all the layers below the foundation were of sandy soil. Table 8 shows the different values in different set of analysis. Similar type of study has been done for sandy upper layer resting on lower plastic clay. The same trend of values has been obtained, which is shown in Table 9.

It can be concluded that the effect of soil below the foundation in the upper layer can be observed upto the depth 2B (B = half of the length of foundation) when its depth decreases from 2 B to B below the foundation the influence of soil lying in the lower layers predominates. In case of plastic clay in lower layers, its effect predominates when its depth changes from 2 B to B but in case of fine sand in lower layers, its influence is observed when its depth changes from 3 B to 2 B. So, this conclusion can be derived that in case of fine sand, the presence of layered soil affects the amplitude of vibration at greater depth when compared with plastic clay.

5.4 Effect of eccentric loading on the amplitude of vibration:

As previously mentioned that sometimes the centre of gravity of foundation and machine and that of the foundation area in contact with soil do not lie on the same vertical line due to error in placing of machine on the foundation. In this case, we have different modes of vibration coupled together, i.e. vertical, horizontal as well rocking. The effects of eccentric loading on the amplitude of vibration of foundation has been analysed by ANSYS program on the same block mentioned earlier. The meshing pattern obtained in concrete allowed us to choose a nodal point at a distance of 0.61 m from the mid nodal point.

At first, a load of a magnitude 5164.2kg has been applied at mid nodal point on the foundation and harmonic analysis has been done in the range of 7.0 Hz to 8.0 Hz so that the value can be obtained at the operating frequency of 7.5 Hz. In case of sandy soil, the amplitude of vibration in vertical direction is obtained as 0.093mm and in horizontal direction as 0.00525mm (Table10). Then the load at the mid-nodal point is deleted and a load of same magnitude of 5164.2 kg has been applied at an eccentricity of value 0.61 m. After analysis, the value of amplitude of vibration at the nodal point of loading is observed 0.0967mm in vertical direction which is greater than the previous value of 0.093mm at the point of loading. Similarly, the value in horizontal direction in case of eccentric loading has been observed as 0.00888mm which is greater than 0.00525mm.

Similarly, analysis for other two types of soil i.e. plastic clay and fine sand has been done and the observed values show the similar behaviour i.e. the amplitude of vibration increase with eccentricity. This is same for all the three types of soils analyzed. From this, it can be inferred that the machine should be placed on the foundation in such a way that the eccentricity should be nearly zero so that the value of amplitude of vibration does not exceed the permissible values.

5.5 Foundation Interaction by Coupling of Motion:

Interaction between foundations is a quite common occurrence in many industrial complex. Due to the lack of space available for setting up the machines, sometimes foundations for different machines are kept nearby each other. These foundations are subjected to vibration in different phases. In these cases, the performance of one foundation gets affected due to the vibration of nearby foundation. This is due to the propagating waves in the soil medium which is caused by the vibrating machines. Although, there are various methods to reduce this effect like, providing piles to the foundation, construction of trenches between the foundations etc. For that, but it is very much necessary to know the effect of the vibrations of foundation in different phases along with the spacing between them and the phase difference in loading them. Due to vibration caused by machines, foundations vibrate in different modes and the waves generated in soil are also in different phases. The net effect of their interactions will result in coupling of motions.

5.5.1 Effect of Spacing between Foundations:

For studying the effect of spacing between foundations by ANSYS program, analyses have been done at three distances at 3 B, 2 B and B, where B is the half of the length of foundation. The soil below the foundation has been considered as 4 B with same distance either sides of each foundation. The distance between them is kept at 3 B. In the preprocessing stage areas having same properties have been added by applying Boolean add command and the areas have been attributed according to their properties.

The Smart Size chosen for meshing is basic 8.0. After meshing has been completed done, the nodes lying on the boundary of soil have been fixed for all degree of freedom. The size of the mesh lying on foundations and the soil between the foundations are refined by half for better accuracy. The model and their meshing pattern have been shown in Fig. 5.1 and Fig. 5.2 respectively. To study the mode of vibration of both foundations along with soil, a modal analysis has been done. The mode shape of vibration of the model at operating frequency 7.5 Hz. has been animated. By animation, it is found that the two foundations are not vibrating in the same phase. When one foundation is moving upward, the other one is moving downward (Fig. 5.3).

5.5.2 Load Acting on Foundation A_1 only:

A load of value 5164.2 kg is applied at the mid nodal point of the foundation A_1 and harmonic analysis is done. The results are obtained by POST26 (general postprocessor) by defining the variable to be read. The amplitude of vibration of the point of loading of foundation A_1 in vertical direction is obtained as 0.081 mm and in horizontal direction as 0.00375 mm, which are shown in Table 11. The value of amplitude of vibration of same size of single foundation lying over same type of sandy soil at a spacing of 3B (7.32m) is obtained as 0.093 mm in vertical direction and 0.00525 mm (Table 10) in horizontal direction. By comparing these two values, it can be inferred that the presence of one foundation, affects the amplitude of vibration of another foundation both in horizontal direction and in vertical direction.

In the second case of study, distance between the foundations A_1 and A_2 is reduced to 2 B (= 4.88 m) to study the effect of distance on the amplitude of vibration. The model for analysis and their meshing pattern along with the mode of vibration at operating frequency 7.5 Hz. have been shown in. Fig. 6.1, Fig. 6.2 and Fig. 6.3 respectively. The amplitude of vibration of foundation A_1 is observed as 0.0735 mm in vertical direction and 0.0016 mm in horizontal direction (Table 11). These values are less than the values obtained in case of 3 B distance between the foundations. But the amplitude of vibration of foundation A_2 in vertical direction is 0.034 mm (Table 11) which is greater than the value obtained at the distance of 3 B, but in the horizontal direction, the value of amplitude of vibration is reduced. All these variations are related to the effects caused by the coupling of motions of the foundations. Above discussions present the effect of the coupling of motion.

When the distance between the foundations is reduced to B(= 2.44 m) i.e. the half of the length of the foundation, the amplitude of vibration of foundation A_1 is vertical direction is observed as 0.0765 mm and in horizontal direction is 0.00446 mm (Table 11). Both these values are greater than those values obtained in the previous case of study (i.e. at the distance of 3B and 2B) . The model and their meshing pattern and mode of vibration at operating frequency have been shown in Fig. 7.1, Fig. 7.2 and Fig. 7.3 respectively. The analysis shows that the amplitude of vibration of foundation A_2 in vertical direction is 0.0378 mm and in horizontal direction is 0.01475 mm(Table 11). It is observed that the value 0.0378 mm in vertical direction is greater than that in case of 2B distance but the value in horizontal direction is reduced (Table 11).

5.5.3 Loads Act on both Foundations A_1 and A_2 :

After the completion of above analysis in which load acts only on foundation A_1, the effect on amplitude has been observed when loads have been applied on both the foundation A_1 and A_2 having same magnitude of 5164.2 kg situated at a distance of

3B(= 7.32m). The amplitudes of vibration of foundations A_1 and A_2 have been shown in Table 12. The amplitude of vibration of foundation A_1 in vertical direction is 0.0618 mm and in horizontal direction 0.01659 mm (Table 12) which are less then the values obtained when no loading is applied on foundation A_2. But the amplitude of vibration of foundation A_2 in vertical direction which is 0.0558 mm and in horizontal direction which is 0.0254 mm (Table 12) which is more than the previous analysis done in case of no loading on foundation on A_2. The values of amplitudes are obtained from Fig. 8.1 to Fig. 8.4.

The stress distribution in horizontal direction and in vertical direction have been shown in Fig. 9.1 and Fig. 9.2 respectively. The strain at operating frequency 7.5 Hz in horizontal direction and in vertical direction have been shown in Fig. 10.1 and Fig. 10.2 respectively.

To study the effect of distance between the foundations, the distance between the blocks is reduced to 2B = 4.88 m which is same as the length of the foundation. In this case the amplitude of vibration of foundation A_1 in vertical direction is 0.064 mm and in horizontal direction is 0.028 mm (Table12). These values are, of course, greater than the values obtained as in previous case. It can be concluded that as the foundations come closer, the interaction between them increases. Similar results are obtained in foundation A_2. The amplitude of vibration of foundation A_2 in vertical direction is 0.088 mm and in horizontal direction is 0.080 mm (Table 12) which are greater than the previous case of foundation at a 3 B (7.32m) distance.

It has been observed from the above discussions that the amplitude of vibration of foundation A_1 decreases due to the presence of loading on foundation A_2. When the distance between the foundation is further reduced to B (2.44m) and analysis is done in this condition, results obtained show some changes in its pattern as obtained in previous cases. The amplitude of vibration of foundation A_1 in vertical direction is 0.0875 mm and in horizontal direction is 0.0239 mm (Table 12). So, the value of amplitude in vertical direction increases from 0.081 mm to 0.0875 mm (Table 11) due to the presence of same loading condition on the foundation A_2. The value of amplitude of vibration increase from 0.00375 mm to 0.0239 mm. Similarly, the amplitude of vibration of foundation A_2 increases from 0.0378 mm to 0.0961 mm in vertical direction and the value decreased from 0.01475 mm to 0.0088 mm in horizontal direction. Thus, the coupling of motion predominates when the distance between the foundations is less than 2B (4.88m).

5.5.4 Conclusion :

It is concluded that the various physical quantities related to the performance of machine foundation, like stress, strain, frequency and amplitude of vibration may change due to the foundation interaction which is caused by coupling of motions. From the above studies of three cases of distances between the foundation, it is found that the effect of interaction in prominent, when the foundations are situated at a distance of the half of the length of the foundation.

5.5.5 Effect of Different Phases of Loading on Foundation Interaction:

Initially the effect of coupling of motion on the performance of machine foundation due to loads on both the foundations (A_1& A_2) having same magnitude and same phase . But, there may be effects of difference in phase of loadings on the foundation interaction. It is studied on the same model of analysis by changing the phase

(θ) of loading on the second foundation (A_2). Real part is given as P Cos θ and imaginary part as P Sinθ. The value of θ has been changed from 0^0 to 360° at a step of 45°. The results obtained have been shown in Table 13. The results are obtained for foundation A_1 at its point of loading. So, these values are compared with the previous analysis when there was no change in phase between the two loadings. The values in this case is 0.0810 mm in vertical direction and 0.00375 mm in horizontal direction. The amplitude of vibration is decreased from 0.0810 mm to 0.0618 mm is vertical direction and from 0.00375 mm to 0.0165 mm in horizontal direction when phase change of 45° has been applied on the loading on foundation A_2. Then change of phases have been applied at 90°, 135°, 180°, 225°, 270°, 315° and 360° and values of amplitude of vibration in vertical direction have been recorded as 0.10 mm, 0.108 mm, 0.104 mm, 0.088 mm, 0.0670 mm, 0.0532 mm and 0.0610 mm respectively. So, it is found that the values increase up to the phase change of 135°, then the values decrease up to the phase change of 315^0 and again goes in the increasing trend. But in horizontal direction, the values increases up to the phase change of 270° and then decreases up to 360° which is shown in Table 13.

It is concluded that the phase change affect the coupling of motion of the machine foundations. Thus, the interaction between foundations results in different modes of vibration which affects the value of amplitude.

6. Experimental Work

6.1 Introduction :

The analysis carried out using ANSYS software has been presented in previous chapters. However, it would be better if the results of the analysis are substantiated by a field testing programe. In view of that, a few block vibration tests as suggested by IS:5249:1969 have been carried out and the details are presented in this chapter.

6.2 Experimental Set up (As per IS 5249 : 1969)

The equipments to be used for this block vibration test can be broadly classified into two categories : (Ref 14).

(i) One required for inducing a known pattern of vibration (e.g. sinusoidal waveform).

(ii) Second, required to measure the vibration.

(i) Equipments for inducing vibration

The principal unit of this group of equipment is the vibrator. It is also called the oscillator. There are different types of oscillator, like mechanical, electromagnetic, hydraulic, etc, depending upon the principle on which each type works. A mechanical type oscillator is commonly used for the testing on the machine foundation. Force generated by a typical rotating mass type oscillator in which a single mass m_e is placed on a rotating shaft at an eccentricity e from the axis of rotation with angular frequency ω. Such an arrangement causes a vertical force given by

$$P = m_e e\, \omega^2 \sin \omega t \qquad (6.1)$$

When two equal masses mounted on two parallel shafts at the same eccentricity, the horizontal components of forces cancel one another and the vertical components are added up. The amplitude of vertical force produced is given by :

$$P_o = 2\, m_e\, e\, \omega^2 \qquad (6.2)$$

The equipments associated with a mechanical oscillator (Fig.11.1) are an electric motor (Fig.11.2) and a speed control unit (Fig. 11.3). The mechanical oscillator consists of two shafts so arranged that they rotate in opposite directions at the same speed when one of them is driven by a motor through a belt or a flexible shaft. Such an arrangement induces a vibratory force which is unidirectional at the base of the oscillator. Either a vertical or horizontal dynamic force can be produced by changing the orientation of the oscillator. The speed of the motor is controlled by a speed control unit which helps controll the rotational speed of the oscillator. Thus, the frequency of vibration caused by the oscillator is controlled .

(ii) Equipments used for measuring vibration response

The equipments under this group are a transducer, an amplifier and a recorder. The transducer which is known as vibration pick-up converts the physical quantity to be measured into an electrical signal which is related to the magnitude of vibration through a calibration factor. The voltage signal sensed by the transducer is amplified by an electronic unit known as "preamplifier". The amplified signal is then fed to the recorder for recording the waveform or to an oscilloscope for a visual display of the same. If the transducer is sufficiently sensitive and is of self-generating type (i.e., voltage is induced in the transducer cable with the movement of transducer) the use of a preamplifier may be dispensed with. It may be either displacement velocity or acceleration type depending on whether the electrical voltage signal induced in it is proportional to one or other of these physical quantities. The transducer may be classified as resistive type (e.g., strain gauge based transducer), inductive type or piezo-electric type depending on the principle of design and construction of the transducer.

The present experimental work is planned and executed using the infrastructual facilities available at Central Building Research Institute, Roorkee.

6.3 Present Experimental Work :

6.3.1 Infrastructural facilities available at CBRI, Roorkee

The various infrastructural facilities available at CBRI, Roorkee are given below

(i) 'LAZAN' type mechanical oscillator

This oscillator (Fig.11.1) is driven by a 10 HP, DC motor and is capable of generating sinusoidal force of 3020 kg peak-to-peak. The speed of DC motor can be varied through the speed control unit. Provision also exist to vary the force by changing the degree of eccentricity of the rotating unbalance masses of the mechanical oscillator. A digital display for measuring the rotating speed of mechanical oscillator is fixed on speed control unit. Adequate protection are provided in the speed control unit for protection of motor against overload and field failure.

(ii) Duel Channel FFT Spectrum Analyser (FA-2005)

It clarifies the level and source of vibration and analyses the dynamic characteristics. Two channels can be connected to this analyser. The frequency range lies between 1 Hz to 100 Hz. It displays the analysis in time domain, frequency domain and amplitude domain (Fig.12.1).

(iii) Galvibro (VM-7000)

It is highly sensitive and accurate handy type vibration meter. It can measure peak /rms acceleration, peak/rms velocity and peak to peak displacement(Fig.12.2).

(iv) Piezo- resistive acceleration pick-up (VP-7000/VP-7002)

It is highly sensitive and accurate light weight vibration prickup at low frequency. Its frequency range is 0.3 Hz to 300 Hz and measuring range is + 2g, + 20 m/sec^2. and it can measure in one direction. Natural frequency over 500 Hz can be measured by this pickup. Its sensitiveity is 102 mV/m/Sec^2 (Fig.12.2).

(v) Measuring amplifier (TYPE 2525)

It is used for automatic gain adjustment and level monitoring with alarm output and signal overload indication and integrator. Its frequency range in acceleration is 0.2 Hz to 100 Hz and in velocity 1 Hz to 10 Hz and for displacement it is 1 Hz to 1 kHz. Its signal peak detector is + peak to - peak.

6.3.2 Testing Procedure

A plain concrete block of size 2.25 m x 0.75 m x 0.70 m (Fig.13) was constructed in the campus of CBRI, Roorkee. The standard size of the block as per IS:5249:1969 is 1.5m x 0.75m x 0.70m. However, the dimension of the present block in along the length is increased to 2.25m such that the oscillator can be placed in an eccentric position also. For casting the block, the site was properly cleaned. The brick walls were constructed all around to create the space to be filled up by concrete. Then M15 concrete mix was filled up to a level where the bottom of the anchor bolt would lie and then compacted properly. The location of the anchor bolts was marked with respect to the position of base plate planned. The anchor bolts have been tied vertically to iron rods (a temporary arragement made) at the located positions. Similarly, to make arrangement for providing eccentric loading, a pair of anchor bolts were fixed at a certain distance so that the base plate should fit with bolt properly. After fixing the position of anchor bolts, concrete mix had been filled up to the depth of the block keeping some portion of anchor bolts was above the block to fix the base plate with oscillator. The curing of the block was done properly for twenty eight days.

