Volume 9, Issue 11, November– 2024 International Journal of Innovative Science and Research Technology
ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

Finite Element Analysis and Optimization of Machine Foundations

Under the guidance of

Dr. B. Siva Konda Reddy[1]

[1]Associate Professor Department of Civil Engineering Jawaharlal Nehru Technological University, Hyderabad

A Thesis on

A Dissertation Submitted in the Partial Fulfillment of the Requirement for the Award of the Degree of

Master of Technology

In

Structural Engineering

Sravya Kasukurthi
17011P0115

Jawaharlal Nehru Technological University College of Engineering Hyderabad (Autonomous)
Kukatpally, Hyderabad-500085, Telangana, India

Publication Date: 2025/06/28

How to Cite: Dr. B. Siva Konda Reddy; (2025) Finite Element Analysis and Optimization of Machine Foundationsions. *International Journal of Innovative Science and Research Technology*, 9(11), 3755-3809. https://doi.org/10.38124/ijisrt/24nov077

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CERTIFICATE

This is to certify that this thesis or dissertation entitled " **Finite Element Analysis and Optimization of Machine Foundations**" is being submitted by **Sravya Kasukurthi (17011P0115),** in partial fulfilment for the award of **MASTER OF TECHNOLOGY** in **STRUCTURAL ENGINEERING**, to the department of civil engineering, Jawaharlal Nehru technological university, college of engineering, Hyderabad is a record of bonafide work carried out by him under our guidance and supervision.

The results embodied in this thesis have not been submitted to any other universities or institutes for the award of any degree or diploma.

PROJECT GUIDE
Dr B Siva Konda Reddy
Associate Professor
Department of Civil Engineering
JNTUH College of Engineering
Hyderabad- 500085

HEAD OF DEPARTMENT
Dr B Dean Kumar
Head & Professor
Department of Civil Engineering
JNTUH College of Engineering
Hyderabad- 500085

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

DECLARATION

I hereby declare that the report of P.G Project work entitled, "**FINITE ELEMENT ANALYSIS AND OPTIMIZATION OF MACHINE FOUNDATIONS**", which is being

submitted to the Jawaharlal Nehru Technology University, Hyderabad, in partial fulfilment of the requirements for the award of the **DEGREE OF MASTER OF TECHNOLOGY** in **STRUCTURAL ENGINEERING**, Department of civil Engineering, is a bonafide report of the work carried out by me. The material contained in this report has not been submitted to any other universities or institutes for the award of any degree or diploma.

KASUKURTHI SRAVYA
H.T. NO: 17011P0115
Department of Civil engineering JNTUH College of Engineering
Hyderabad- 500085

Place:

Date:

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

ACKNOWLEDGEMENT

I take this opportunity to express my gratitude to the people who helped in the successful completion of this project. The success of this project depends largely on the encouragement and guidelines of the following people.

I wish to express my sincere gratitude I would like to report my profound thanks to my honourable guide Dr B SIVA KONDA REDDY, Associate Professor of Civil Engineering, JNTUH College of Engineering, Hyderabad for his valuable guidance, suggestion and good support at every stage of my dissertation.

I am extremely grateful to Dr B. DEAN KUMAR, Professor and Head, Department of Civil Engineering, JNTUH College of Engineering, Hyderabad for his encouragement and concern shown in providing the laboratory and administrable facility during the project work. I also thank all the staff members of the civil engineering department faculty and my friends for their support and encouragement throughout the project completion.

SRAVYA KASUKURTHI

ABSTRACT

Constantly facing the increased requirements for machine efficiency and reliability, designers have to pay attention to every detail including machine foundations. The turbine foundations play an important role in large-scale power stations. Obviously. it has to be designed safely to ensure safety and the normal operation of the whole electricity generating system. Specifically, it is necessary to reduce the vibration level of the foundation under working loads caused by the eccentricity of the rotating parts at a reasonable construction cost. A turbine foundation is typically a frame structure with beams and columns made of reinforced concrete. The external loads acting on the foundation include dead weight and harmonic excitations caused by the eccentricity of the rotating parts. This paper presents the Static and Dynamic analysis of machine foundations and a procedure for optimization of reinforced concrete columns of the spring-mounted machine foundation subjected to dynamic loads. Codes used for determining the unbalanced forces from machines on to foundations are IS 2974(1992)-part 1, IS 2974(1992)- part 3. The software used is 'ANSYS WORKBENCH 2022 R1' The results obtained from the finite element analysis of the constructed model are used to obtain optimal design variables as the dimensions of columns.

Keywords: *Static and Dynamic Analysis, Optimisation, Spring Mounted Machine Foundation.*

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

LIST OF FIGURES

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

LIST OF TABLES

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

TABLE OF CONTENTS

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER ONE
INTRODUCTION

A. Introduction:

Machine foundations are a particular type of foundations, which are built underneath the machines with frequent vibrations or have some mechanical motion that results in a lot of forces being generated. Therefore, it became prominent in the considerations of designing machine foundations to ensure performance improvement and simultaneously safety concerns.

➢ *Necessity of Machine Foundations:*

The design of machine foundations is generally more complicated than that of a foundation under static loads. In this type of foundation, the engineer must con-sider dynamic loads caused by the machine in addition to the static loads. These dynamic forces are transferred to the foundation system supporting the machine. Therefore, it is required to have good knowledge about the load transmission method and the dynamic behaviour of the system and the soil beneath the foundation.

The analysis and design of structures and foundations subjected to dynamical loading are considered to be more complex due to its interaction of structural engineering, Geo-tech and the theory of vibration. Thus, a detailed dynamic or vibrational analysis had become necessary, also it was realised that a detailed investigation of soil is also crucial as the elastic properties of soil influence the design of the foundation.

The cost of a foundation is a mere fraction of that of the equipment, If constructed inadequately may result in failures and shutdowns, in return these failure costs may exceed many times the cost of a properly designed foundation. The machine foundation system can be modelled as a 2D structure or 3D structure. Some assumptions are made which has to be followed during the modelling of a foundation system. They are as follows:

- The model has to be harmonious to the Prototype structure with a considerable degree of accuracy.
- The model has to be such that it can be analysed with the available mathematical tools.
- The influence of each assumption should be quantitatively known with regard to the response of the foundation.

B. Design Philosophy:

Machine foundation system, comprises of the machine, rested over by a foundation supported by soil subjected to dynamic loads 1) generated by the machine, 2) applied externally, or 3) caused by external sources and transmitted through soil. A typical system is shown in Figure 1

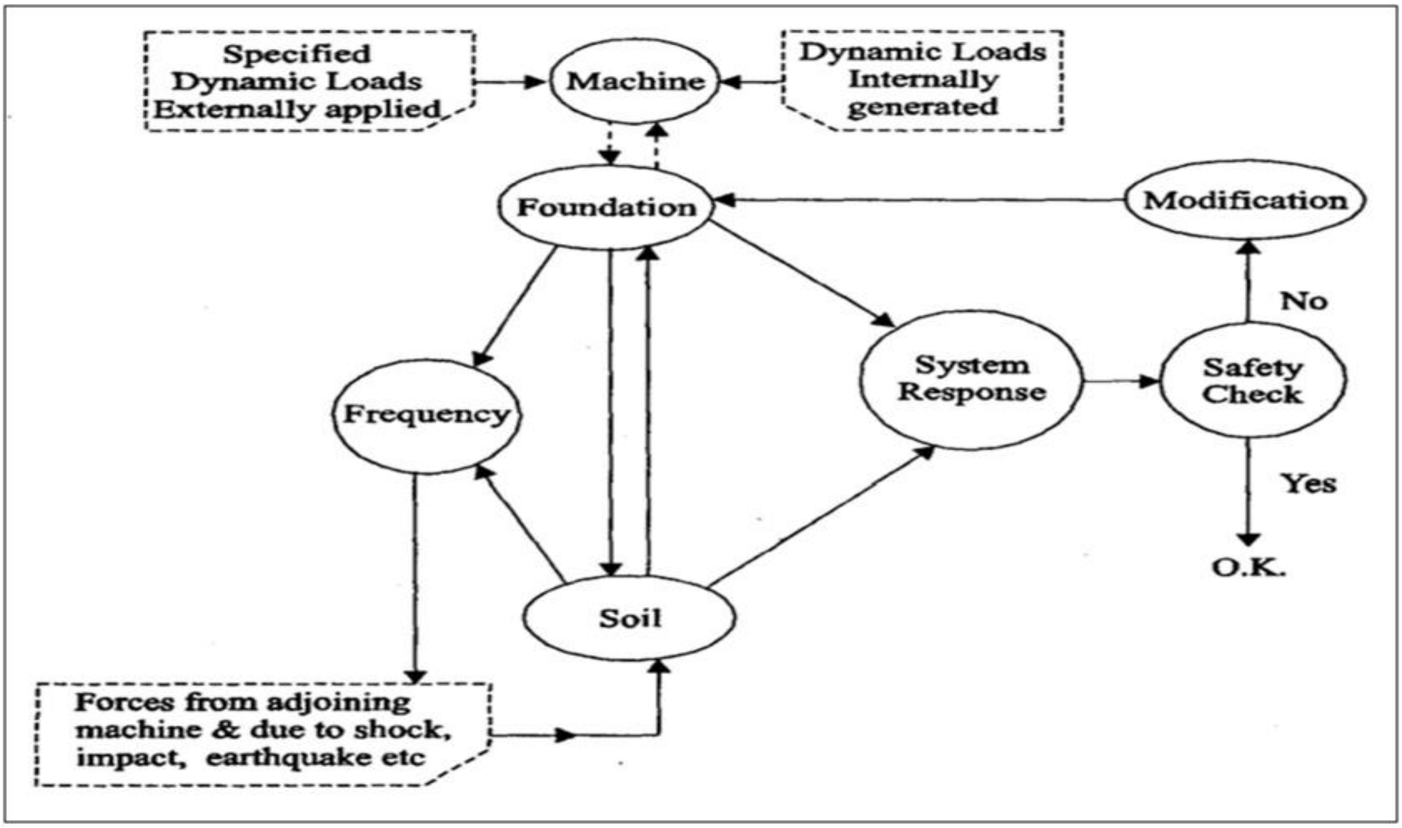

Fig 1 A Flow Chart for the Design of the Machine Foundation

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

➢ *Irrespective of the source of Dynamic load, the Basic Principles of Designing a Machine Foundation are:*

- The dynamic forces caused by machines are transferred to the soil through a medium called foundation in a way that, all types of harmful effects are removed and the amplitudes of vibrations of the machine and foundation are within the appreciable limits,
- Foundation has to be structurally safe to resist both static and dynamic forces generated by the machine. To achieve these objectives, every system (foundation system) has to be analysed dynamically and for strength design.