After complete setting of block, the brick walls and other iron rods were removed carefully from all around the block. The soil from the sides of the block is removed up to the bottom of the block so that there is no any effect of embedment of block. After that, an arrangement was made to place the motor to operate the oscillator. A pavement of bricks was made to place the motor and intermediate supports were made of bricks to keep the cable straight such that the required vibrating force is supplied to the oscillator without losses. A diagram of an arrangement of motor, oscillator, techogenerator etc. is shown in (Fig. 14).

6.3.3 Testing Phases :

The tests were conducted in four phases as explained below :

(i) Oscillator at the centre of block:

(a) Vertical vibration test :

At first, the base plate was fixed over the grouted bolts and was tightened with the help of nuts (Fig.15). Then the oscillator was placed. The oscillator was attached to the block with the help of four nuts and bolts (Fig.16). The eccentricity of the rotating masses inside the oscillator was fixed at an angle of 90^{0} . The oscillator was connected to the motor by a long flexible coil, which rotates the shaft of the oscillator. It was rotated at the same speed as the speed of the motor. The speed of the motor was controlled by a panel control. Thus, the speed of the oscillator was controlled. The permissible frequency range up to which the oscillator can be rotated was 25 Hz. The operating frequency of the block was measured by both the tecogenerator and FFT spectrum analyser. The value of amplitude of vibration was noted from the vibration meter, [GALVIBRO (VM-7000)], which is highly sensitive and accurate handy type vibration meter. The value of amplitude of vibration noted from the vibration meter, gives the value from peak-to-peak. It means that the actual amplitude of vibration is half of the values noted from the equipment. The set-up for vertical vibration with oscillator at the centre of the block has been shown in Fig 17.

With the help of speed control unit the speed of the oscillator was increased from 3Hz to 19 Hz at the step of 2 Hz increments. For each setting of speed, the amplitude of vibration of the block was noted through vibration meter and the speed (rotational) was measured through FFT spectrum analyzer. A graph of frequency versus amplitude of vibration was plotted. The resonant frequency and corresponding maximum amplitude of vibration was noted. Then, the value of coefficient of elastic uniform compression is given by the expression:-

$$C_u = \frac{4\pi^2 f_{nz}^{\ 2} m}{A_b} \tag{6.3}$$

where,

f_{nz} = Resonant frequency in Hz.

m = Mass of the machine and foundation

A_b = Base area of the foundation in contact with soil

(b) Horizontal vibration test

To study the effect of horizontal vibration, the oscillator was set up at the centre of the top surface of the block by keeping it normal to the previous position, which is shown in Fig.18. For setting the oscillator at this position, the bolts were loosened in the previous set up of vertical vibration and the oscillator was kept in the required position and tightened with the base plate with other set of nuts and bolts. To record the amplitude of vibration, the pickups were kept on the vertical force of the block and it was turned on its side so that it can capture the horizontal response of the block. Horizontal loading on the block produces both types of motions, namely sliding and rocking of the block. The frequency of rotation was the oscillator was recorded by both the tecogenerator and the FFT spectrum analyser. The amplitude of vibration of recorded by vibration meter GALVIBRO (VM-7000)

Horizontal amplitude versus frequency curves were plotted for each of the three pick-ups show two peaks, the frequencies at which correspond to the two resonant frequencies (f_x) of the coupled system (sliding and rocking)

The values of various dynamic properties of soil for the testing- site have been obtained, which are given below:

C_u = coefficient of elastic uniform compression = 5.01×10^6 kg / m^3

C_θ = coefficient of elastic non-uniform compression = 10.02×10^6 kg / m^3

C_τ = coefficient of elastic uniform shear = 2.5×10^6 kg / m^3

C_ψ = coefficient of elastic non-uniform shear = 3.75×10^6 kg / m^3

(ii) Oscillator at the eccentric point

(a) Vertical vibration test

After the completion of both the vertical and horizontal test on the block at its centre, the oscillator was shifted to eccentric point. The eccentricity provided is equal to 63.8 cm. After fixing the base plate at the eccentric point, the LAZAN type oscillator was fixed in such a way to produce vertical vibration at that point. The arrangement has been shown in Fig.19. Since the oscillator is placed at the eccentric point, the speed of oscillator was increased slowly. The frequency upto 25 Hz can be applied to the oscillator by motor.

(b) Horizontal vibration test:

Similar arrangement, as mentioned in case of horizontal loading at the central point was made at the eccentric point. (Fig 20)

6.4 Results and Discussion:

The value of resonant frequency and maximum amplitude of vibration have been obtained as 8.42 Hz and 0.136 mm in case of vertical mode of vibration when the oscillator was fixed at the central point (Fig. 21). Whereas, in case of eccentric loading i.e. the oscillator was placed at an eccentricity of 0.638 m, the value of resonant frequency has been obtained 6.0 Hz (Fig. 23). This value is less than the value in case of central loading. Similarly, the value of the peak amplitude of vibration at the eccentric point obtained is 0.02805 mm which is very less than the value obtained in case of central loading Comparison of results have been shown in Table 14. Therefore, both the resonant frequency and the peak amplitude of vibration have decreased in case of eccentric loading in vertical mode of vibration which is due to coupling of various modes of vibration.

When the horizontal test was conducted on central point, the values of the first resonant frequency is obtained 6.4 Hz and the peak amplitude of vibration 0.0025mm (Fig. 22). When the test was conducted on eccentric point, the values of the resonant frequency and the amplitude of vibration has been recorded as 5.26 Hz and 0.002mm respectively.(Fig. 24). The values in case of eccentric loading are less than the values in case of central loading. But there is not much difference in values between these two types of loading in case of horizontal first mode of vibration.

Finally, it can be concluded that the values of peak amplitude of vibration as well as resonant frequency have decreased in case of eccentric loading due to coupling of various modes of vibration.

6.5 Analysis of Experimental Work by ANSYS program:

The experimental work has been analyzed by ANSYS program. The model used for the analysis by the ANSYS program has been shown in Fig 25. The material properties of concrete has been taken similar to those used in the analysis of bench mark problem in the chapter 4 (Table 1) the various properties of soil has been taken as E= 508856.7 kg/m^2 and mass density = 165.0 kg-sec m^{-4} and Poisson's ratio = 0.3. The Smart Size for free meshing is chosen as basic 6.0 (default) and element type chosen is solid eight noded (PLANE 82) the elements formed in the model has been shown in Fig.26. The elements in the concrete block has been refined by half to get the more number of elements in this potion so that greater number of nodes be obtained to apply load at the required eccentric point nearer to that of tested point of application. At first, a modal analysis has been done to find out the natural frequencies in vertical and horizontal mode of vibration which are shown in Fig. 27 and Fig. 28 respectively. The values are found to be 7.5 Hz and 6.42 Hz in respective modes of vibration.

6.5.1 Various steps of Analysis for Forced vibration :

For analysing the block for vertically applied load at central point of the block, loads corresponding to the various operating frequencies have been applied at the center. The values of forces are obtained from the Fig. 29 which is a graph of operating frequency (in rpm) versus force (in KN) for the oscillator set at eccentricity of rotating masses at an angle of 90^0. The amplitude of vibrations of the block has been obtained for various frequencies. The value of maximum amplitude of vibration has been obtained as 0.211 mm for an operating speed of 7.5 Hz. Similarly, the horizontal forces corresponding to the various frequencies have been applied at the central point and the value of maximum amplitude of vibration has been optioned as 0.390 mm at an operating frequency of 6.42 Hz, which is shown in Table 15.

To find out the effect of coupling of motion on the performance of machine foundation, loads are applied vertically at an eccentricity of 0.56m corresponding to various frequencies of the oscillator. The eccentricity at which the actual load was applied in the testing is a function of the size of the base plate. It was kept at 0.638 m from the center. However in the analysis the mesh size has been refined to the best of the knowledge to get a node nearer to the point. The nodes in the mesh were found to be at 0.56 m and at 0.75 m. Out of which a node at 0.56m has been taken for computational purposes. The value of amplitude of vibration is obtained as 0.140mm at the frequency of 8.0 Hz.

Horizontal forces corresponding to various frequencies have been applied at the same point . The maximum amplitude of vibration has been obtained as 0.39 mm at an operating frequency of 6.42 Hz.

6.5.2 Comparison of the results obtained from the experiment and the results obtained from ANSYS program.

From the results obtained from experiment the values of maximum amplitude of vibration of foundation and corresponding frequency have been reduced in case of eccentric loading. But the results obtained from ANSYS program for vertical loading at eccentric point shows that the value of maximum amplitude of vibration is reduced but the corresponding value of frequency has been increased (Table 15). In case of horizontal loading at eccentric point, the values of maximum amplitude of vibration and corresponding frequency remain the same as in case of central loading.

A better correlation could be obtained between the results from ANSYS program and experiments by analysing the problem using 3-D analysis, though it is time consuming. Further the accuracy of the analysis could be improved by considering the radiation damping of the soil into account.

7. Conclusions

The effects of coupling of motions on the frequency and amplitude of vibration of machine foundation have been studied by using a finite element package named ANSYS. The conclusions are limited to the extent of the experiences gained with the use of the facilities (e.g. Basic structural analysis and Dynamic analysis) of the package.

An experimental work has also been done to study the effect of eccentric loading on the foundation, which gives some useful results.

7.1 Applicability of Preprocessor and Postprocessor of ANSYS to the Present Problem:

The preprocessor of the ANSYS package is very efficient in generating the model for a machine foundation with soil medium having different layers. The construction of layers of changing depth and attributing their properties can be done easily with the various preprocessor commands like Boolean add and glue. The meshing of the model is done by free or mapped meshing according to the requirements of the analysis. Not only this, the refinement of the elements can be done according to own choice and requirement. Thus, the model generation needs a thorough knowledge of various commands and features of AN SYS. For different types of analysis, a number of methods are available. Among these, the suitable one can be chosen according to the problem. There is an excellent facility to animate the mode shapes of the vibration of the model. The contour plots of various physical quantities like displacement, stress and strain etc.

be obtained from the postprocessor (POST 1) command. In general postprocessor (POST 26), the listing of the any variables defined can be done and the graphs of the corresponding variables can be obtained on the same screen for comparison of the results. All these contour plots, graphs and animated pictures can be captured by plot controls and can be used for display. Thus, this package has very powerful postprocessor along with utility menu.

7.2 Coupling of Motion:

An eccentric loading and interaction between two foundations cause the coupling of motion of a machine foundation. When the foundation is subjected to dynamic loading at an eccentricity, the responses of the foundation changes from the case, when it is loaded centrally. The amplitude of vibration increases in at the point of loading in case of an eccentricity. Therefore, the machine should be placed on the foundation in such a way that the combined centre of gravity of machine and foundation should lie on the same vertical line passing through the centroid of the base area of the foundation in contact with the soil.

Spacing between the foundations is an important factor which influence the performance of the machine foundations lying nearby. At different spacing, the effects observed are different. When the two foundations are situated at a distance less than the length of the foundation, the amplitude of vibration of the foundation increases.

Similarly, when the two foundation are subjected to loadings of different phases, the interaction between them is observed. The amplitude of vibration increase upto certain value of phase difference, but it starts decreasing with the increase with the phase difference.

Apart from the above important factors, some others, like the various properties of the soil below the foundation and the layering in the soil medium affect the response of the machine foundation. In case of plastic clay, having greater value of mass density, the amplitude of vibration is more than in case of fine sand. Similarly, the effect of plastic clay in lower layers influences the performance of the machine foundation at the less depth if compared with the fine sand.

7.3 Suggestions for future study:

It is suggested for future study that more number of block vibration tests with oscillator at different eccentricities should be carried out. Problems of machine foundation should be analyed by 3D model, although it is time consuming. At the same time, effects of radiation damping in the soil medium along with the nonlinear properties of soil should be taken into account for analysis.

Table 1: Required informations for analysis of the cantilever beam

Preference of analysis	Structural
Element type used	PLANE 42 (Solid 4-noded) PLANE 82 (Solid 8-noded)
Analysis type	Modal
Analysis Option	Block Lanczos method
Material Properties	
Young's Modulus of elasticity (E) (in kg/m^2)	2.1 x 10^9
Mass density (ρ) (in Kg-sec/m^4)	262.4
Poisson's Ratio	0.24

Table 2: Comparison of results for cantilever beam

Mode of Vibration	Natural frequency obtained from formulae (Eq. 4.1 to 4.6) (Hz)	Natural frequency obtained from ANSYS analysis using solid-4 noded element (Hz)	Natural frequency obtained from ANSYS analysis using solid-8 noded element (Hz)
First Mode of Bending	15.31	14.37	14.37
Second mode of bending	95.96	89.93	89.70
Third mode of bending	268.73	250.94	249.59
First mode of axial vibration	790.00	741.91	741.83
Second mode of axial vibration	2370.00	2226.60	2225.40

Table 3 : Material properties

Material Properties	Soil	Concrete
Young's modulus of elasticity (E) in Kg/m^2	2.8×10^7	2.1×10^9
Mass density (ρ) in $Kg\text{-}sec/m^4$	174.77	262.4
Poisson's ratio	0.33	0.24

Table 4: Effect of different types of damping on the amplitude of vibration in case of sandy soil

Sl. No.	Types of damping		Amplitude of vibration of foundation (mm)
	Stiffness matrix multiplier	Constant damping ratio	
1	0.005	0.00	0.104
2	0.005	0.10	0.094
3	0.005	0.15	0.079
4	0.01	0.15	0.071
5	0.05	0.15	0.043

Table 5 : Material properties of concrete and different soils

Material	Young's Modulus Kg/m^2	Mass density $Kg,sec\ m^{-4}$	Poisson's ratio
Concrete	$2.1x10^9$	245.0	0.24
Sandy soil	$2.61x10^7$	163.67	0.33
Plastic silty clay with sand and organic silt	$0.310x10^7$	180.0	0.35
Fine sand	$0.85x10^7$	165.0	0.20

Table 6: Input data for ANSYS Analysis

Element used	PLANE 82 (Solid-8 noded)
Analysis type	Harmonic
Analysis option	Full method
Operating frequency	7.5 Hz
Force	5164.2 kg
Boundary condition	All fixed
Stiffness-matrix damping	0.005
Constant damping ratio	0.10 i.e. (10%)

Table 7: Influence of soil properties on the performance of machine Foundation

Type of soil	Amplitude of vibration in vertical direction (mm)	Amplitude of vibration in Horizontal direction (mm)
Sandy soil	0.094	0.00525
Plastic clay	0.303	0.00375
Fine sand	0.205	0.0041

Table 8: Effect of layered soil on the performance of machine foundation

Depth of top layer (L_1) in metres with type of soil	Depth of bottom layer (L_2) in metres with type of soil	Amplitude of vibration (in mm)
17.08 (=7B) Sandy soil	4.88 (=2B) Plastic clay	0.0930
12.2 (=5B) Sandy soil	9.76 (=4B) Plastic clay	0.0835
9.76 (=4B) Sandy soil	12.2 (=5B) Plastic clay	0.0850
7.32 (=3B) Sandy soil	14.64 (=6B) Plastic clay	0.0890
4.88 (=2B) Sandy soil	17.08 (=7B) Plastic clay	0.1460

Table 9 : Effect of layered soil on the performance of machine foundation

Depth of top layer (L_1) in metres with type of soil	Depth of bottom layer (L_2) in metres with type of soil	Amplitude of vibration (in mm)
17.08 (=7B) Sandy soil	4.88 (=2B) Fine sand	0.0904
12.2 (=5B) Sandy soil	9.76 (=4B) Fine sand	0.0835
9.76 (=4B) Sandy soil	12.2 (=5B) Fine sand	0.0850
7.32 (=3B) Sandy soil	14.64 (=6B)Fine sand	0.0890
4.88 (=2B) Sandy soil	17.08 (=7B) Fine sand	0.1460

Table 10 : Effect of eccentric loading on the amplitude of vibration of foundation at an operating frequency of 7.5 Hz.

Type of uniform soil below the foundation	Amplitude of vibration of foundation at the point of loading (mm)		Amplitude of vibration of foundation at the point of loading (mm)	
	Force acts at centrally eccentricity e = 0		Force acts at an eccentricity e = 0.61 m	
	Vertical	Horizontal	Vertical	Horizontal
Sandy soil	0.093	0.00525	0.0967	0.00888
Plastic clay	0.0304	0.00375	0.331	0.0328
Fine sand	0.205	0.0041	0.213	0.011

Table 11 : Effect of spacing between Foundations on their performance

Distance between foundation A_1 & foundation A_2 (in metres)	Force acting on foundation A_1 only			
	Amplitude of vibration of foundation A_1 at its mid point (mm)		Amplitude of vibration of foundation A_2 at its mid point (mm)	
	Vertical	Horizontal	Vertical	Horizontal
3B=7.32	0.081	0.00375	0.0275	0.0248
2B = 4.88	0.0735	0.0016	0.034	0.0168
B = 2.44	0.0765	0.00446	0.0378	0.01475

Table 12 : Effect of spacing on their performance.