➢ *Machine Foundation System:*

As discussed earlier, a typical machine foundation system is a machine supported by a particular type of foundation which is in turn rested over the soil. A typical machine foundation system is as shown in Figure 2

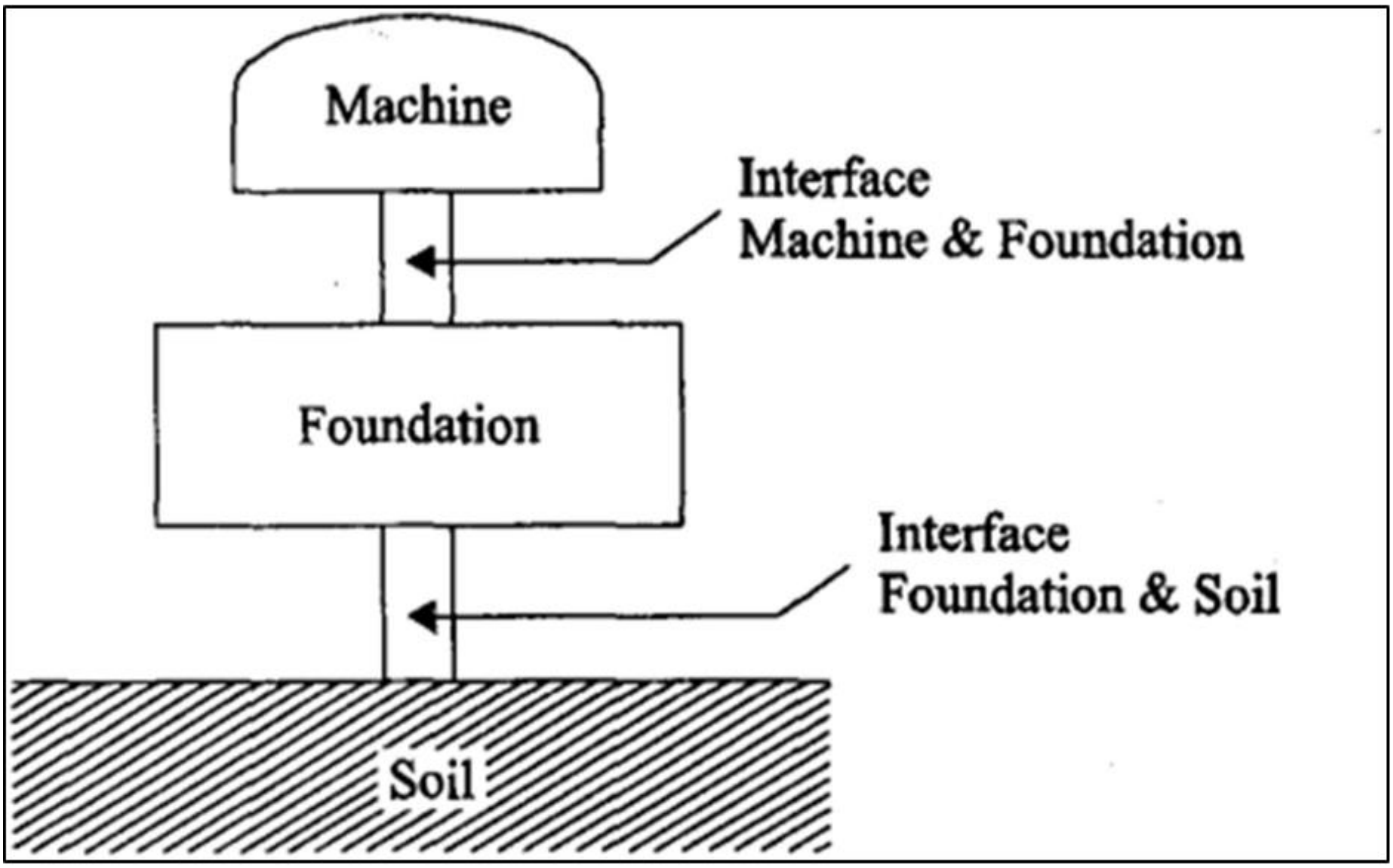

Fig 2 A Typical Machine Foundation System

➢ *It is Necessary to Acknowledge the Connections Between the Equipment, Foundation and soil:*

- The machine can be connected to the foundation through bolts, or it can be connected through isolation devices.
- Foundation can either be a solid pedestal resting directly on soil or it may be resting on a group of piles.

These interfaces as shown in figure 2 has to be carefully addressed to evaluate the responses of the entire system accurately.

➢ *Machines:*

Based on the type of motion, the machines are classified as below,

- *Rotary Machines:*

As the name suggests rotary means turning or being able to turn around an axis or a fixed point, having a part or parts that turn on an axis, as a machine.

- *Reciprocating Machines:*

A machine in which pistons connected to a crankshaft move back and forth in cylinders.

- *Impact type Machines:*

This type of machine was generally used in metal shaping and metal cutting industries.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

➢ *Based on the speed of operation, they are classified as follows:*

- Very low-speed machines (up to 100 rpm),
- Low-speed machines (100 to 1500 rpm),
- Medium speed machines (1500 to 3000 rpm),
- High-speed machines (3000 rpm and above).

➢ *Types of Machine Foundation Systems:*

There are many different types of machine foundation systems, out of all block types and frame types machine foundation systems are most commonly used across the world. Machine type and its characteristics play an important role in the selection of a particular foundation type. Some typical foundation types are given below,

- *Block Type of Machine Foundation:*

Block type of machine foundations consists of a pedestal resting on a mat type of foundation as shown in the figure. The foundation comprises of large mass and has a small natural frequency.

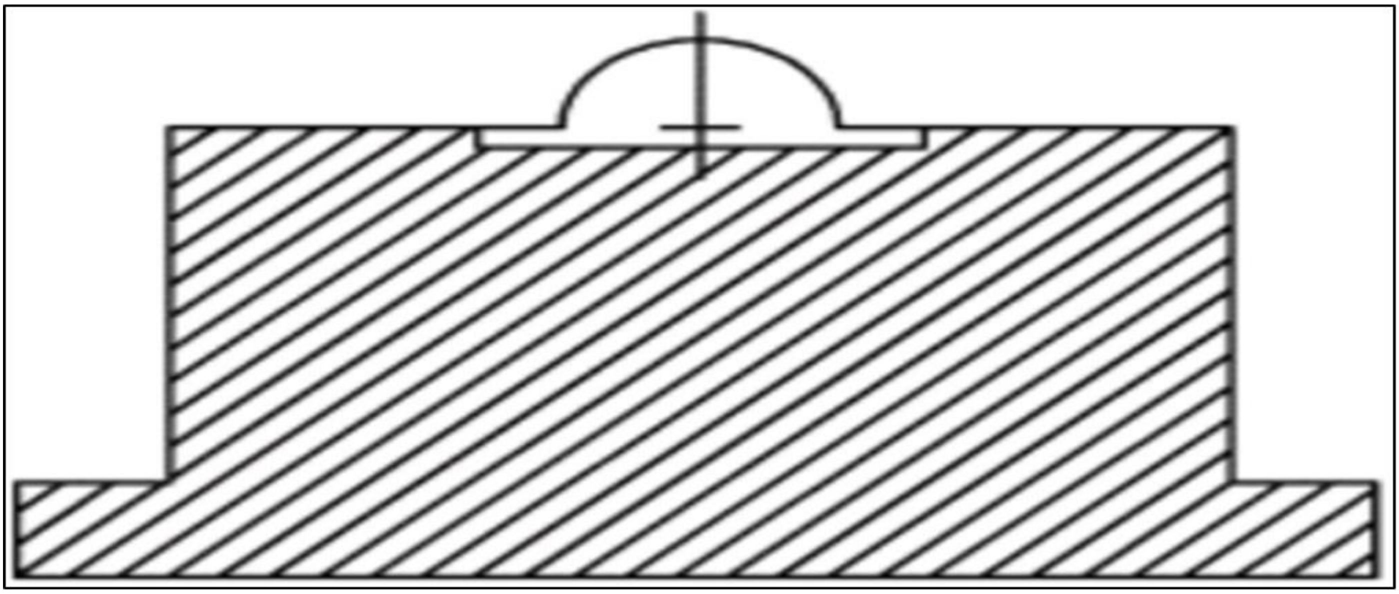

Fig 3 A block type of machine foundation

- *Box or Caisson type of Machine Foundation:*

This type of foundation consists of a hollow concrete block or pedestal as shown in figure 4. The mass of the box type of foundation is very less compared to the block type of machine foundation and the natural frequency is increased.