Distance between foundation A_1 and Foundation A_2 (in meters)	Force acting on foundation A_1 & A_2			
	Amplitude of vibration of foundation A_1 at its point of loading (mm)		Amplitude of vibration of foundation A_2 at its point of loading (mm)	
	Vertical	Horizontal	Vertical	Horizontal
3B=7.32	0.061	0.0165	0.0558	0.0254
2B = 4.88	0.064	0.028	0.088	0.080
B = 2.44	0.0875	0.0239	0.0961	0.0088

Table 13 : Effect of phase-change of loading on the foundation interaction by coupling of motion for sandy soil

Ref. (Table 11):

Amplitude of vibration of foundation A_1 at its point of loading when load acts on foundation A_1 only	Vertical = 0.0810 mm Horizontal = 0.00375

Value of phase change (θ) of loading on foundation A_2	Amplitude of vibration of foundation A_1 at its point of loading when loads act on both A_1 & A_2 but with changing phase on A_2 only.	
θ (in degree)	Vertical (mm)	Horizontal (mm)
0.0	0.610	0.0165
45	0.082	0.00385
90	0.100	0.013
135	0.108	0.0148
180	0.104	0.0177
225	0.088	0.0199
270	0.0670	0.0207
315	0.0532	0.01925
360	0.0610	0.0165

Table 14: Amplitude of vibration and resonant frequency for both the central loading as well as eccentric loading from experiment

Mode of Vibration	Central Loading		Eccentric Loading	
	Resonant Frequency (in Hz)	Peak amplitude of Vibration (in mm)	Resonant Frequency (in Hz)	Peak amplitude of Vibration (in mm)
Vertical	8.42	0.136	6.0	0.02805
Horizontal mode	6.40	0.0025	5.26	0.002

Table 15: Amplitude of vibration and resonant frequency for both the central loading as well as eccentric loading obtained from ANSYS program.

Mode of Vibration	Central Loading		Eccentric Loading	
	Resonant frequency (in Hz)	Amplitude of vibration (in mm)	Resonant frequency (in Hz)	Amplitude of vibration (in mm)
Vertical	7.5	0.211	8.0	0.140
Horizontal	6.42	0.390	6.42	0.390

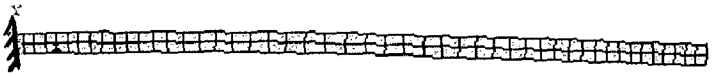

Fig. 1.1 First mode of bending of a cantilever beam

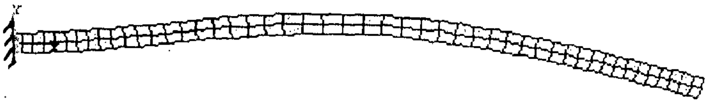

Fig. 1.2 Second mode of bending of a cantilever beam

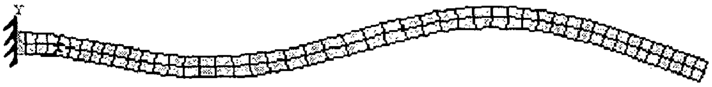

Fig. 1.3 Third mode of bending of a cantilever beam

Fig. 1.4 First mode of axial compression of a cantilever beam

Fig. 1.5 Second mode of axial compression of a cantilever beam

Fig. 1 Bench Mark Problem; Cantilever Beam (Richart)

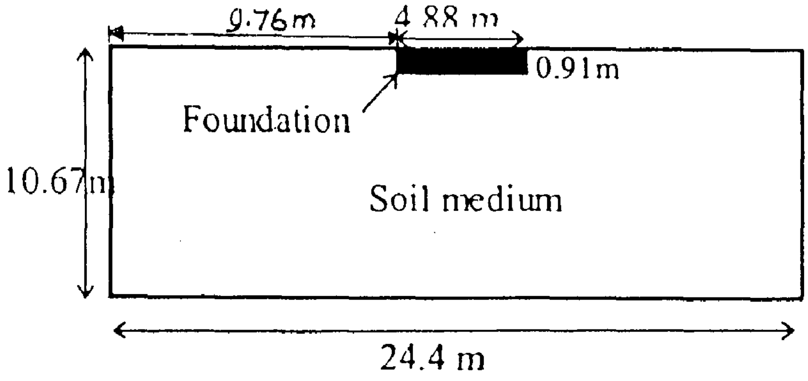

Fig. 2.1 A machine foundation and surrounding soil (Richart)

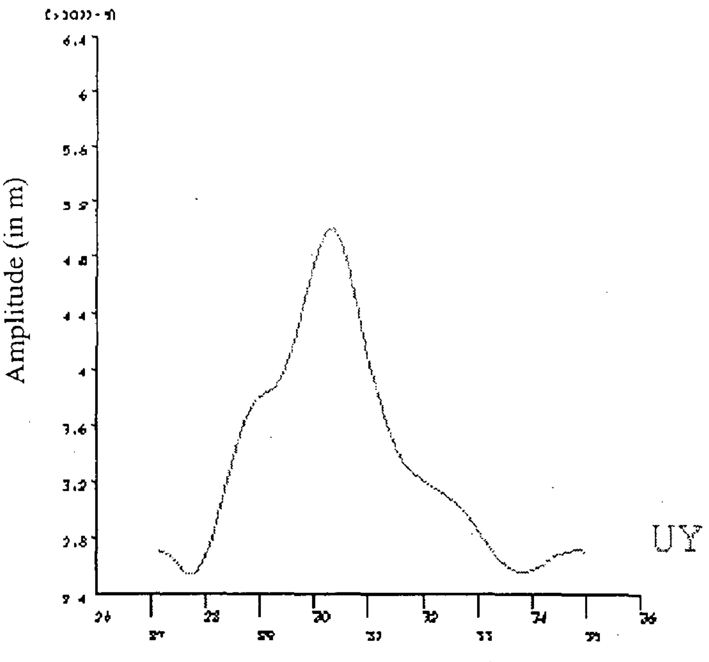

Fig. 2.2 Frequency versus amplitude of vibration of the block foundation

Fig. 2 Bench Mark Problem; Vibration of block (Richart)

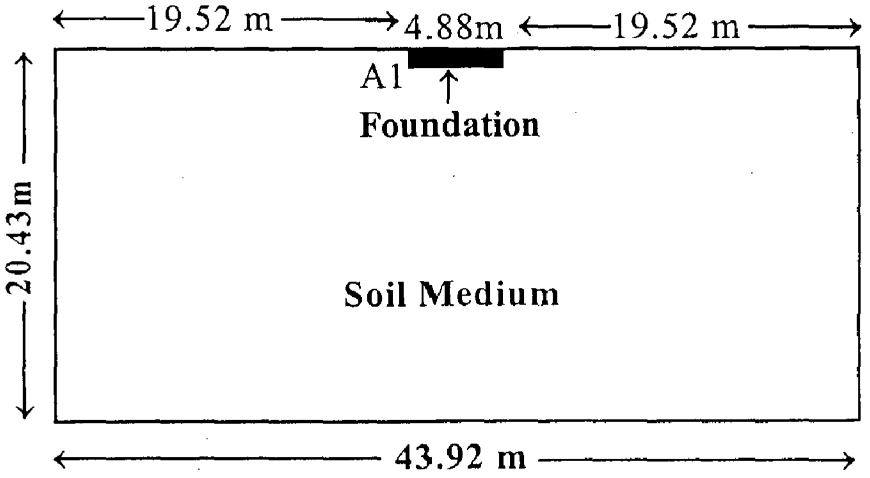

Fig. 3 Uniform soil below the foundation

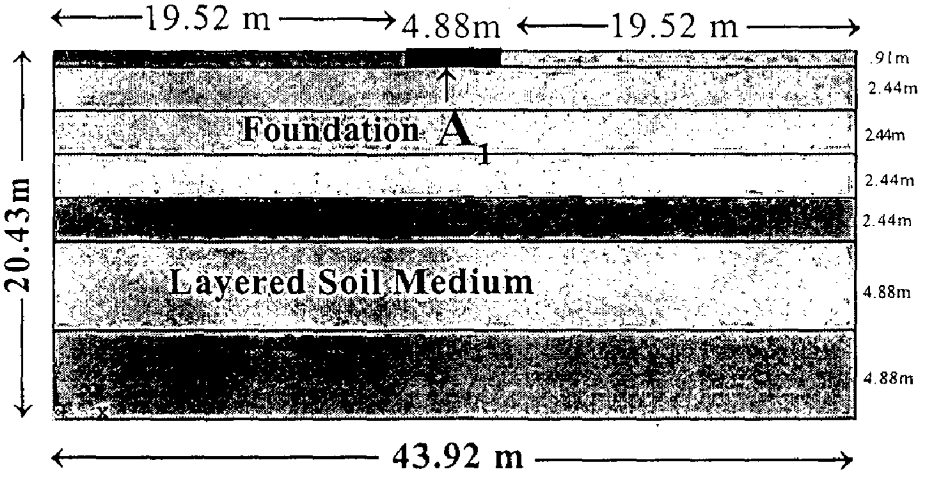

Fig. 4 Layered soil below the foundation

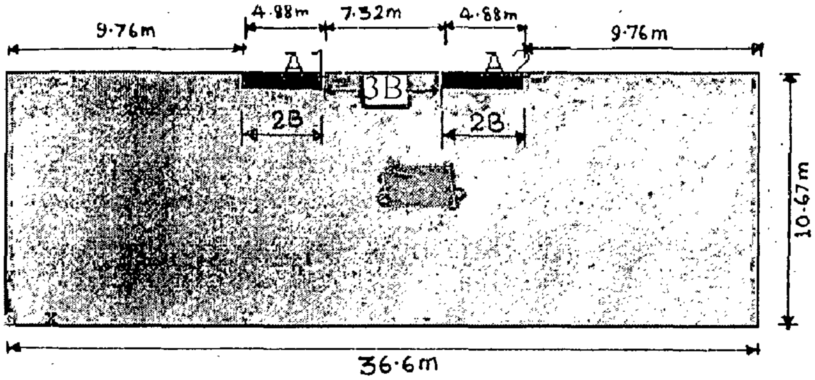

Fig. 5.1 Foundations located at a distance of 3B (= 7.32m)

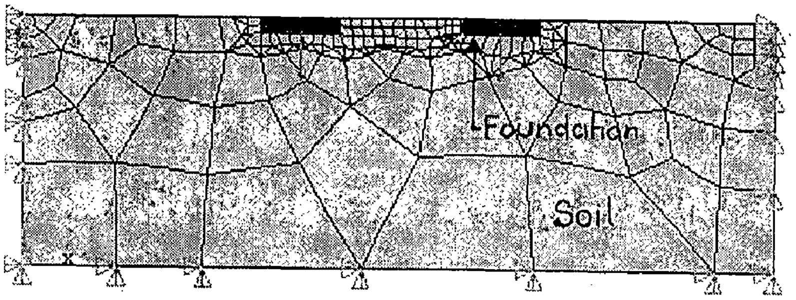

Fig. 5.2 Meshing pattern of the machine foundation- soil system.

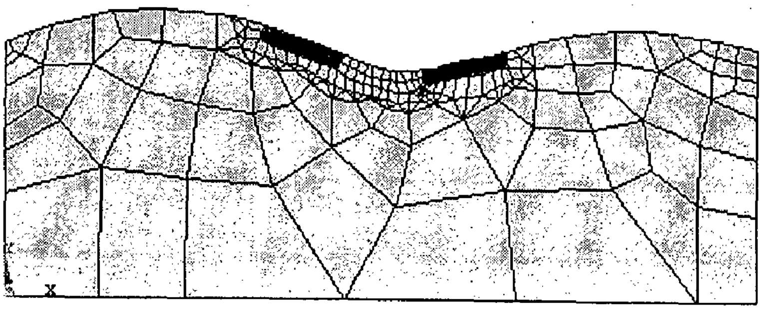

Fig. 5.3 Mode of vibration of machine foundation at on operating frequency (= 7.5 Hz).

Fig 5 Foundation Interaction for spacing (D) = 3B

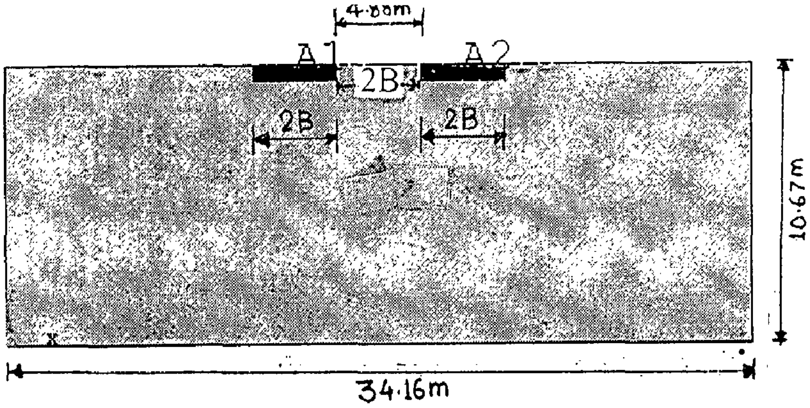

Fig. 6.1 Foundations located at a distance of 2B (= 4.88 m)

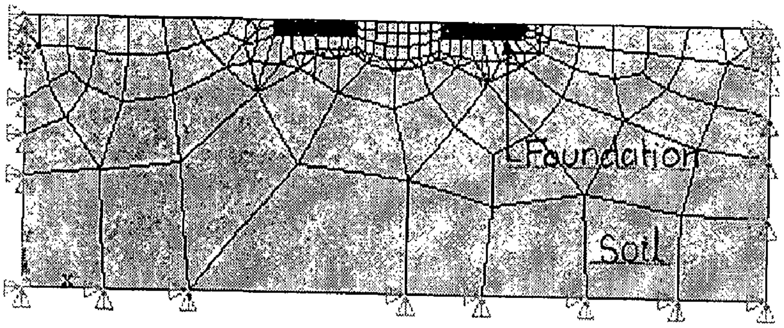

Fig. 6.2 Meshing pattern of the machine foundation-soil system.

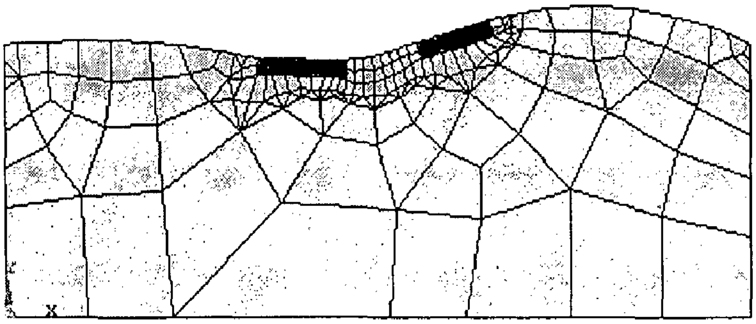

Fig. 6.3 Mode of vibration of machine foundation at an operating frequency (= 7.5Hz).

Fig. 6 Foundation Interaction for spacing (D) = 2B

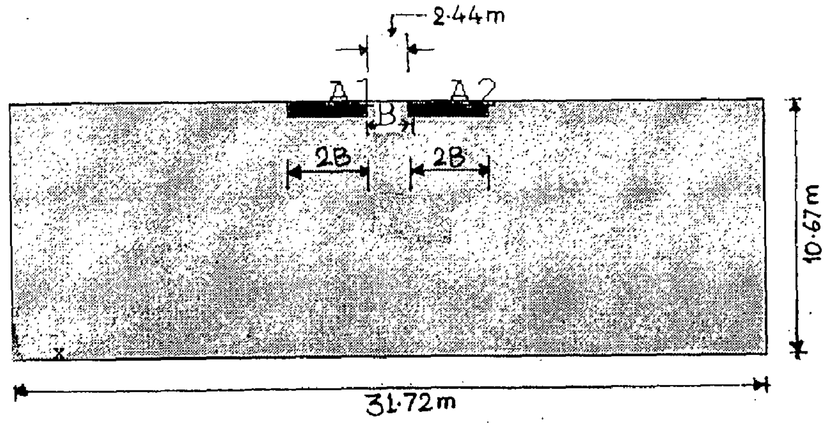

Fig. 7.1 Foundations located at a distance of B (= 2.44 m)

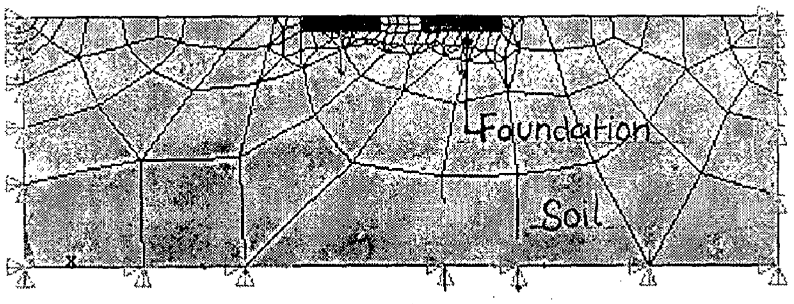

Fig. 7.2 Meshing pattern of the machine foundation – soil system.

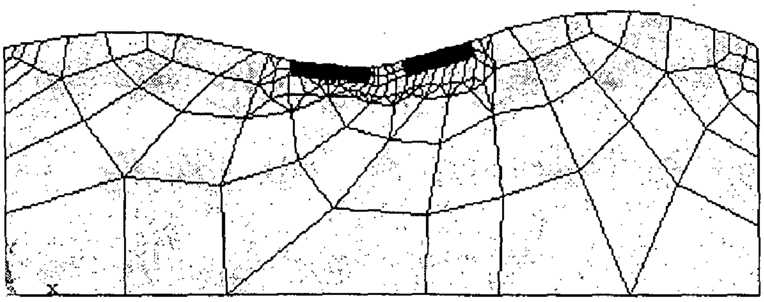

Fig. 7.3 Mode of vibration of machine foundation at an operating frequency (= 7.5 Hz)

Fig. 7 Foundation Interaction for spacing (D) = B

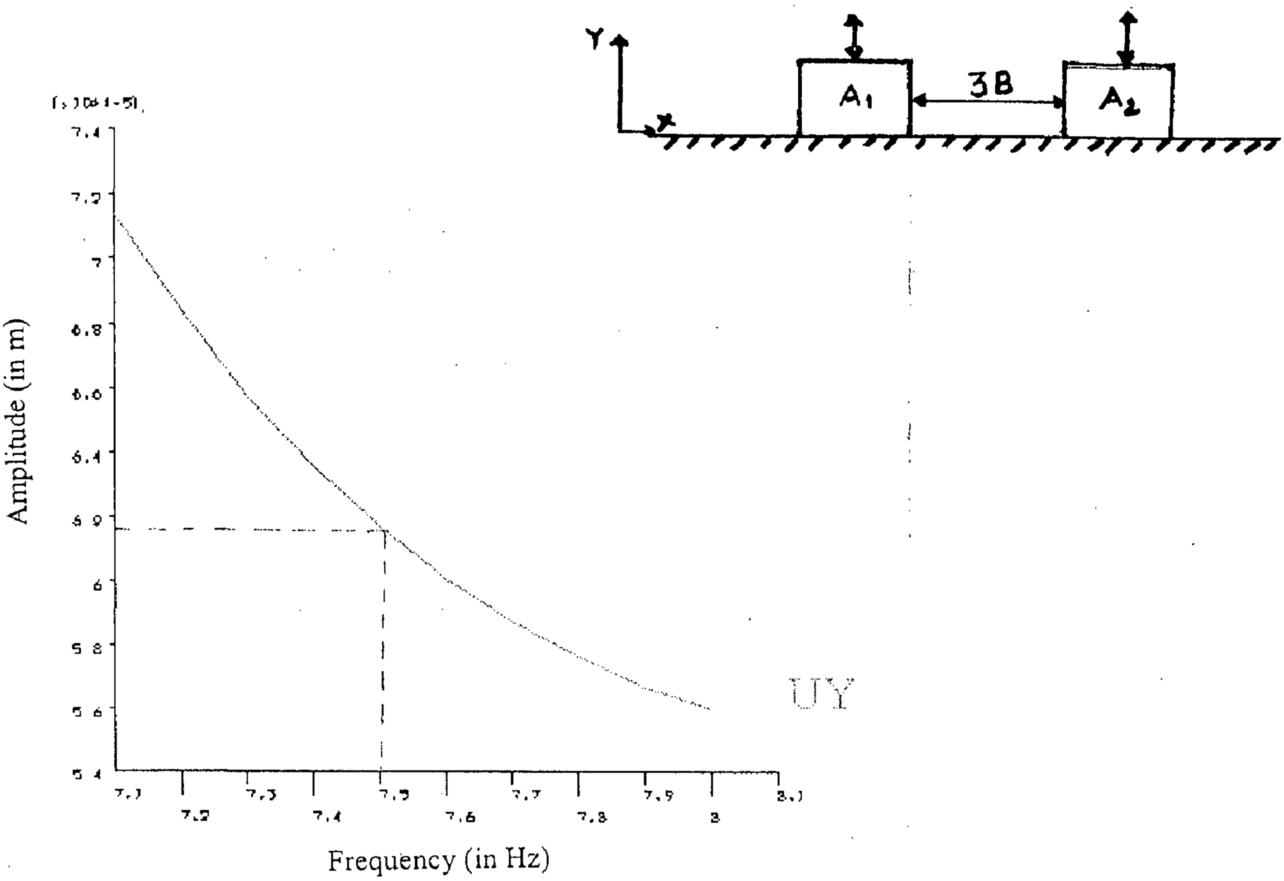

Fig. 8.1 Frequency versus amplitude in Y-direction of foundation A$_1$ (For D = 3B)

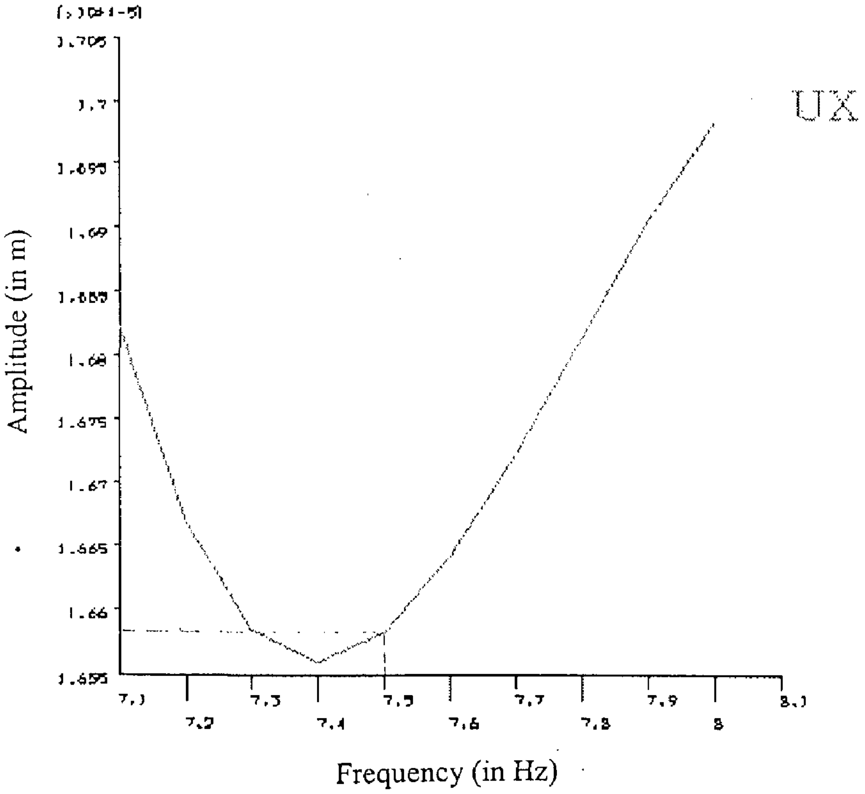

Fig. 8.2 Frequency versus amplitude in X-direction of foundation A$_1$ (For D= 3 B)

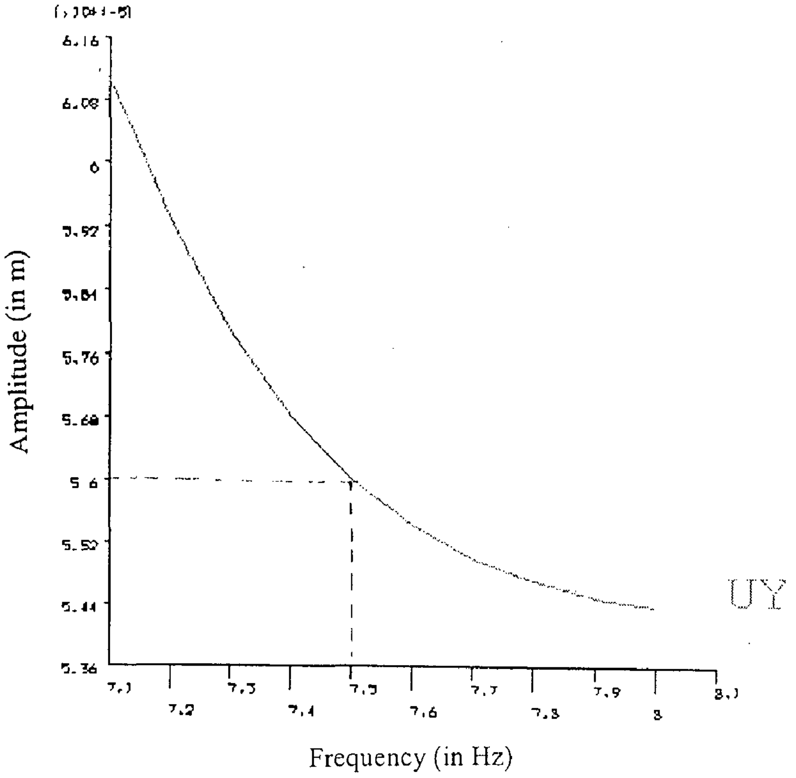

Fig. 8.3 Frequency versus amplitude in Y – direction of foundation A_2 (For D= 3 B)

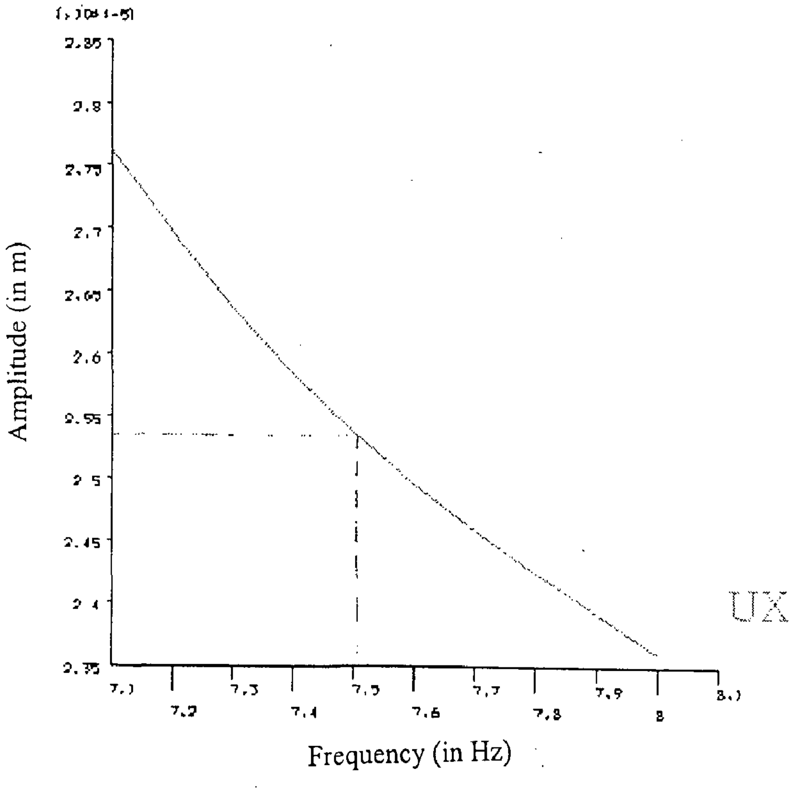

Fig. 8.4 Frequency versus amplitude in X-direction of foundation A_2 (For D = 3B)

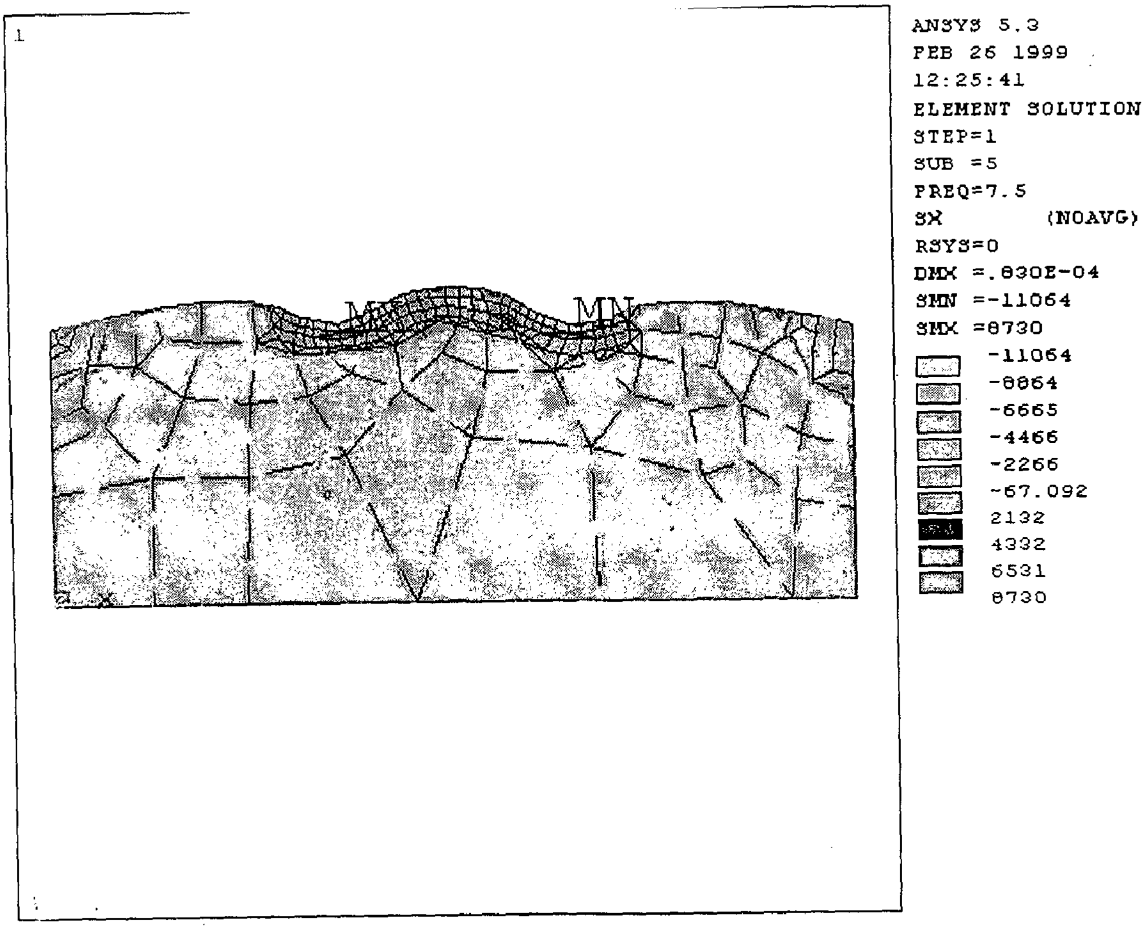

Fig. 9.1 Stress in X – direction (For D = 3B)

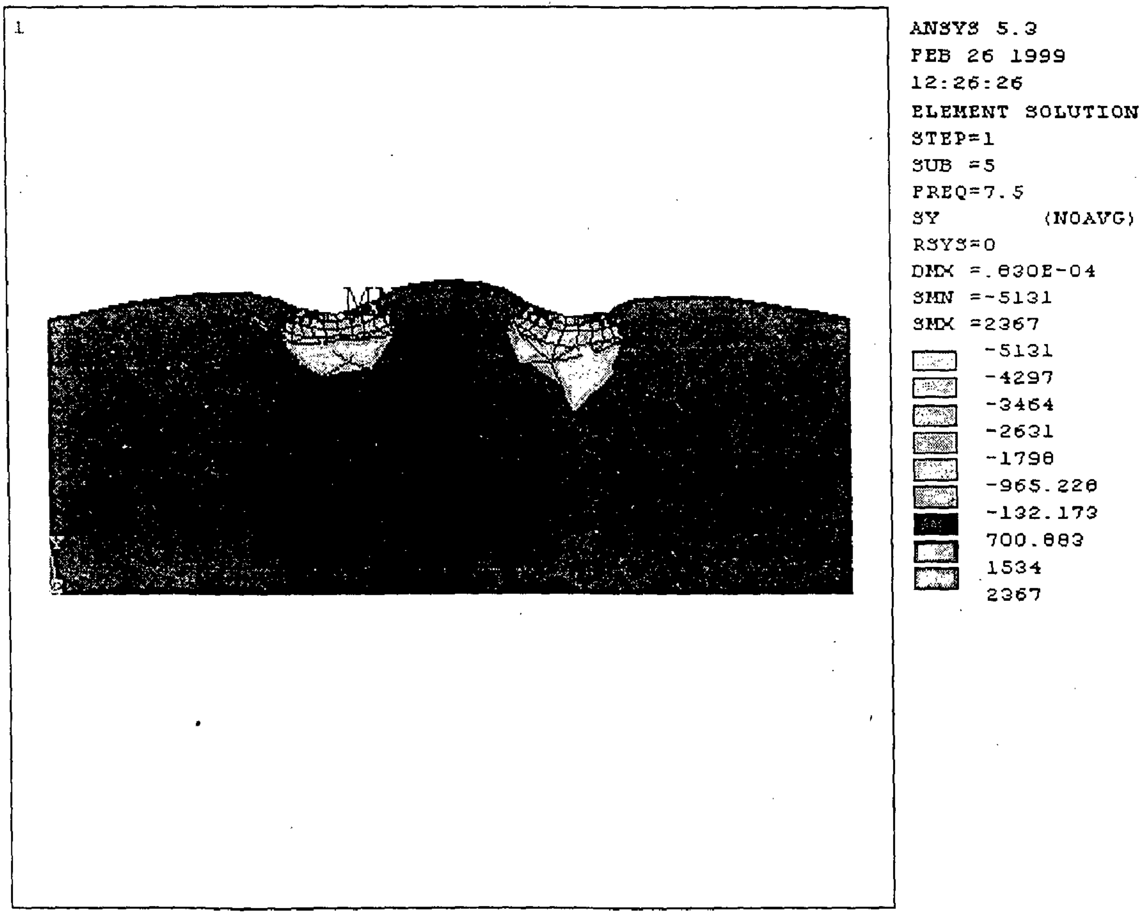

Fig. 9.2 Stress in Y-direction (For D = 3B)