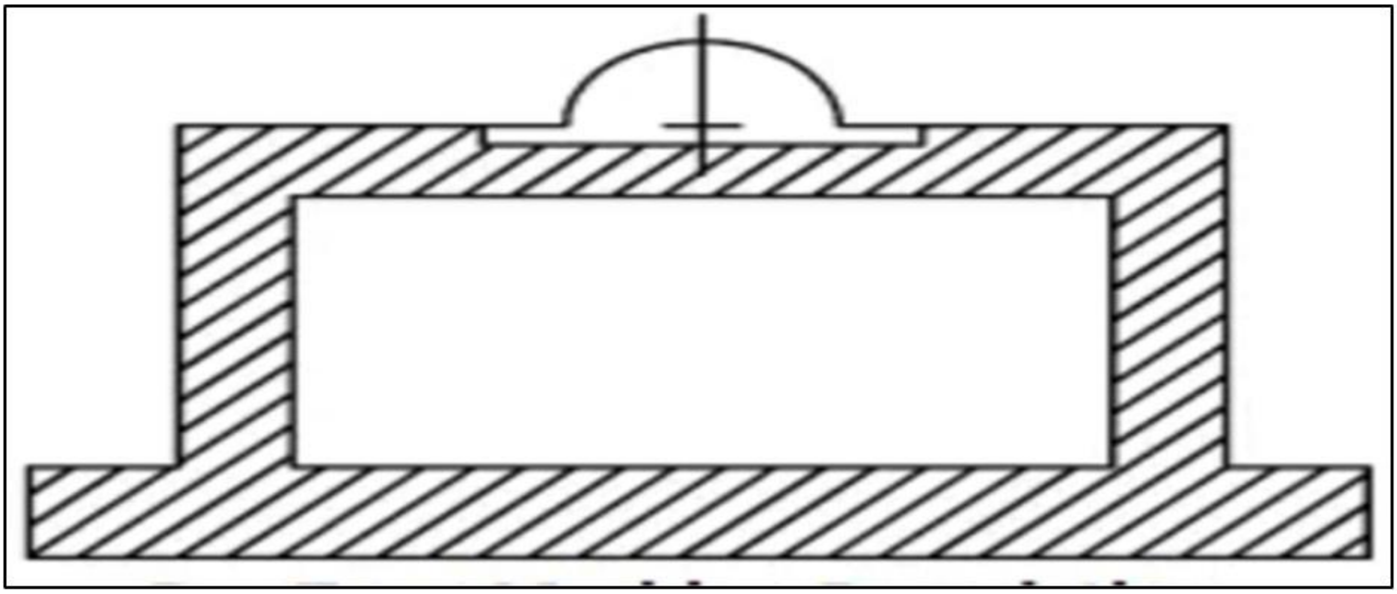

Fig 4 Box type of machine foundation

- *Wall type of machine foundation:*

This type of foundation consists of pair of walls with a top slab resting over them. The machine is supported by this top slab as shown in the figure. This type of foundation is economical for smaller projects. In this type of foundation, the vertical and horizontal members are made from the same homogenous material.

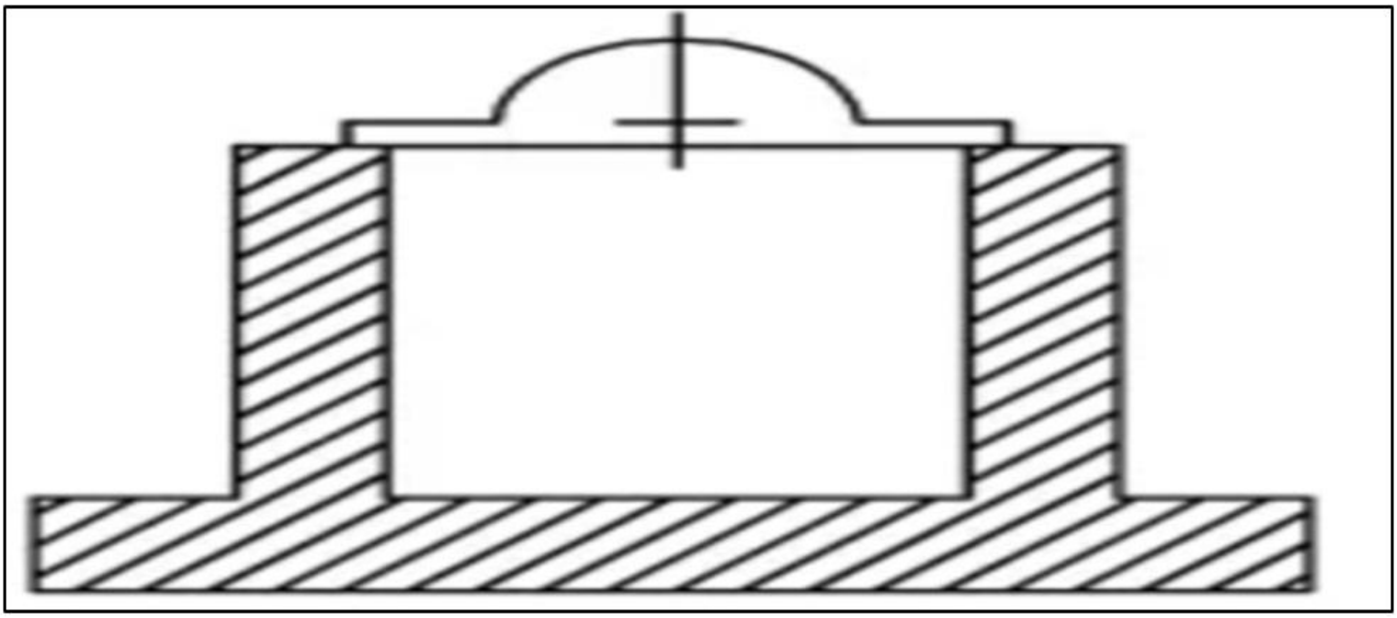

Fig 5 Wall type of Machine Foundation

- *Frame type of Machine Foundation:*

Frame type of machine foundations consists of vertical columns having a slab over their tops, where the machine is supported over the frame as shown in figure 6. This type of foundation is economical for larger projects. In this type of foundation, two different types of materials can be used for making vertical and horizontal sections of the foundation system.

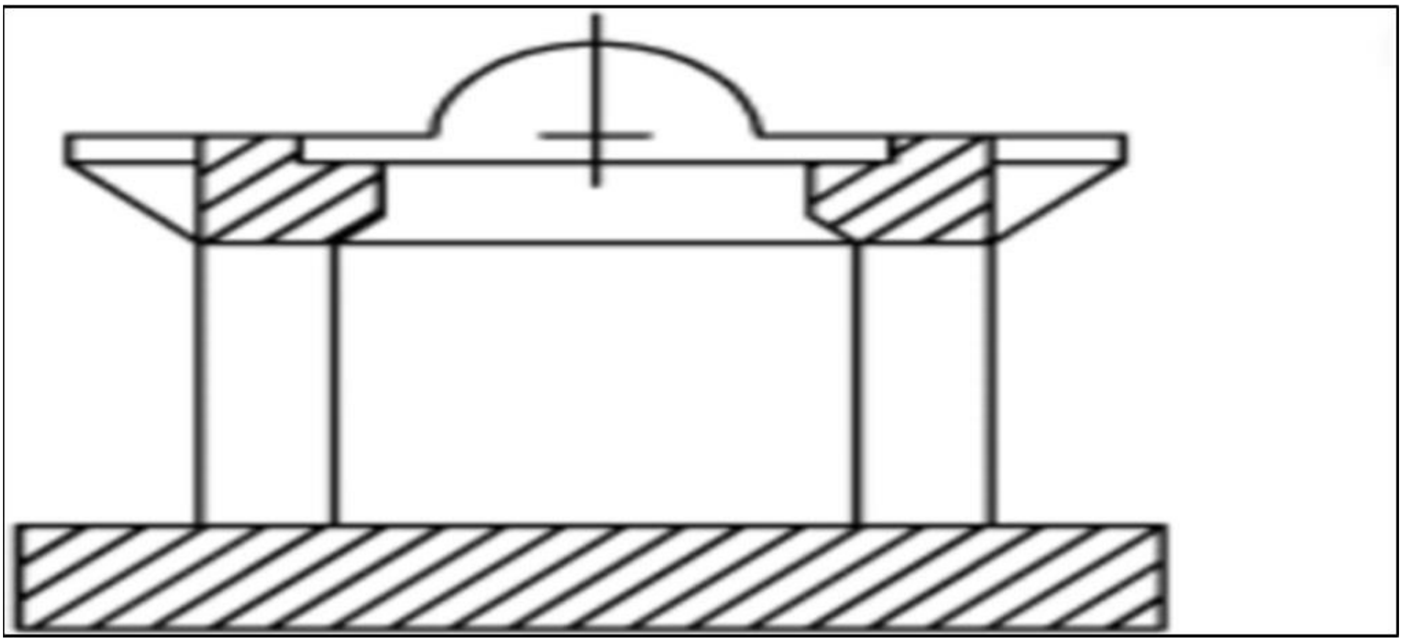

Fig 6 Frame type of machine foundation

➢ *Designing Requirements of Machine Foundations:*

- The foremost important requirement is that the foundation should be capable of carrying all superimposed loads to the ground by avoiding any type of deformations or crushing and cracks due to shear.
- The Centre of gravity of the structure and combined centre of gravity of the machine should be as far as possible in the same vertical line.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

- Resonance conditions must be avoided. The natural frequency of the system should be either higher or lower than the operating frequency of the machine.
- The amplitude should be within acceptable limits.
- The settlement should be also within acceptable limits.