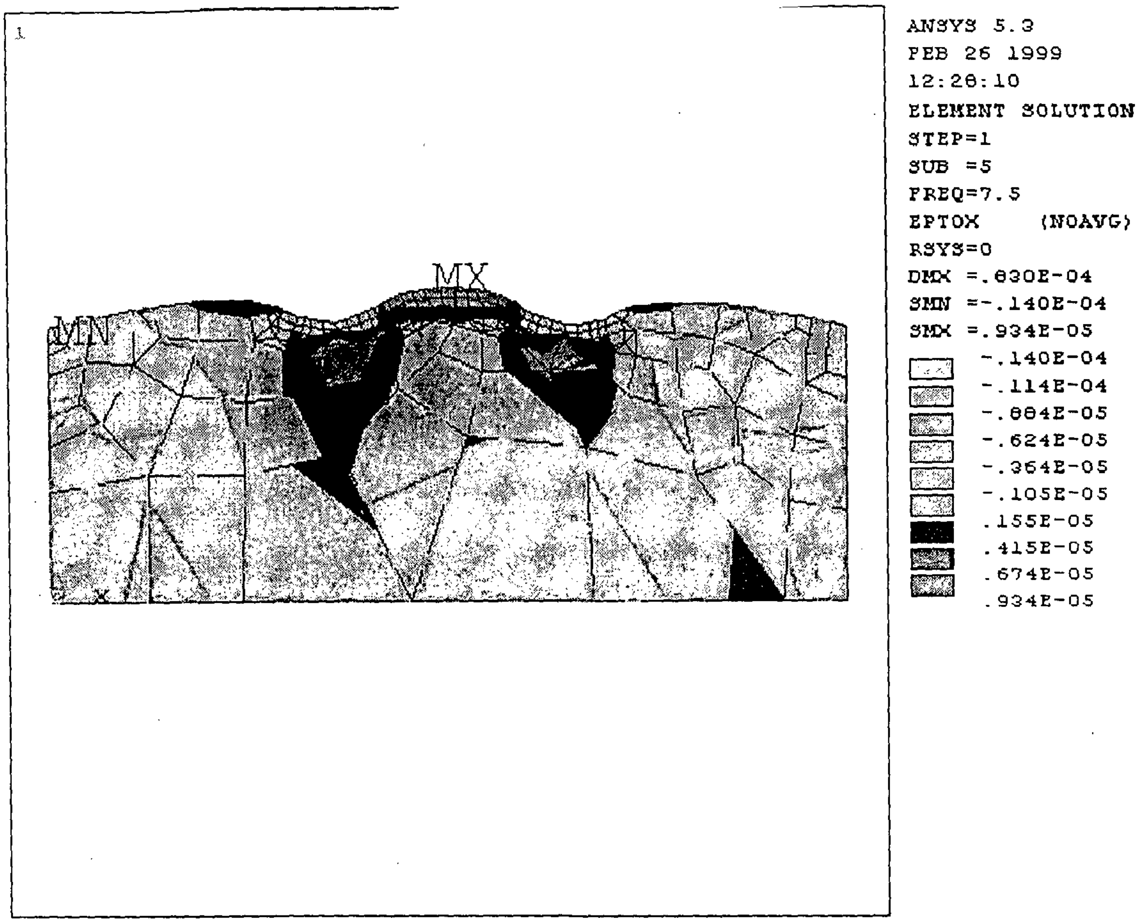

Fig. 10.1 Strain in X – direction (For D = 3B)

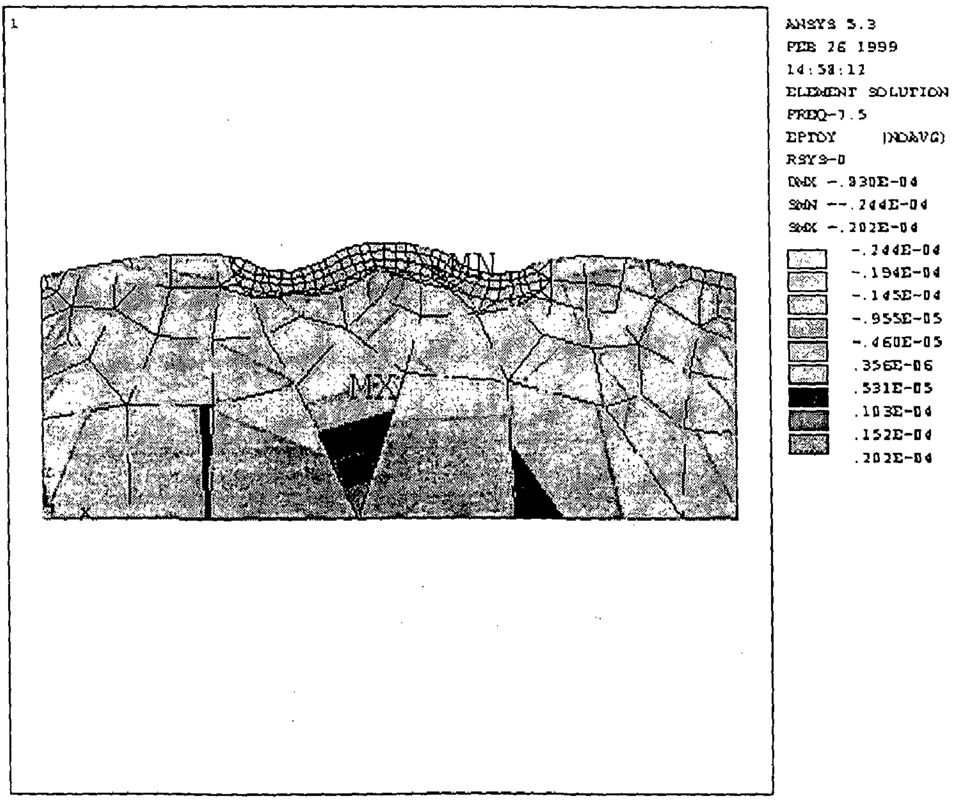

Fig. 10.2 Strain in Y-direction (For D = 3B)