➢ *Other General Requirements are as follows:*

- *Equipment requirements:*
  For the design of foundations, the following information on equipment is required:

✓ The geometric configuration of the machine,

✓ Loads from the machine: Mass of stationary as well as rotating parts of the machine and load-transfer mechanism from the machine to the foundation.
✓ Critical machine performance parameters: Critical speeds of rotors, balance grade and acceptable levels of amplitude.
✓ Dynamic forces generated by the machine: Forces generated during various operating processes and their transfer mechanism to the foundation for dynamic response analysis.
✓ Additional Forces: Forces generated under emergency or faulted conditions, test condition, Erection Condition& Maintenance Condition of the machine, Forces due to bearing failure (if applicable) for strength analysis of the foundation.

- *Soil requirements:*
  Typical soil parameters and dynamic parameters of soil used in machine foundation design are as shown in the figure:

E	Young's Modulus of Elasticity
G	Shear modulus
ν	Poisson's ratio
ρ	Mass density
ζ	Soil damping
C_u	Coefficients of uniform compression of the soil
C_ϕ	Coefficients of non-uniform compression of the soil
C_τ	Coefficients of uniform shear of the soil
C_ψ - •	Coefficients of non-uniform shear of the soil

Fig 7 Soil parameters required for the design of machine foundation.

- *Foundation Requirements:*
  The foundation parameters which are necessary for the design of the machine foundation are:

✓ Foundation geometry.
✓ Material Properties i.e., mass density, dynamic modulus of elasticity, Poisson's ratio, coefficient of thermal expansion, etc.,
✓ Strength parameters i.e., Yield stress, UTS, Allowable stress in compression, tension, bending and shear etc.

➢ *Important points to be considered while designing/planning a machine foundation:*

- The machine foundation should be separated from any building component, this can be achieved by installing an expansion joint surrounding components which are adjacent to the machine system. This gap avoids the transmission of vibrations from the system. A proper damping system has to be placed in the expansion joint.
- The machine foundation should be at a lower level than the surrounding structures.

C. Aim of the Project:

Analysis of machine foundation requires various loads and parameters. Therefore, equal importance is given to each of them to avoid any failures or safety concerns.

➢ *The Specific Aims of the Project are:*

- Static and Dynamic analysis of Block type and Frame type of Foundations
- Finding out the optimal parameters that can be optimised.
- Optimising the foundation accordingly.

D. Objectives of the Project:

➢ Studying the responses of the different machine foundations under vibrational and static loads caused by the rotary type of machines.
➢ Later on, investigating the optimal parameters required for the optimization of the foundation like dimensions, natural frequencies of the whole foundation or any particular individual members of the foundation system.
➢ Natural frequency has to be either far higher or far lower than the operating frequency of the machine.

The basic criteria to avoid resonance in the system is, that when the frequency of the foundation is closer to the operating speed of the machine, this results in a resonance condition where any structure can be failed very drastically. Based on the relation between the frequency of the system and the operating speed of the machine, there are two cases. One is under-tuned, where the natural frequency of the system is less than the operating speed of the machine as shown in the figure. The second one is over-tuned, in this case, the natural frequency of the system is higher than the operating speed of the machine as shown in the figure.

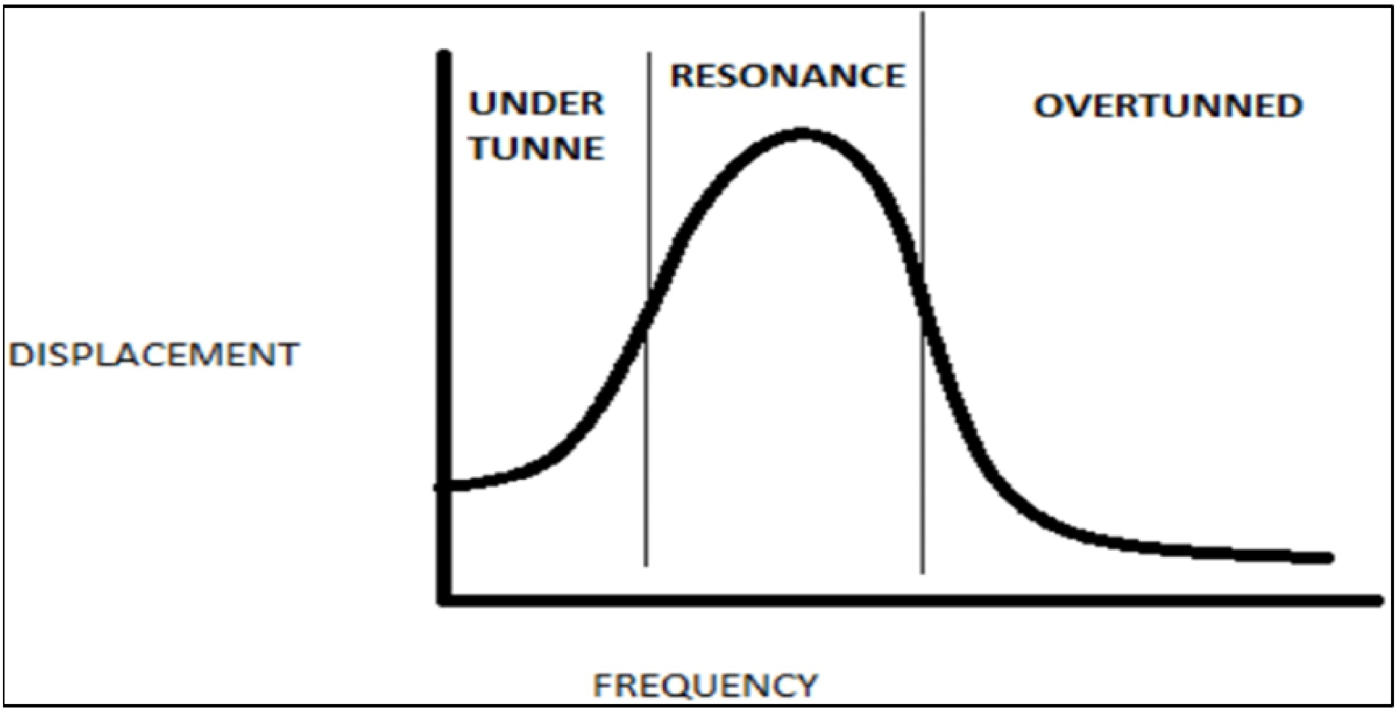

Fig 8 Machine Foundation Tuning

From the figure, we can realise that the frequency values in the middle portion cause resonance in the system.

E. *Methodology*:

➢ A theoretical approach to analysing the block and frame type of machine foundation is done with complete numerical calculations which involve the calculation of the centre of gravities, combined C.G, eccentricity, Natural frequencies and dynamic responses w.r.t to the dynamic forces.
➢ Finite Element Modelling and dynamic analysis are carried out in the ANSYS workbench 18.1, which involves the process of drafting the model in the software, defining and assigning all the required parameters like material property, extrusion etc., and meshing followed by analysis.
➢ Various optimization cases are evaluated and a critical case is considered for further optimizing the system as per requirements and without reducing the strength and stiffness considerations.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER TWO
LITERATURE REVIEW

The research papers, textbooks and projects related to the topic of study of this thesis were studied for gaining sufficient knowledge on the topic are listed below and the areas of study of those topics are explained briefly.

- Stefano Giorgetti, Alessandro Giorgetti, and Reza Jahroomi in the paper " Machinery foundation dynamic analysis: A case study on compressor foundation", covered the study of a faulty dynamical interaction between a machine and the foundation. This faulty interaction can lead to various unexpected and dangerous failures. For this reason, a careful analysis should be done on the foundation during the design phase to avoid resonance range of frequencies and low vibration speeds. In this paper, they have studied a reciprocating compressor foundation in both the analytical model and finite element model for obtaining different dynamical responses with different geometries is analysed. In particular, this paper reveals that using an analytical method causes acceptable results with the rigid body assumption, whereas in the FEA the block foundation has become less rigid, this scenario happens when the L/H(shorter length of foundation vs the height of the block) ratio is more than five.
- P.M.Shimpalle and Sindhu Panapalli in their paper ' Dynamic analysis of machine foundations using ANSYS discussed the importance of dynamic responses of foundation apart from static analysis due to static loads. Therefore, here they observed the natural frequencies of individual members of the foundation and check it with the operating frequency of the machine. They have analysed both the block type and the frame type of the machine foundation.
- Goranka Stimac, Sanjin Brat, & Roberto Zigulic on the topic 'Optimization of machine foundations using frequency constraints' have studied a frame type of machine foundation supporting a turbo generator machine. They have analysed the whole structure dynamically and evaluated the critical member that has to be optimised to avoid resonance in the system. As critical members, they have chosen the columns and optimized them individually using genetic algorithms and then these optimised columns are again analysed by incorporating them into the whole model to ensure stiffness and safety considerations. Later on, which is satisfied with all the requirements is considered a solution.
- M.A.Lopes, F.J.C.P Soeiro, & J.G. Santos Da Silva on the topic 'Structural optimization of concrete volume for machine foundation using genetic algorithm' have explained the optimization of machine foundations using genetic algorithms. In this paper, they have adopted a block type of foundation and evaluated the loading points on the foundation where the loads due to the machine are concentrated. Later the foundation is analysed dynamically and these loading points are considered crucial points for optimization. The analysed frequency values at these loading points are used to carryout the optimization procedure in the Matlab software with a user-defined objective function for concrete volume.
- K.G.Bhatia, in his handbook for design of machine foundations, explained very clearly and elaborately the machine foundations that are used in this world using both theoretical and finite element analysis. He also included the various machine foundation types for reciprocating machines, rotatory machines and impact machines. Also designed them accurately according to the problem definition. He also included the degrees of freedom system of foundations and damping of systems. A topic about usage and analysis of vibration isolation systems to the foundations is explained very adequately and in very easy literature.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER THREE
THEORETICAL ANALYSIS OF MACHINE FOUNDATIONS

In the present investigation two types of machine foundations, Block type and Frame type of foundations are chosen for modelling and analysing accordingly. The below section describes the theoretical analysis of the foundations in a detailed approach according to the textbook "Handbook of machine foundations" by K.G. Bhatia. Firstly, the Block type of foundation is analysed and later the frame type of foundation.

➢ *Block type of Machine Foundation:*

The Block foundation, in general, has six degrees of freedom and therefore six natural frequencies (one corresponding to each mode of vibration). Three of them are translations along three principal axes and the other three are rotations about the axes, as shown in Figure 9 The natural frequency is calculated and checked against the operating frequency of the machine.

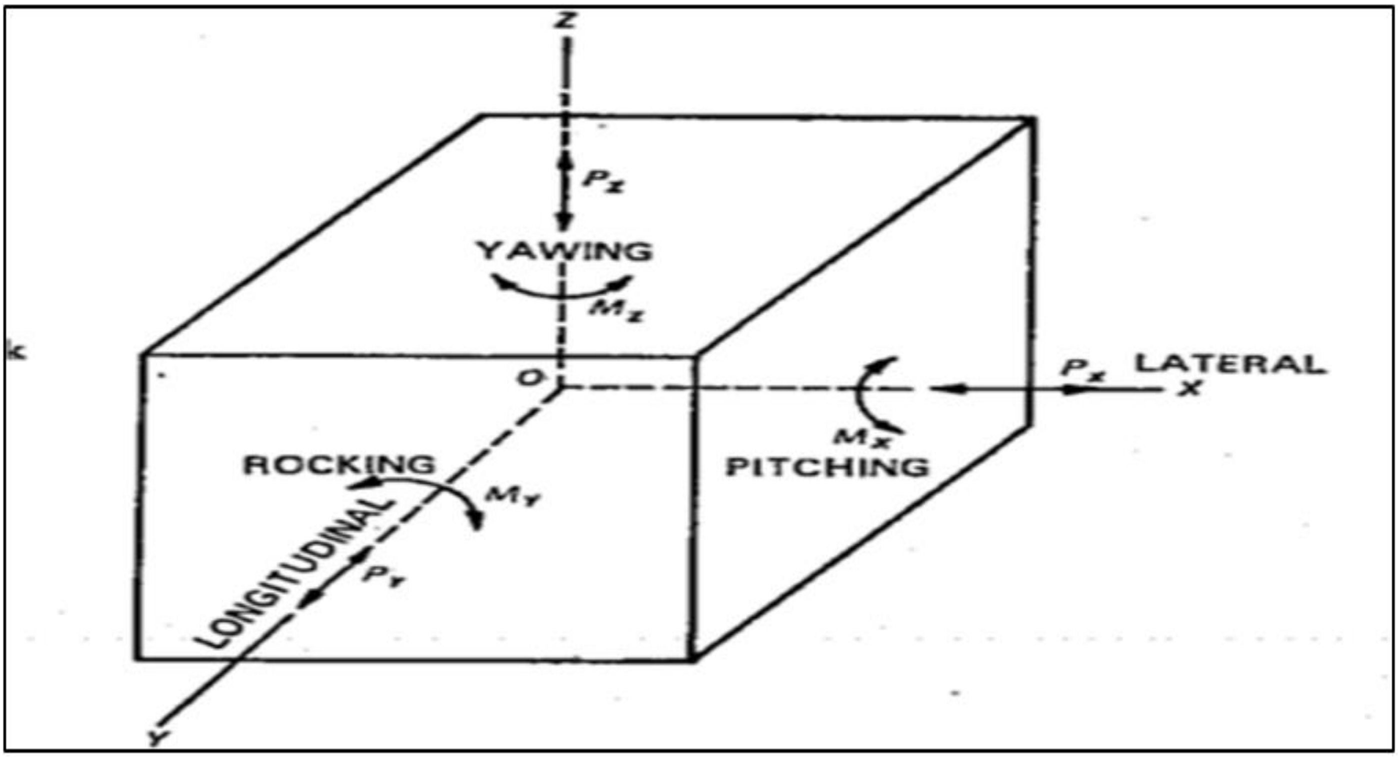

Fig 9 Modes of Vibration of a Block Foundation.

Out of many empirical methods described in the handbook and from the years of practise, Barkan's method is recommended for the dynamic analysis of block foundations. This method is based on linear spring theory, which neglects the effects of damping and participating of soil mass. This method is simple and fairly predicts the true behaviour of the structure. The theoretical basis of this method is that the combined centre of gravity of the machine and foundation lies in the same vertical line as the centre of gravity of the base plane.

➢ *Problem Definition:*

A 132kw rotatory compressed machine is supported by a block type of foundation, whose dimensions are as described below along with the soil parameters required for the analysis.

- *Base Dimensions of the Foundation*

Table 1 Base Dimensions of the Foundation

Dimension	**Length (m)**	**Axis**
Length	3.3	X-axis
Breadth	1.35	Y-axis
Width	0.8	Z-axis

- ✓ The width of the base raft is 0.3 m
- ✓ Area of the block foundation is = Af = 4.455 m2
- ✓ The sectional views of the block foundation are as shown in figures 10, and 11

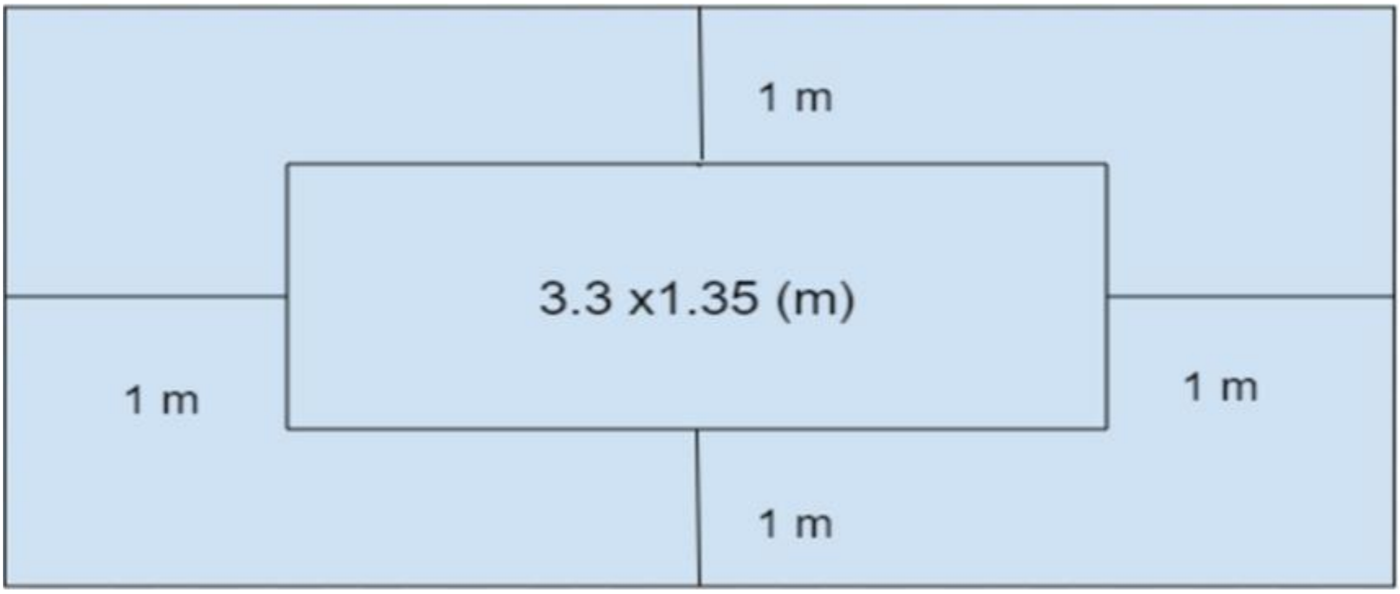

Fig 10 Top view of the block foundation

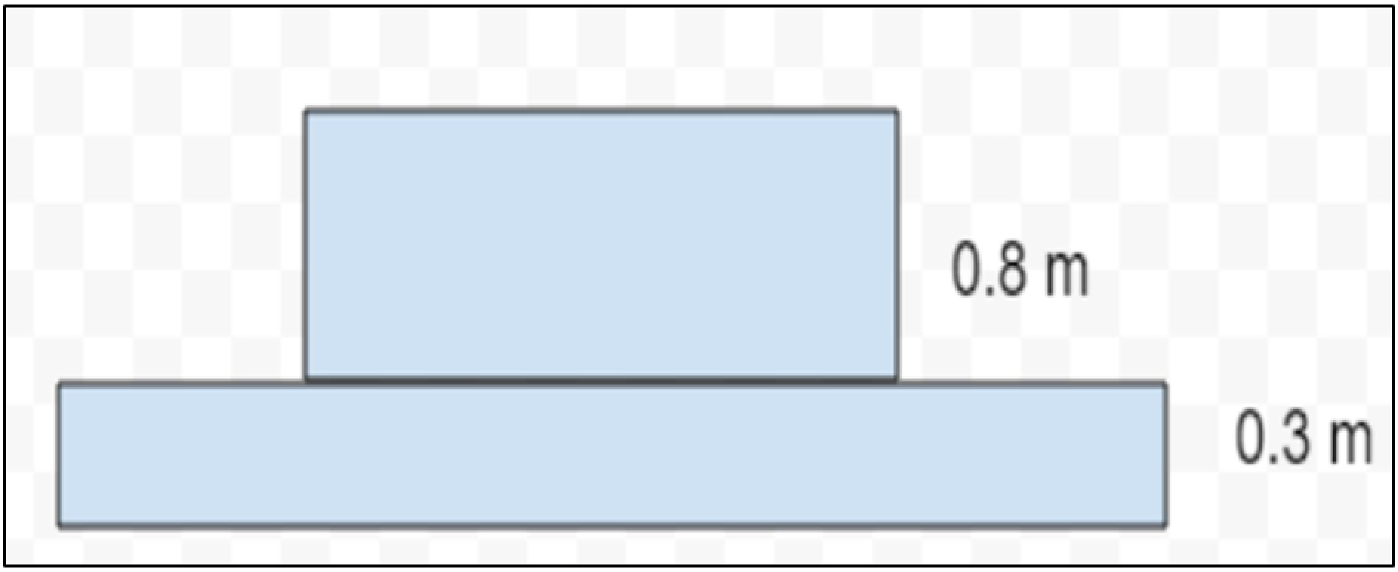

Fig 11 Sectional view of the block foundation

- *Soil parameters:*

Table 2 Soil Parameters for Block Foundation

S.no	Coefficient	Value (t/m3)
1.	Coefficient of elastic uniform compression (assumed) Cz	3000
2.	Coefficient of elastic non-uniform compression Co	6000
3.	Coefficient of elastic uniform shear	1500
4.	Coefficient of elastic Non-Uniform shear	2250