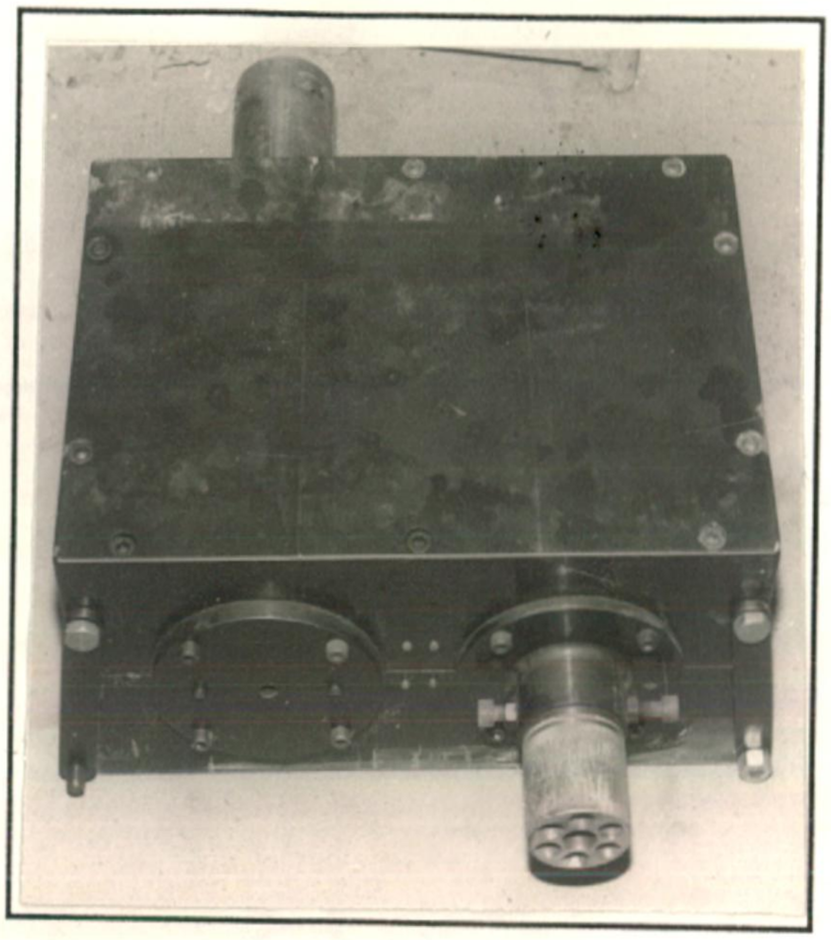

Fig 11.1 'LAZAN' type mechanical oscillator

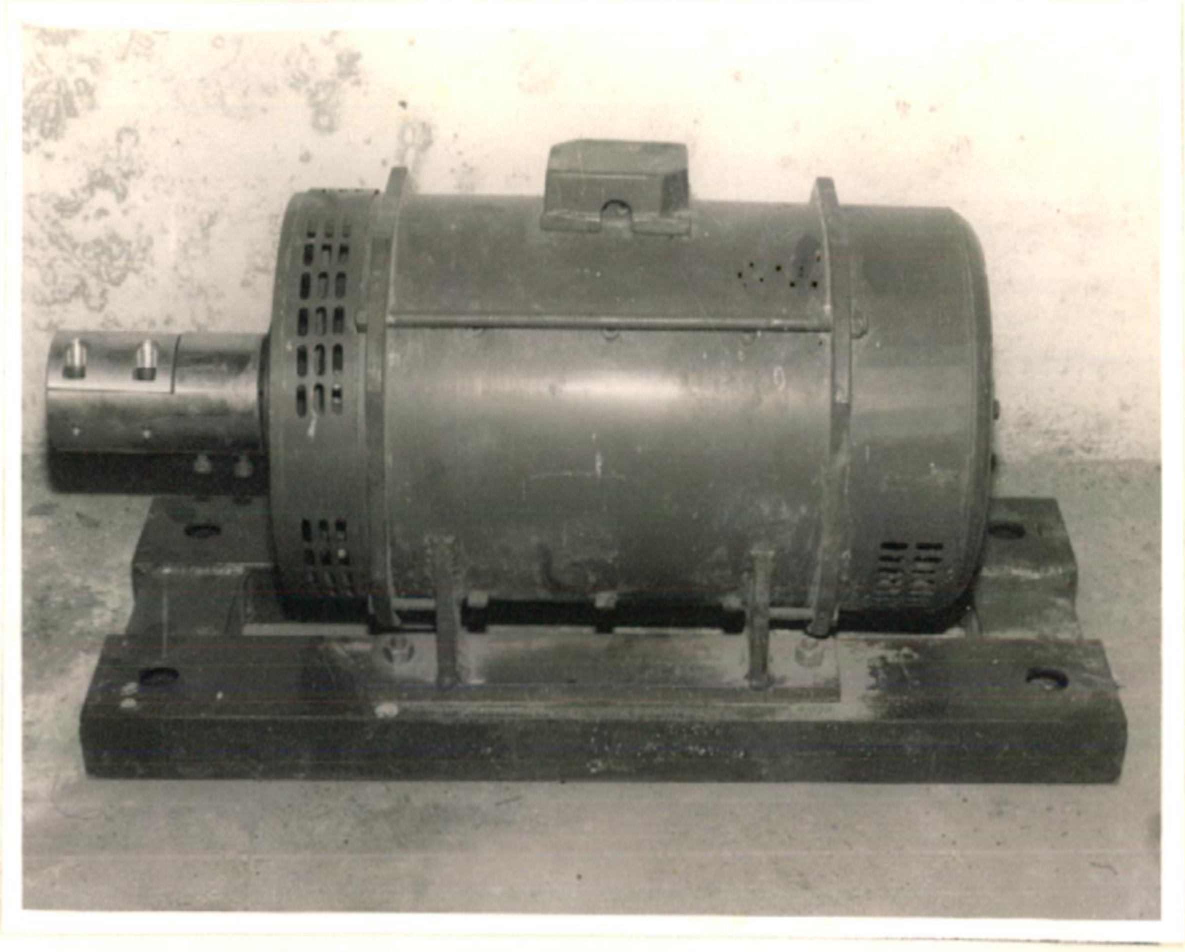

Fig 11.2 Motor

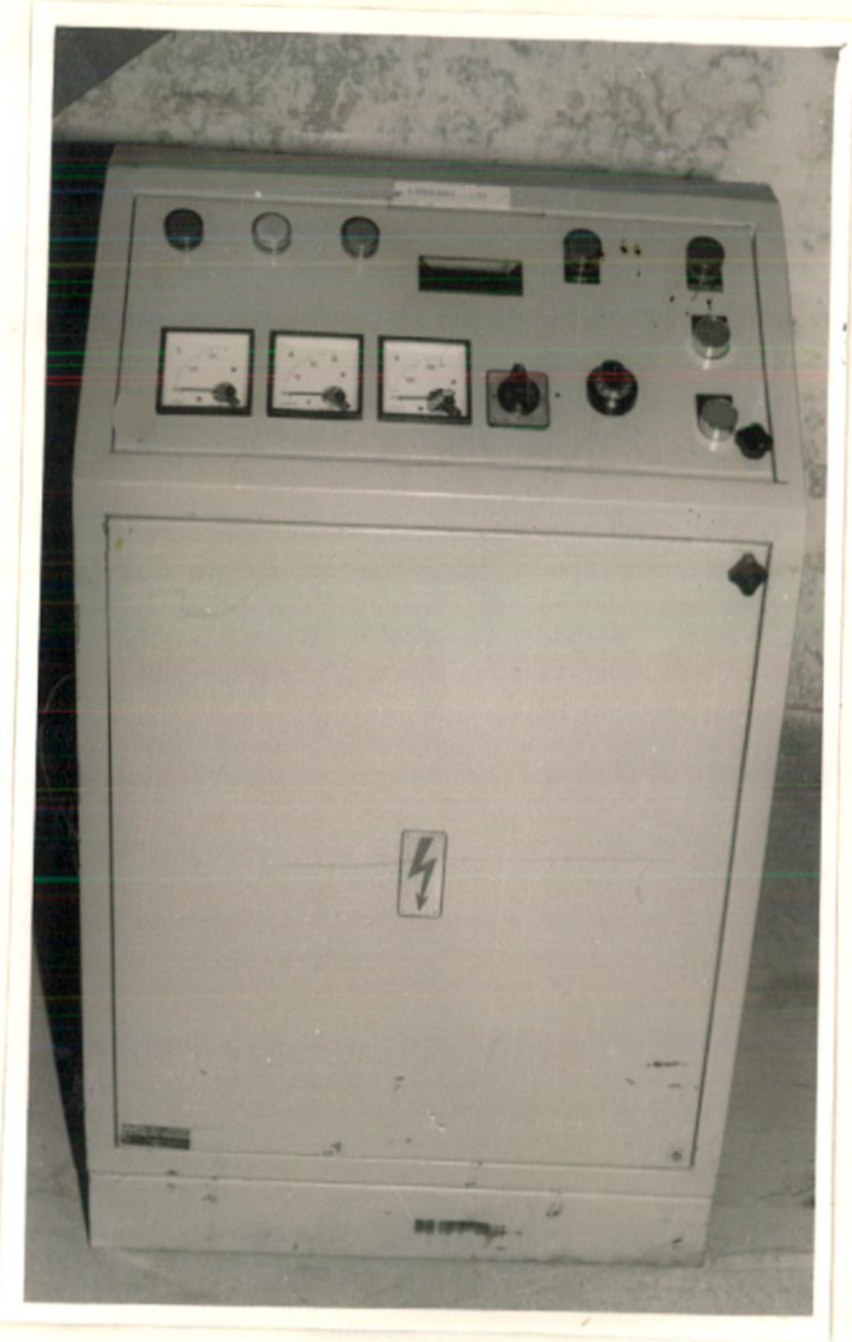

Fig 11.3 Speed Control Unit

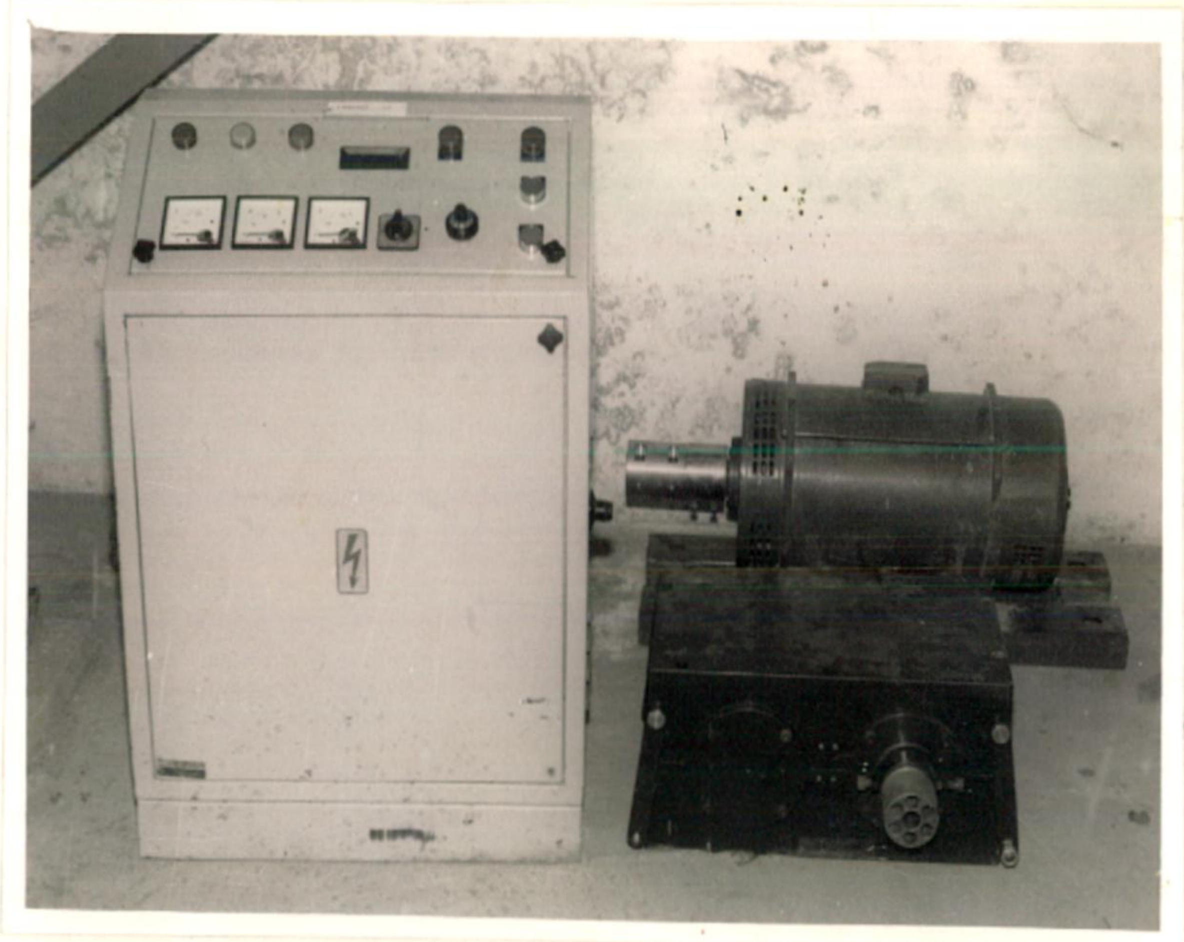

Fig 11.4 An assembly of oscillator, motor and speed control unit.

Fig 11 Equipments used for producing vibration

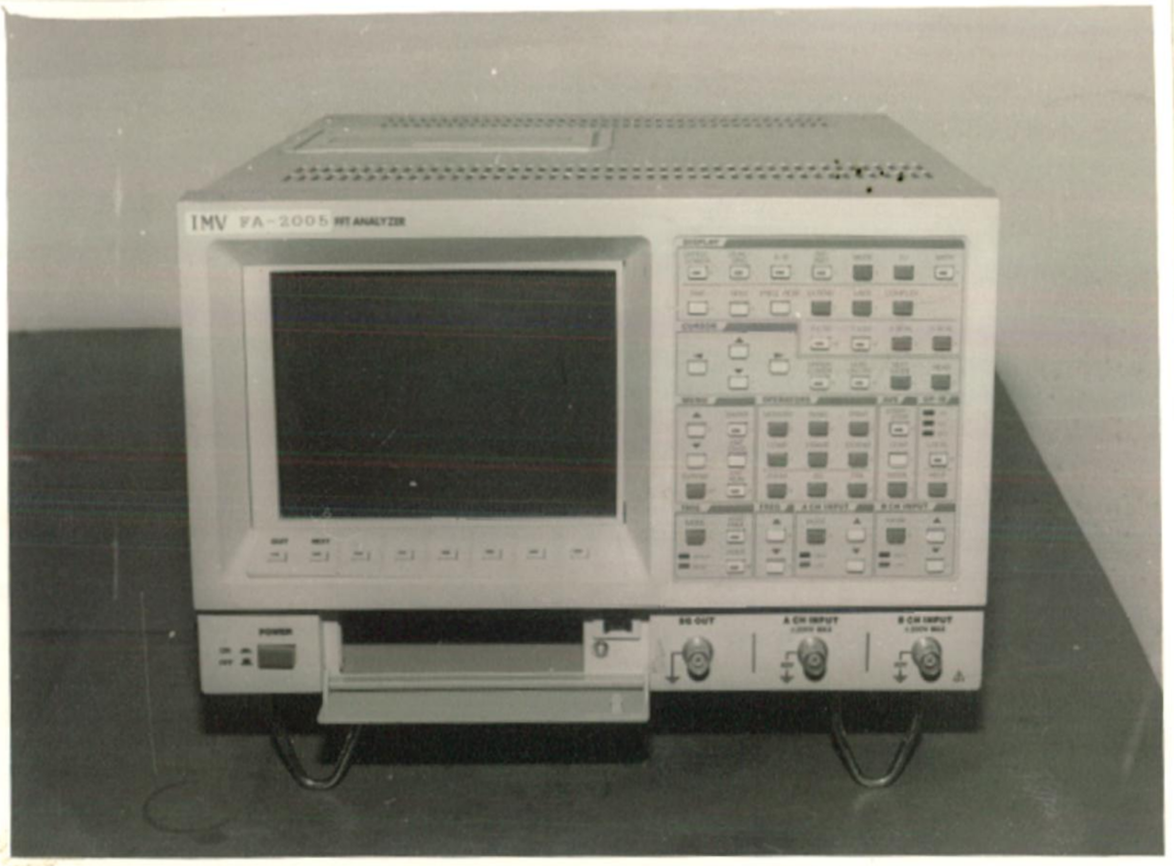

Fig 12.1 Duel Channel FFT Spectrum Analyser

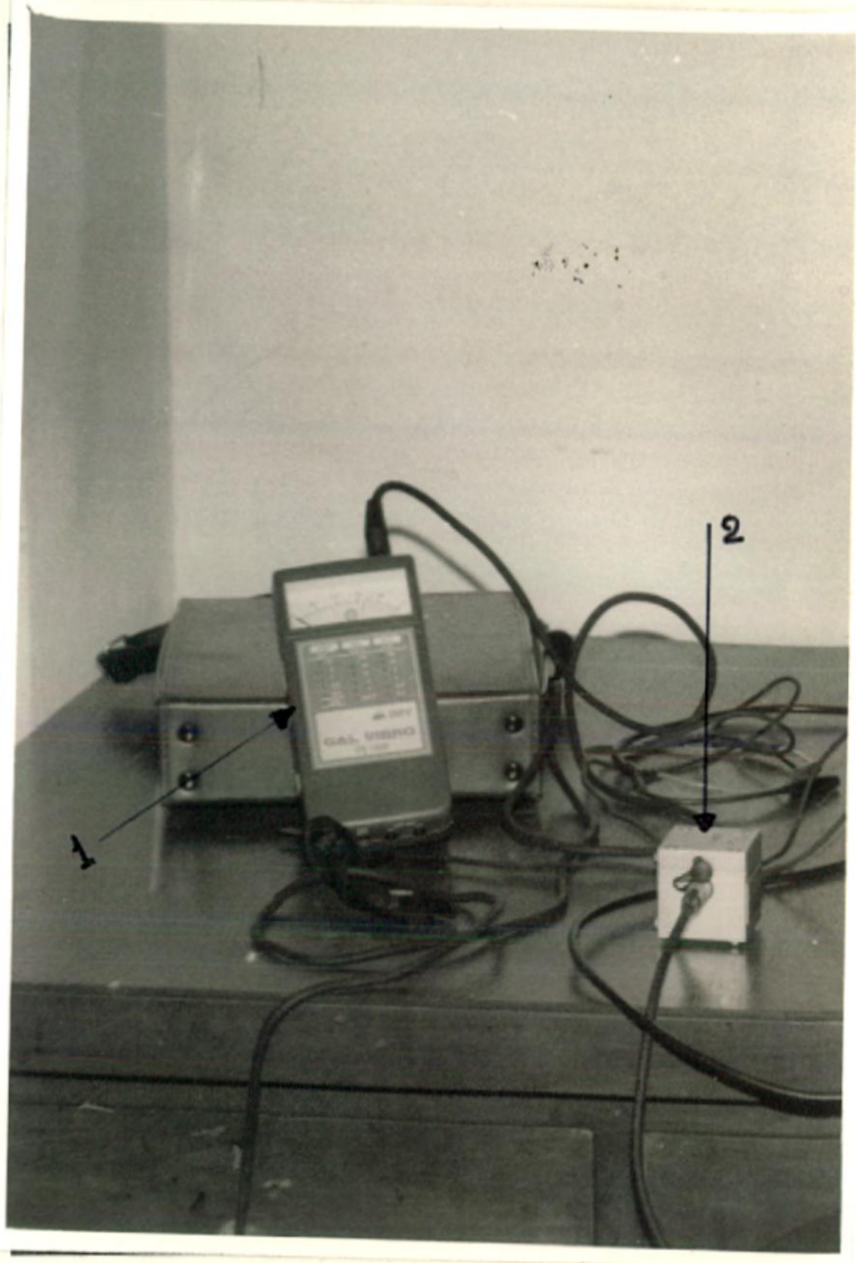

Fig 12.2 Vibration meter with pick-up

1. Vibration meter
2. Pick-up

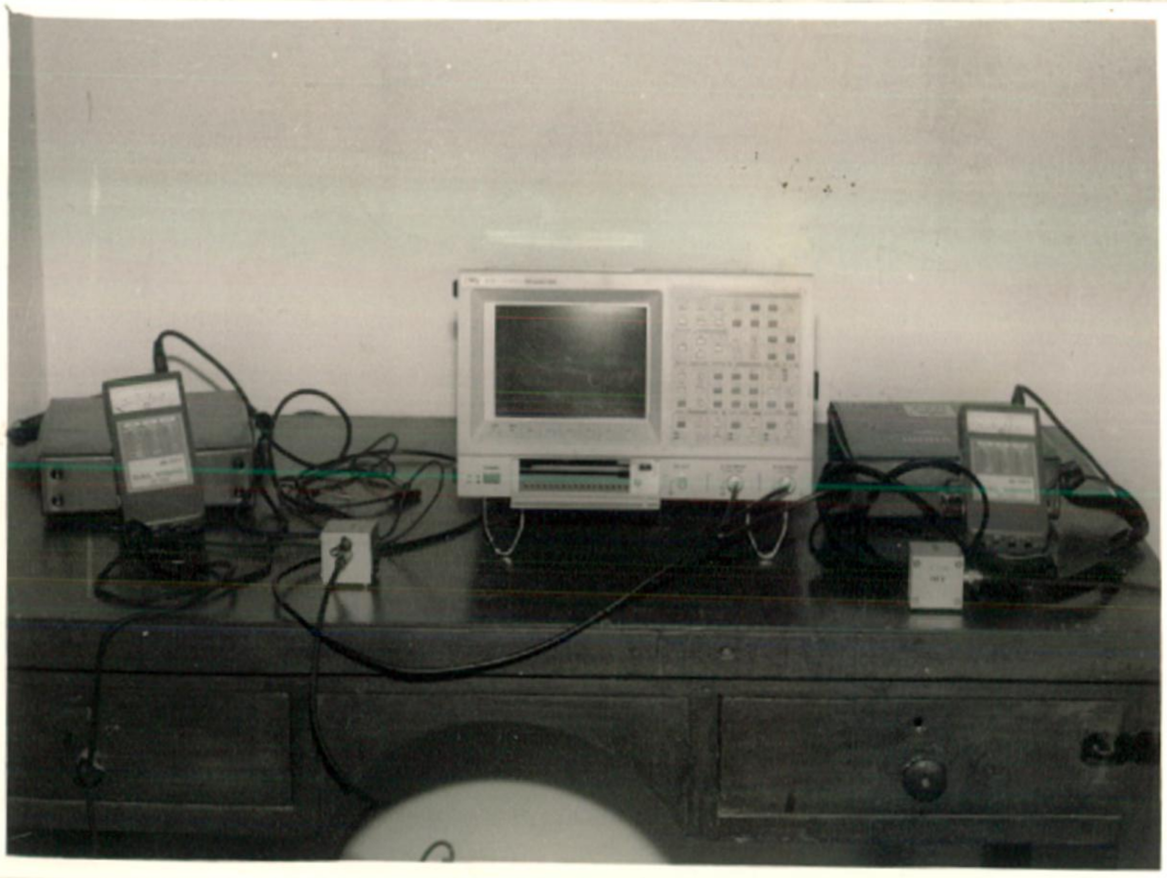

Fig 12.3 An assembly of FFT spectrum analyser, vibration meter and pick-up.

Fig 12 Equipments used for measuring vibration

Fig.13 A concrete block of dimension 2.25m x 0.75m x 0.7m

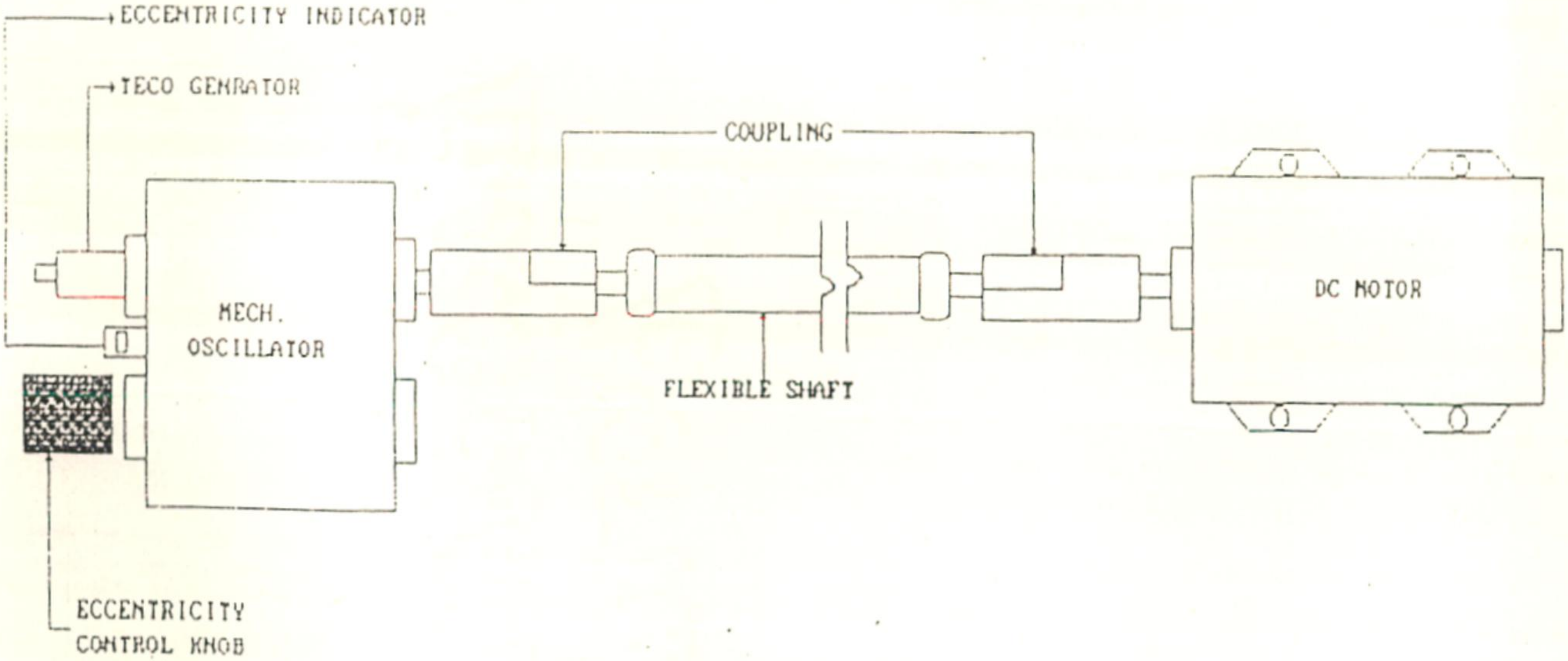

Fig 14 A diagram of an arrangement of motor, oscillator and a tecogenerator

Fig. 15 A concrete block of dimension 2.25m x 0.75 m x 0.7m with base plate

Fig. 16 A concrete block of dimension 2.25m x 0.75m x 0.7m with oscillator.