- *Computation of Centre of gravity: Centre of gravity of Machine:*

Table 3 CG and Static moment of mass of the machine

Sr No.	**Item Name**	**w (ton)**	**Mass**	**Centre of Gravity**			**A static moment of Mass**		
		m	***X_i***	***Z_i***	***Y_i***		***mx_i***	***mz_i***	***my_i***
		t- sec/m3					***KN m2/Sec***	***KN m2/Sec***	***KN m2/Sec***
1	Pump	0.386	0.039	0.935	1.490	0.675	0.037	0.059	0.027
2	Gear Box	0.014	0.001	1.835	1.490	0.675	0.003	0.002	0.001
3	Turbine	1.237	0.126	2.635	1.490	0.675	0.332	0.188	0.085

- *Centre of Gravity of base Foundation:*

Table 4 CG and Static moment of mass of Foundation

Sr No.	Lx	Ly	Lz	Weight Mass			Centre of Gravity			A static moment of Mass	
	m	***m***	***m***				***X_i***	***Z_i***	***Y_i***	***mx_i***	***mz_i***
		t	***t-sec/m3***					***KN m2/Sec***	***KN m2/Sec***	***KN m2/Sec***	***t***
Foundation											
1	3.3	1.35	0.800	8.910	0.908	1.650	0.400	0.675	1.499	0.363	0.613
				10.547	**1.075**				**1.870**	**0.612**	**0.726**

Common Centre of gravity of machine and foundation:

Xc= 1.740m (In length)
Yc= 0.569m (In vertical)
Zc= 0.675m (In width)

% Eccentricity of the common centre of gravity of Machine & Foundation
Eccentricity in X-direction: ex= (Xo/B)x 100 = 2.7125% [within the permissibility]
Eccentricity in Z-direction: ey= (Yo/L)x 100= 8.24 e-5% [within the permissibility]

According to K.G.Bhatia textbook The eccentricity of Block Foundations should be as close to zero and in no case, it should not exceed 5% of the base dimensions in the respective direction. For a very long foundation having L/B > 3, it is further desirable (but not necessary) that the eccentricity be contained within 2% along the length, Whereas the eccentricity along the width remains within 5%.

- *Calculation of mass moment of inertia:*
  Perpendicular to plane X: $Qx = mi$ Similarly holds for the axis Y and Z

$$Qx = mi \frac{\sqrt{L\_y^2+L\_x^2}}{12} \quad [3.1]\ ;$$

$$Xxi = mi(Zo^2 + Yo^2) \quad [3.2]$$

The mass moment of inertia of the machine and foundation are as described in Table 3.5

Table 5 mass moment of inertia of Block foundation

Sr No.	***Item***	***Q´x***	***Q´z***	***Q´Y***	***X_0=Xc- X_i***	***Z_0=Zc- Z_i***	***y_0=Yc- Y_i***	***Xx_i***	***Zz_i***	***Yy_i***
		t- sec2-m	***t-sec2-m***	***t-sec2- m***	***m***	***m***	***m***	***$t\text{-}sec^{2\text{-}}m$***	***$t\text{-}sec^{2\text{-}}$ m***	***$t\text{-}sec^{2\text{-}}$ m***
1	Pump				0.805	-0.921	0.000	0.033	0.025	0.059
2	Gear Box				-0.095	-0.921	0.000	0.001	0.000	0.001
3	Turbine				-0.895	-0.921	0.000	0.107	0.101	0.208
4	Foundation	0.186	0.962	0.873	0.090	0.169	0.000	0.026	0.007	0.033
		0.186	0.962	0.873				0.133	0.108	0.241

- *Calculation of Horizontal Forces:*

Unbalanced horizontal force parallel to the shaft

Fx1 (max horizontal force Primary Force) = 2500 N
Fx2 (max Horizontal force secondary Force) = 2000 N

Total Fx= Fx1+Fx2 = <u>4500 N</u>
<u>0.45 t</u>

Unbalanced horizontal force perpendicular to the shaft

Fy = 4500 N

Unbalanced vertical forces
Fz = 11000 N

- *Calculation of Moments at the centre of gravity:*

Moment @ X-X axis:

Mx1= (Fy) x (Distance up to centre of gravity) = 0.671 t-m
Mx2= max vertical primary moment @axis X-X = 0.15 t-m
Mx3= max vertical secondary moment @ axis X-X = 0.12 t-m
Mx = 0.941 t-m My is also the same as Mx. Therefore, My= 0.941 t-m.

Moment @ Z-Zaxis:

Mz1= maximum vertical primary moment @Y-Y =0.12 t-m
Mz2= maximum vertical secondary moment @Y-Y= 0.12 t-m Mz= 0.24 t-m

- *Calculation of Stiffness factor:*

vertical stiffness factor = Kz= (Cz) x (Af) = 1.34E+04 t/m
Horizontal Stiffness factor = Kt= (Ct) x (Af) = 6.68E+03 t/m
Stiffness facto for torsion (K_si)= (Csi) x (Iz) = 1.06E+04 t/m

Where,

Ix= Moment of inertia along X-axis = 0.6766031 m4 Iy= Moment of inertia about Y-axis= 4.0429125 m4 Iz= Moment of inertia about Z-axis= 4.7195156 m4

Stiffness factor for rotation (in vertical Plane Xz) K_oy = (Co) x (Iy)= 2.43E+04 t-m/rad
Stiffness factor for rotation (in vertical Plane Yz) K_ox = (Co)x (Ix) = 4.06E+03 t-m/rad
An overall summary of the stiffness factors is given in table 3.6.

Table 6 Stiffness factors of block foundation

S.no	Stiffness factor	Value
1.	Kz (Vertical stiffness factor)	1.34E+04 t/m
2.	Kt (Horizontal stiffness factor)	6.68E+03 t/m
3.	K_si (Stiffness factor for torsion)	1.06E+04 t/m
4.	K_oy (Rotation stiffness factor in plane XZ)	2.43E+04 t-m/rad
5.	K_ox (Rotation stiffness factor in plane YZ)	4.06E+03 t-m/rad

- *Calculation of natural frequency:*

The machine foundation undergoes six modes of vibration i.e., three translational and three rotational. As far as possible these natural frequencies shouldn't be in direct resonance with the operating speeds of the machine. A margin of ±20 % away from the operating speed of the machine is considered the safest range of natural frequencies. In the case of resonance, the foundation has to be redesigned according to the requirements. The natural frequencies corresponding to these modes are as follows:

- *For Coupled translation along Z-axis:*

W(z)= $\sqrt{Kz/m}$ = 111.495 1/sec
f(z)= W(z)/(2π) = 17.754 cps

The operating speed of the machine is = Om= 2975 rpm = 49.583 cps 3.1.1 (h-1)
The ratio of natural frequency to the operating speed = Fz/(Om) = 0.358
Since it is not closer to 1, no resonance is possible in this mode of vibration.

- *For coupled motion (Translation along X-axis and rocking along Y-axis) in the X-Z plane:*

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

The natural frequencies are as follows:

$$\omega\_n1^2 = \frac{1}{\gamma y}(\omega\_oy^2+\omega\_x^2) - \frac{1}{2\gamma y}(\sqrt{(\omega_x{}^2 + \omega_{oy}{}^2) - 4\gamma y\omega_x{}^2\omega_{oy}{}^2} \quad [3.3]$$

$$\omega\_n2^2 = \frac{1}{2\gamma y}(\omega_{oy}{}^2 + \omega_x{}^2) + \frac{1}{2\gamma y}(\sqrt{(\omega_x{}^2 + \omega_{oy}{}^2) - 4(\gamma y)\omega_x{}^2\omega_{oy}{}^2} \quad [3.4]$$

Where,

$$\omega\ oy^2 = \frac{(Co*Iy)-(W*Zc)}{Q\_oy} \quad [3.5]$$

$$\omega\ x^2 = \frac{Ct*Af}{m} \quad [3.6]$$

$$Q_{oy} = (Qy + (m * Zc^2)) \quad [3.7]$$

$$Qy = (Q'y + yy) \quad [3.8]$$

$$\gamma v = \frac{Qoy}{Qy} \quad [3.9]$$

After calculating all the values necessary for the coupled natural frequency in the X-Z plane, the values are as follows in the table 3.7

Table 7 Coupled Natural frequency in X-Z plane

S.no	Terms	Value	Units
1	Qy	1.1140	t-m/sec^2
2	Qoy	1.46229	t-m/sec^2
3	γy	0.762	-
4	ωx2	6215.54	sec^-2
5	ωoy2	16588.71	sec^-2
6	ωn1	156.15	sec^-1
7	ωn2	74.50	sec^-1