Fig. 17 Vertical vibration test with the oscillator at the centre of the block

Fig. 18 Horizontal vibration test with the oscillator at the center of the block

Fig. 19 Vertical vibration test with the oscillator at the eccentric point

Fig. 20 Horizontal vibration test with the oscillator at the eccentric point

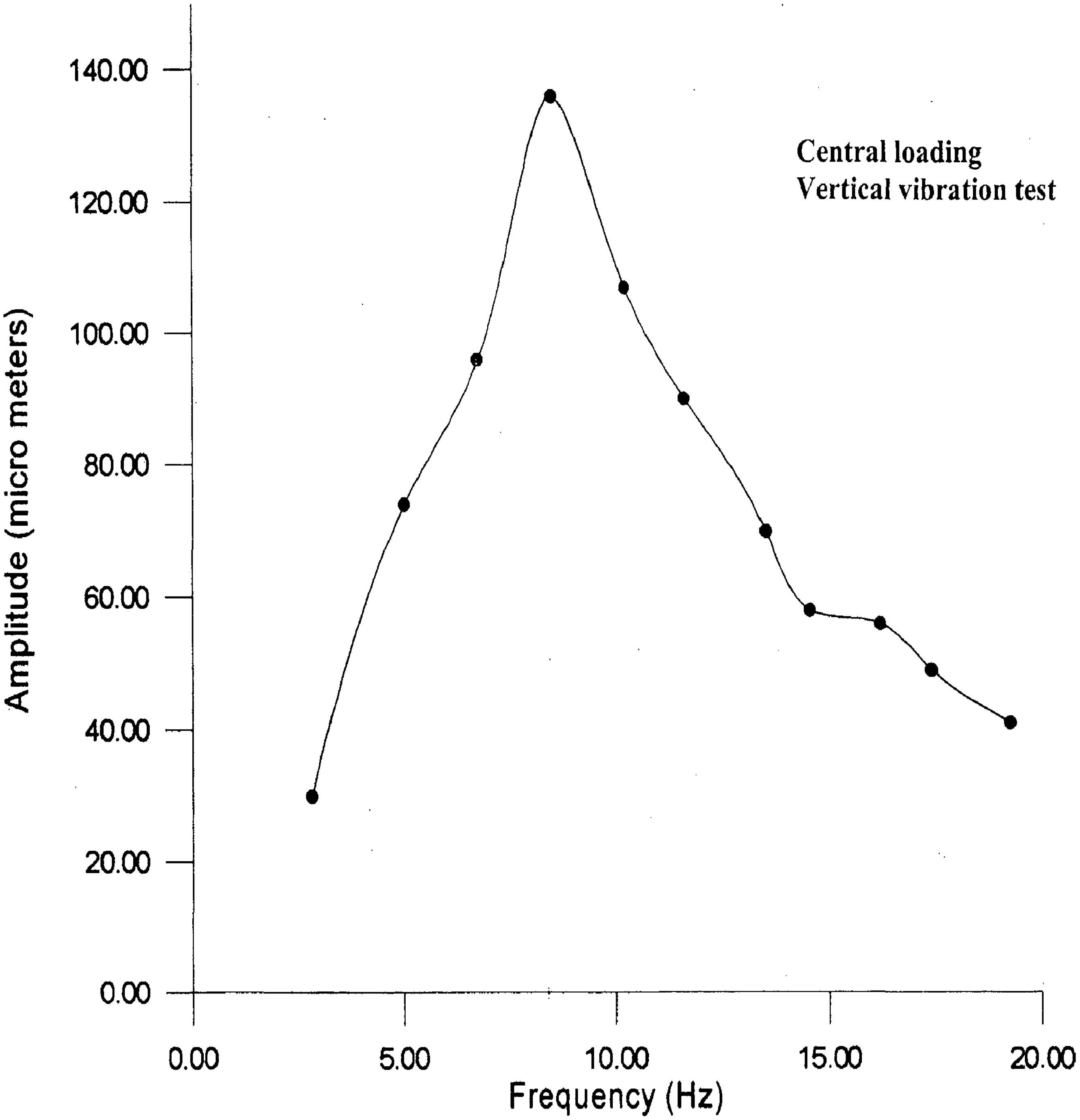

Fig. 21 Frequency versus amplitude for central loading when the oscillator was placed for vertical mode of vibration.

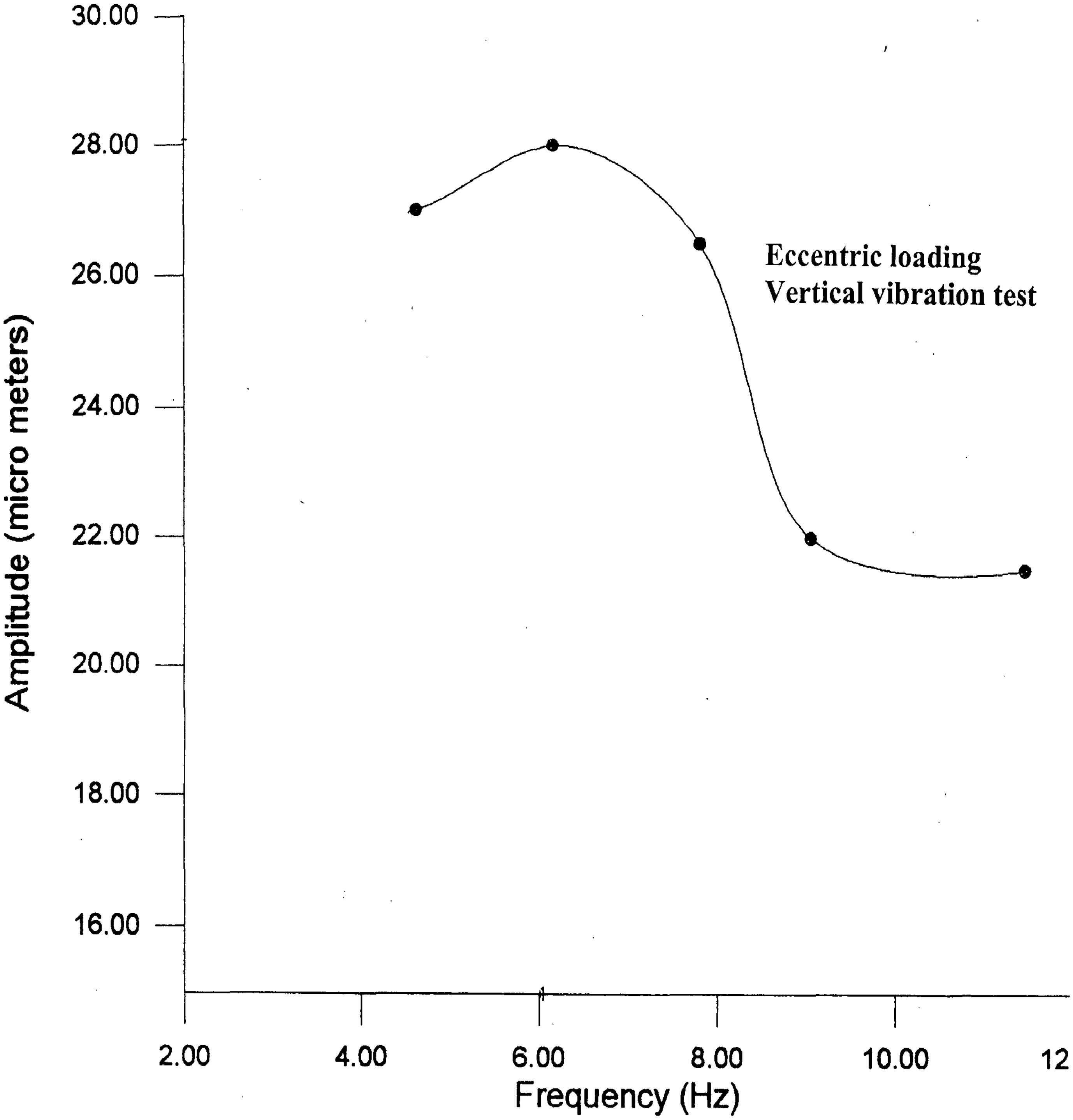

Fig. 22 Frequency versus amplitude for eccentric loading when the oscillator was placed for vertical mode of vibration.

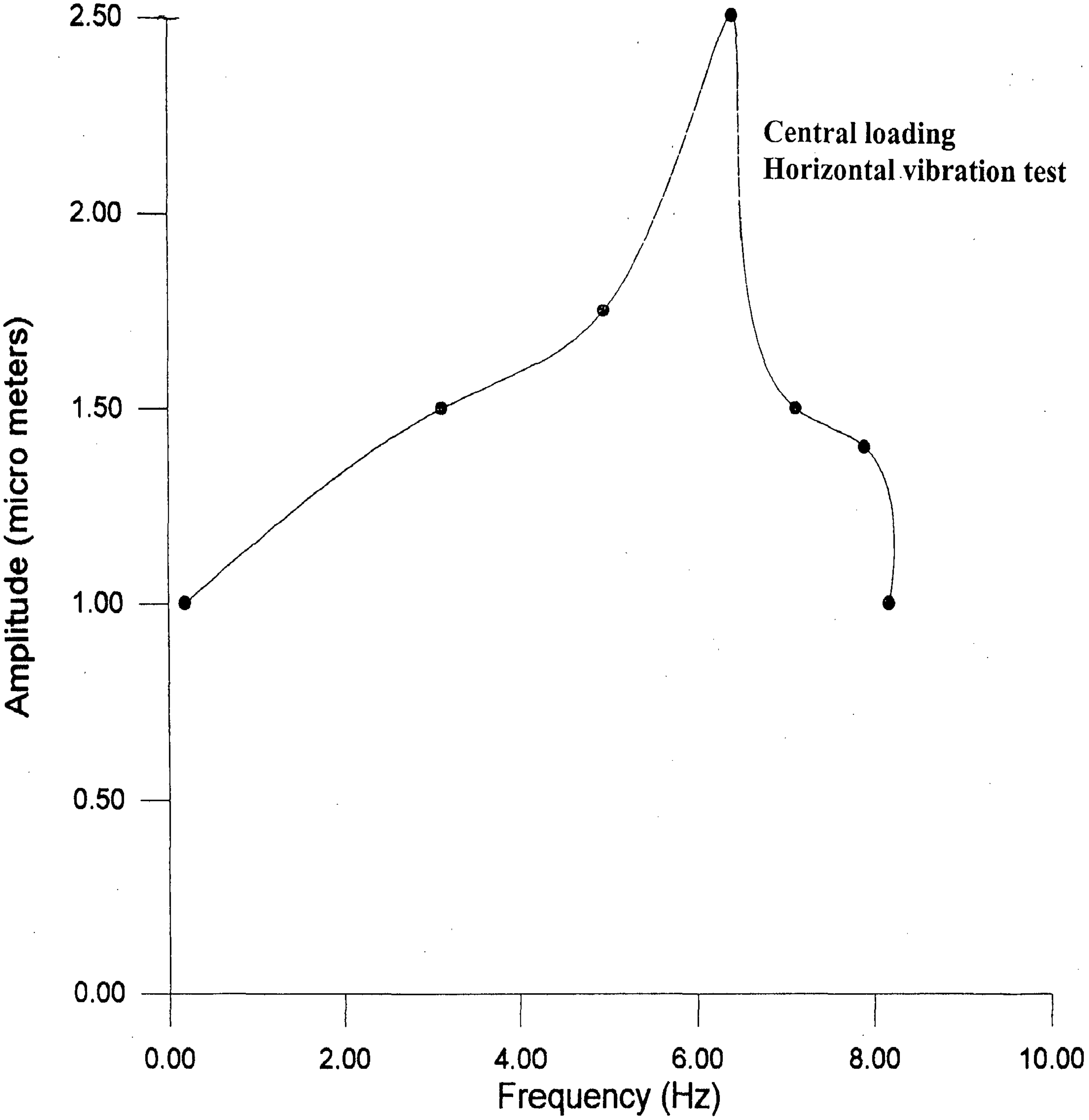

Fig. 23 Frequency versus amplitude for central loading when the oscillator was placed for horizontal mode of vibration.

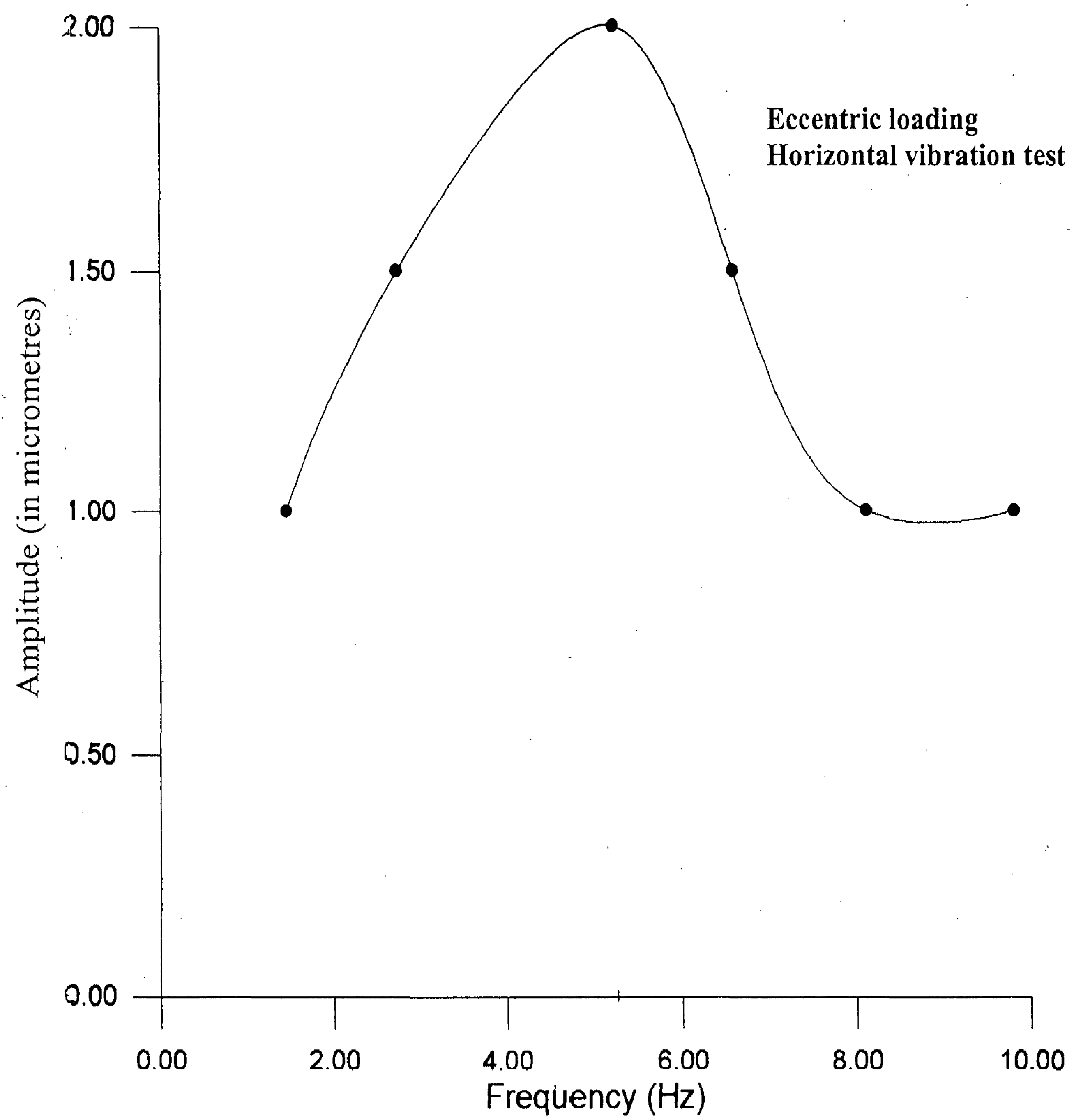

Fig. 24 Frequency versus amplitude for eccentric loading when the oscillator was placed for horizontal mode of vibration.

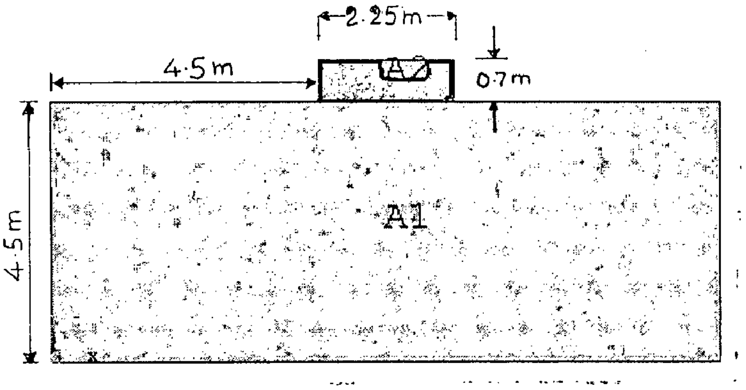

Fig. 25 A model for FEM analysis for experimental block (dimension 2.25m x 0.75m x 0.7m)

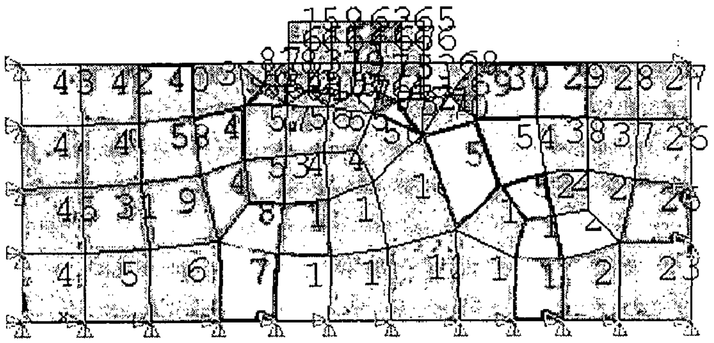

Fig. 26 Element pattern of the machine foundation and soil system with boundary condition.

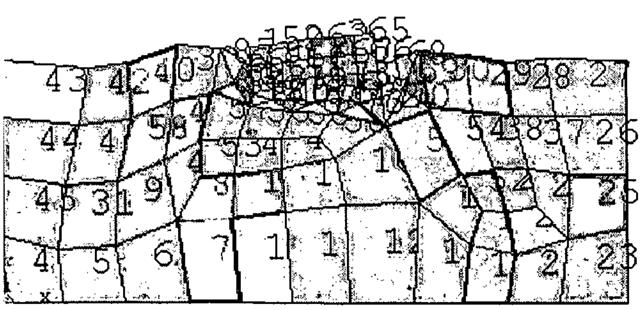

Fig. 27 Vertical mode of vibration of block at frequency (=7.5Hz).

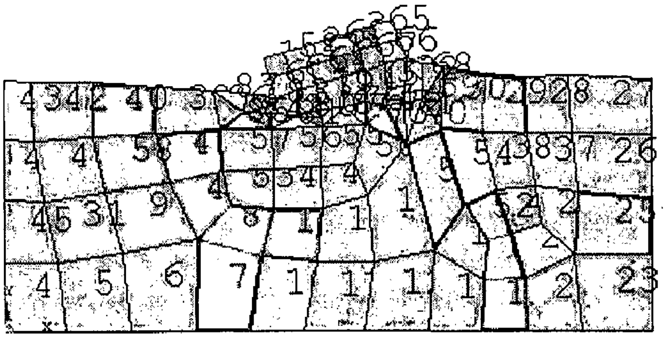

Fig. 28 Horizontal and rocking mode of vibration of block at frequency (=6.426Hz).

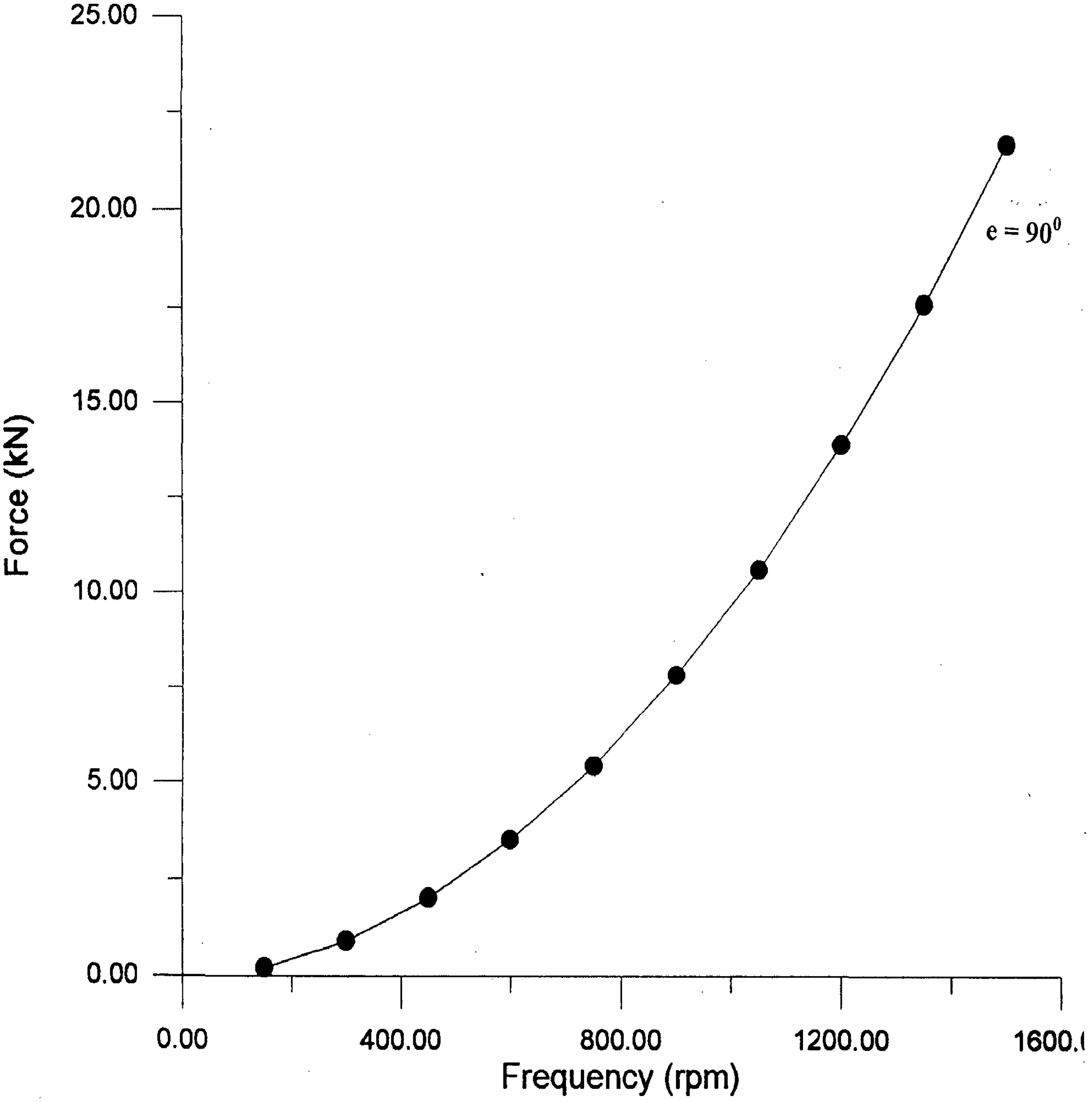

Fig. 29 Operating frequency (Hz) versus force (KN) produced by the oscillator with (e = 90^0)

APPENDIX

TABLE 1 Structural Element types

Category	Shape or Characteristic	Element Name(s)
Spars	General	LINK1, LINK8
	Bilinear (Cable)	LINK10
Beams	General	BEAM3, BEAM4
	Tapered	BEAM54, BEAM44
	Plastic	BEAM23, BEAM24
Pipes	General	PIPE16, PIPE17, PIPE18
	Immersed	PIPE59
	Plastic	PIPE20, PIPE60
2–D Solids	Quadrilateral	PLANE42, PLANE82
	Triangle	PLANE2
	Hyperelastic	HYPER84, HYPER56, HYPER74
	Viscoelastic	VISCO88
	Large Strain	VISCO106, VISCO108
	Harmonic	PLANE83, PLANE25
	p–Element	PLANE145, PLANE146
3–D Solids	Brick	SOLID45, SOLID95, SOLID73
	Tetrahedron	SOLID92, SOLID72
	Layered	SOLID46
	Anisotropic	SOLID64, SOLID65
	Hyperelastic	HYPER86, HYPER58, HYPER158
	Viscoelastic	VISCO89
	Large Strain	VISCO107
	p–Element	SOLID147, SOLID148
Shells	Quadrilateral	SHELL93, SHELL63, SHELL41, SHELL43, SHELL181
	Axisymmetric	SHELL51, SHELL61
	Layered	SHELL91, SHELL99
	Shear Panel	SHELL28
	p–Element	SHELL150
Contact	Point–to–Surface	CONTAC48, CONTAC49
	Point–to–Point	CONTAC12, CONTAC52
	Rigid Surface	CONTAC26
Coupled–Field	Acoustic	FLUID29, FLUID30, INFIN110, INFIN111
	Piezoelectric	PLANE13, SOLID5, SOLID98
	Thermal–Stress	PLANE13, SOLID5, SOLID98
	Magnetic–Structural	PLANE13, SOLID5, SOLID62, SOLID98
	Fluid–Structural	FLUID38, FLUID79, FLUID80, FLUID81
Specialty	Spring	COMBIN14, COMBIN40, COMBIN39
	Mass	MASS21
	Control Element	COMBIN37
	Surface Effect	SURF19, SURF22
	Pin Joint	COMBIN7
	Linear Actuator	LINK11
	Matrix	MATRIX27, MATRIX50
Explicit Dynamics	Spar	LINK160
	Beam	BEAM161
	Shell	SHELL163
	Solid	SOLID164
	Spring–Damper	COMBI165
	Mass	MASS166
	Cable	LINK167

TABLE 2 DOF constraints available in each discipline

Discipline	Degree of Freedom	ANSYS Label
Structural	Translations Rotations	UX, UY, UZ ROTX, ROTY, ROTZ
Thermal	Temperature	TEMP
Magnetic	Vector Potentials Scalar Potential	AX, AY, AZ MAG
Electric	Voltage	VOLT
Fluid	Velocities Pressure Turbulent Kinetic Energy Turbulent Dissipation Rate	VX, VY, VZ PRES ENKE ENDS

TABLE 3 Commands for DOF constraints

Discipline	Primary Data	Derived Data
Structural	Displacement	Stress, strain, reaction, etc.
Thermal	Temperature	Thermal flux, thermal gradient, etc.
Magnetic	Magnetic Potential	Magnetic flux, current density, etc.
Electric	Electric Scalar Potential	Electric field, flux density, etc.
Fluid	Velocity, Pressure	Pressure gradient, heat flux, etc.

TABLE 4 Primary and derived data for different discipline

Location	Basic Commands	Additional Commands
Nodes	D, DLIST, DDELE	DSYM, DSCALE, DCUM
Keypoints	DK, DKLIST, DKDELE	—
Lines	DL, DLLIST, DLDELE	—
Areas	DA, DALIST, DADELE	—
Transfer	SBCTRAN	DTRAN

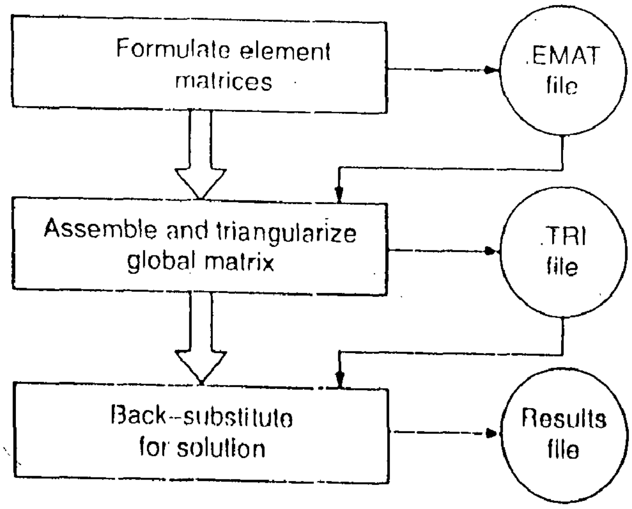

FIGURE 1 Typical steps and files in a frontal solution

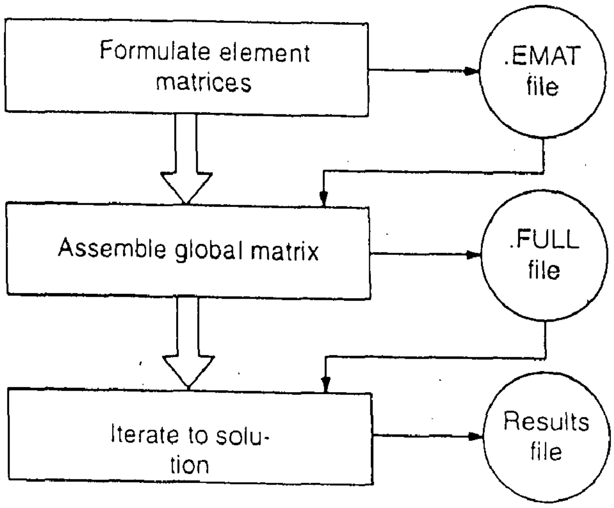

FIGURE 2 Typical steps and files in a JCG solution.

REFERENCES

1. Arya, Suresh C., Michael W.O'Neill, George Pincus " Design of Structures and Foundations for vibrating Machines" Gulf Publishing Book Division, Houston, London, Paris, Tokyo.
2. Barkan, D.D. "Dynamics of Bases and Foundation" McGraw Hill Book Company, Inc. New York, San Francisco, Toronto, London.
3. Bharade, D.S., Dewaikar, D.M. "Effect of Shape on Torsional Vibration of Block Foundations" Indian Geotechnical Journal, Volume 19, No. 4 Oct. 1989.
4. Biswas, Partha Pratim, "Dynamic Response of Soil Mass on Two Layered Non-Cohesive Soil" Indian Geotechnical Journal Volume 25, No.2, April 1995.
5. Borja, Ronaldo I.&Wu, Wen-Hwa "Vibration of Foundation on Incompressible Soils with no Elastic Region," GT, ASCE, Sept. 1994.
6. Desai, Chandrakant S. & Abel, John F. "Introduction to the Finite Element Method."Von Nostrand Reinhold Company, New York.
7. Dunham Clarence W. "Foundations of Structures".
8. Humar J.L. " Dynamics of Structure", Prentice Hall International Series in Civil Engg. & Engg. Mech.
9. Irish K. and Walker, W.P."Foundations for Reciprocating Machines", Published by Concrete Publications Limited, 60 Buckingham Gate, London, S.W.1.
10. Major Alexander, "Dynamics in Civil Engineering (Analysis and Design)," Published by Akademiai Kiado, Budapest and Collets Holding Limited, London, 1980, Vol. 1,2,3,4.
11. Murthy V. N.S., "A Text Book of Soil Mechanics and Foundation Engg.," Third

Edition SAITECH.

12. Richart F.E. and Others "Vibrations of Soils and Foundations", Prentice – Hall, INC, Englewood Cliffs, N.J.

13. Sridharan, A., Baidya, D.K., Raju, D.M.,(i) "Foundation Response to Rocking Vibrations", (ii) "Prediction of Displacement Amplitude of foundation at any frequency" Indian Geotechnical Journal, Volume 22, No.3, July 1992.

14. Srinivasulu P., Vaidyanathan C.V., "Handbook of Machine Foundation", Published by Tata McGraw Hill Publishing Company Ltd., New Delhi.

15. Tham, L.G., Qian, J. & Cheung, Y.K., "Dynamic response of a group of flexible foundations to incident seismic waves", "Soil Dynamics and Earthquake Engineering", Volume 17, Number 2, 1998.

16. IS Code of Practice for Design and Construction of Machine Foundations, IS: 2974(Part 1): 1982. Part 1 : Foundation for Reciprocating type Machines.

17. IS: 2974 (Part II) : 1980 , " Foundation for Impact Type Machines (Hammer Foundation)".

18. IS: 2974(Part III) : 1975, "Foundation for Rotary Type Machine (Medium & High Frequency)

19. IS : 5249: 1969 Methods of test for determination of in situ dynamic properties of soil.

20. The ANSYS manual, Basic Analysis Procedures Guide, Release 5.3, ANSYS, Inc. 201, Johnson Road, Houston, PA 15342-1300.

21. The ANSYS manual, Modelling and Meshing Guide, Release 5.3, ANSYS, Inc. 201, Johnson Road, Houston, PA 15342-1300.

22. The ANSYS manual, Structural Analysis Guide, Release 5.3, ANSYS, Inc. 201, Johnson Road, Houston, PA 15342-1300.