Now, checking for resonance
Ratio of ωn1 and operating speed of the machine *O*m: 0.501 (No resonance)
Ratio of ωn2 and operating speed of the machine *O*m: 0.239 (No resonance)

- *Coupled natural frequency (translation along Z-axis and rocking about X-axis) in X- Z plane*

Similarly, the same equations hold for the coupled natural frequencies in plane Y-Z plane, with very slight changes in the subscripts, The various calculated values are tabulated in table 3.8.

$$\omega\_n3^2 = \frac{1}{2\gamma x}(\omega\_ox^2+\omega\_y^2) - \frac{1}{2\gamma y}(\sqrt{(\omega_{ox}{}^2 + \omega_y{}^2) - 4\gamma y\omega_{ox}{}^2\omega_y{}^2} \quad [3.10]$$

$$\omega\_n4^2 = \frac{1}{2\gamma x}(\omega_{ox}{}^2 + \omega_y{}^2) + \frac{1}{2\gamma y}(\sqrt{(\omega_{ox}{}^2 + \omega_y{}^2) - 4(\gamma y)\omega_{ox}{}^2\omega_y{}^2} \quad [3.11]$$

Where,

$$\omega\ ox^2 = \frac{(Co*Ix)-(W*Zc)}{Q\_ox} \quad [3.12]$$

$$\omega\ y^2 = \frac{Ct*Af}{m} \quad [3.13]$$

$$Q_{ox} = (Qy + (m * Zc^2)) \quad [3.14]$$

$$Qx = (Q'y + yy) \quad [3.15]$$

$$\gamma x = \frac{Qox}{Qx} \quad [3.16]$$

- *Therefor the calculated values are as follows:*

Table 8 Coupled natural frequency values in the Y-Z plane

S.no	Term	Value	Units
1	Qox	0.667599	t-m sec^2
2	Qx	.319	t-m sec^2
3	γx	0.4783	-
4	ω_y2	6215.54	sec^-2
5	ω_ox2	6071.93	sec^-2
6	ω_n3	148.744	sec^-1
7	ω_n4	59.72	sec^-1

Now, checking for resonance
The ratio of ω_n3 and the operating speed of the machine Om gives= 0.477 (No resonance)
The ratio of ω_n4 and the operating speed of the machine Om gives= 0.192 (No resonance)

- *For torsional motion about Z-axis:*

$Qz = Q'z + Zz$

= 1.0706 t-m sec^2

$$\omega_{si} = \sqrt{(C_{si} * \frac{Iz}{Qz})} \quad [3.17]$$

= 99.594 sec^-1

The ratio of natural frequency to machine frequency Om gives the resonance check. In this case, the ratio is 0.32 <1, Hence, No resonance.

- *Amplitude of vibration:*

Whenever the natural frequencies of the foundation lie within ±20% of the operating speed of the machine, it is necessary to calculate the amplitude at the corresponding mode of vibration. Since there is no resonance condition in the present foundation system, there is no need of determining the amplitudes of vibration.

➢ *Frame type of foundation:*

- *Foundation Data:*

Foundation material properties are as follows: Concrete grade M25
The mass density of concrete 2.50 t/m3
Elastic Modulus E 3.00e+07 kN/m2
Poisson’s ratio 0.15
Top Deck L= 13.80m; B= 8.00 m; thickness= 1.80m Base raft L= 13.30 m; B= 8.00 m; thickness= 2.00m

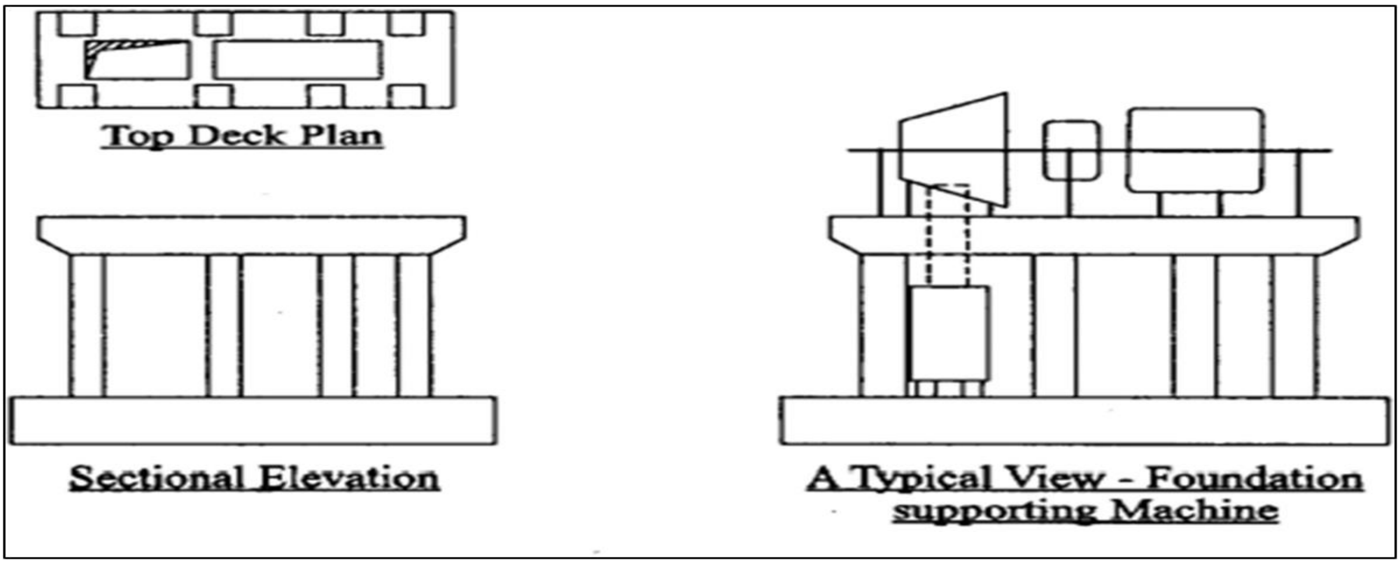

Fig 12 Typical Machine foundation geometry used for FE analysis

- *Frame sizes of the foundation:*

Table 9 Frame sizes of Machine Foundation

Frame Number	Frame 1	Frame 2	Frame 3	units
Frame beam width	1	1	1	M
Frame beam Depth	1.8	1.8	1.8	m
Frame span	9.7	9.7	9.7	m
Beam moment of inertia	0.49	0.49	0.49	m3
Column moment of inertia	0.08	0.08	0.11	m3

- *Machine Mass on Frames*

Frame 1
Mass@ Beam centre W1= 400kN
Total Mass =400kN

Frame 2
Mass@ Frame Beam Centre W2= 360kN
Mass W3@ 1.7m from left column W3= 100kN
Mass W3@ 1.7m from right column W3= 100kN
Total Mass on Frame 2 = 360kN

Frame 3
Mass@ Frame Beam Centre NIL
Mass W4@ 1.7m from left column W4= 100kN
Mass W4@ 1.7m from Right column W4= 100kN
Total Mass on Frame 3 = 200kN

Total Machine Mass 1160kN

Fig 13 Machine Mass on Frames

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

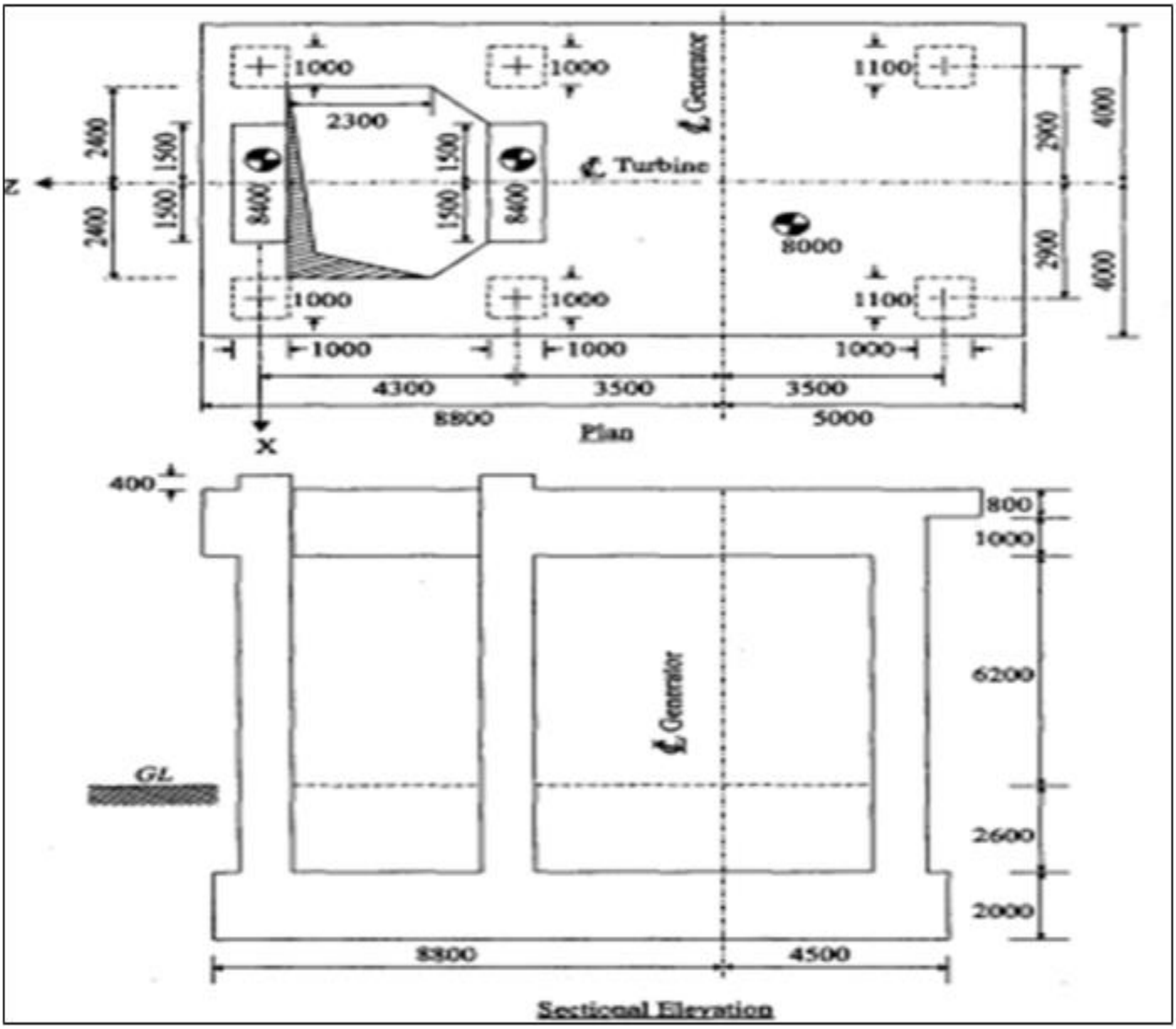

Fig 14 Sectional Elevation of Frame Foundation

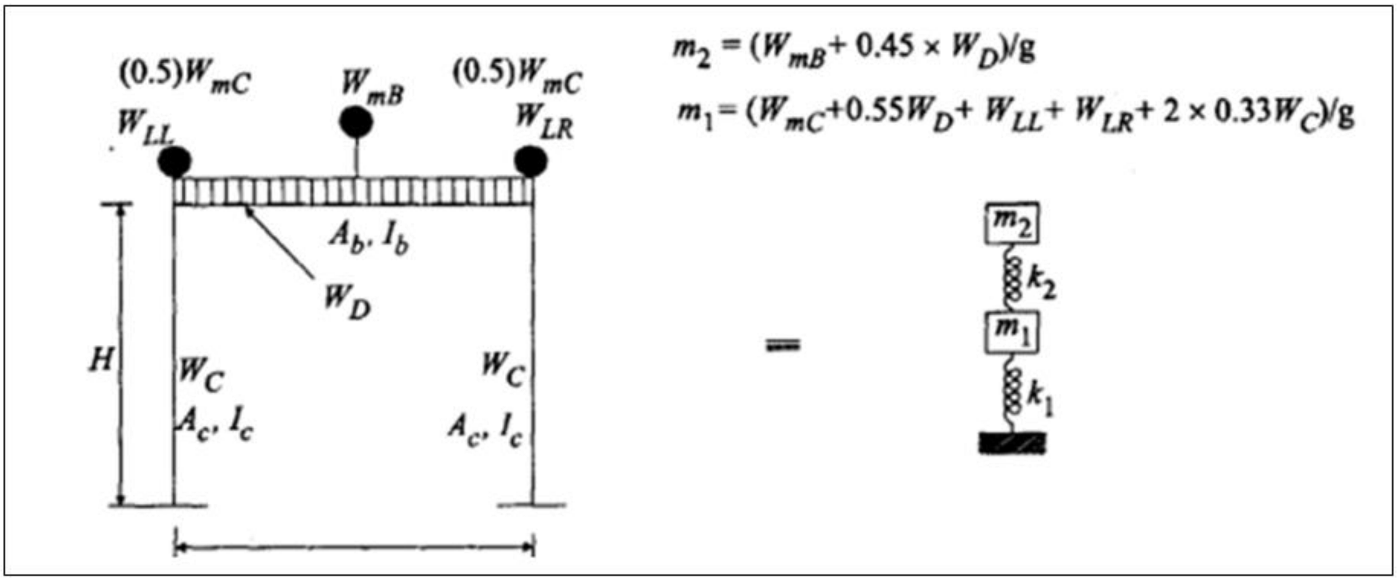

Fig 15 Mass-spring system of frame

Volume 9, Issue 11, November– 2024 International Journal of Innovative Science and Research Technology
ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

- *Soil data:*

Table 10 Soil parameters for frame foundation

S.no	Coefficient	Value (kN/m3)
1.	Coefficient of elastic uniform compression (assumed) C_u	4×10^4
2.	Coefficient of elastic non-uniform compression $C_\emptyset$	8×10^4
3.	Coefficient of elastic uniform shear C_t	2×10^4
4.	Coefficient of elastic Non-Uniform shear C_ψ	3×10^4

- *Eccentricity:*

For frame foundations, there are two conditions to the term eccentricity, as described below:

- *Overall Eccentricity:*

It is the total distance between the Centre of mass of the overall system (machine+ foundation) and the centre of stiffness i.e., CG of the base contact area of the foundation with the soil. This overall Eccentricity should be restricted to 5%of the respective base dimension of the foundation.

- *Top Deck Eccentricity:*

It is the distance between the Centre of mass C_m (i.e. combined CG of machine mass, top deck mass and 23% of column mass) and the Centre of stiffness of the frame C_k in transverse as well as the longitudinal direction. It is desirable if top deck eccentricity is restricted to 1% of the corresponding top deck dimension.

Top Deck Eccentricity: Frame 1 and Frame 2

$$k = \frac{I_b / L}{I_c / H} \quad [3.18]$$

$$= 9.75$$

Frame 3: k= 7.33

Lateral stiffness is given by following equation:

$$k_x = \frac{12EI_c}{H^3} \times \frac{1+6k}{2+3k} \quad [3.19]$$

Frame 1: Lateral Stiffness is Calculated as

$$k_x = \frac{12 \times 3 \times 10^7 \times 0.08}{9.7^3} \times \frac{1 + 6 \times 9.75}{2 + 3 \times 9.75}$$

$$= 6.01 \times 10^4 \text{ kN/m}$$

Frame 2: Lateral stiffness is Derives as

$$k_x = \frac{12 \times 3 \times 10^7 \times 0.08}{9.7^3} \times \frac{1 + 6 \times 9.75}{2 + 3 \times 9.75}$$

$$= 6.01 \times 10^4 \text{ kN/m}$$

Frame 3: Lateral stiffness is determined as

$$k_x = \frac{12 \times 3 \times 10^7 \times 0.11}{9.7^3} \times \frac{1 + 6 \times 7.33}{2 + 3 \times 7.33}$$

$$= 8.14 \times 10^4 \text{ kN/m}$$

Total Lateral Stiffness is k_x= (6.01 × 104 + 6.01 × 104 + 8.14 × 104) kN/m
= 2.02 × 105 kN/m

Centre of stiffness with respect to frame 1= C_k = 5.83 m

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

Centre of mass with respect to frame 1 = Cm = 5.93 m
Eccentricity = (Cm-Ck) x 100/13.8 = 0.10 x 100/13.8 = 0.72% (O.K)

- *Natural frequency:*

To calculate the natural frequency of the system we require the stiffness and mass of the system. Here in this investigation, instead of the whole system, we considered frames as the components to derive the natural frequencies to check the resonance of the system. Therefore, the calculation of stiffness and mass of the frames has become necessary. The following section describes and determines the stiffness and mass of 3 frames of the foundation system.

Vertical Vibration (Two Dof System) Frame 1

- *Mass:*

Total Machine weight on frame 1 401 kN
Machine weight at beam centre. 400 kN
Machine weight @off centre location NIL
Total machine weight at frame beam centre (Wmb) 400 kN
Machine weight transferred to column top (Wmc) 0

Frame 1

W_m	**Machine weight on Frame Beam**	$W_m = 400$ kN
W_D	**Distributed Load on Frame Beam**	
	Self weight of Frame Beam BC	$1.0 \times 1.8 \times 5.8 \times 2.5 \times 9.81 = 256$ kN
	Weight of Cantilever slab projection	$0.5 \times 1.8 \times 5.8 \times 2.5 \times 9.81 = 128$ kN
		$W_D = 256 + 128 = 384$ kN
W_{LL}	**Load Transferred from Longitudinal Beam on column Top - Left**	
	Self Weight of Beam BF + Projection	$(4-2.4) \times 1.8 \times 0.5 \times 4.3 \times 2.5 \times 9.81 = 151.8$ kN
	Portion of the slab at corner	$(4-2.4) \times 1.8 \times 1.0 \times 2.5 \times 9.81 = 70.6$ kN
		$W_{LL} = 222$ kN
W_{LR}	**Load Transferred from Longitudinal Beam on column Top - Left**	
	Reaction from Beam BF (Self Weight of Beam BF + Projection)	$(4-2.4) \times 1.8 \times 0.5 \times 4.3 \times 2.5 \times 9.81 = 152$ kN
	Portion of the slab at corner	$(4-2.4) \times 1.8 \times 1.0 \times 2.5 \times 9.81 = 70$ kN
		$W_{LR} = 152 + 70 = 222$ kN
W_C	**Self Weight of each Column**	$W_C = 1.0 \times 1.0 \times 9.7 \times 2.5 \times 9.81 = 238$ kN
Total Mass of Frame 1		$m_x = (400 + 384 + 222 + 222 + 0.23 \times 2 \times 238)/9.81 = 136$ t

Fig 16 Total Mass on frame 1 of frame foundation.

From figure 16 we can see that all the loads acting on the column are determined and can be concluded as follows:

$$WD = 384\ kN;\ WLL = 222kN;\ WLR = 222kN;\ Wc = 238kN$$

But, in this investigation, the frame is considered a mass-spring system as shown in figure 3.5. Therefore, it is required to calculate the masses of the mass-spring system using the equations 3.20 & 3.21

$$m_2 = \frac{W_{mB}+0.45\times W_D}{g} \quad [3.20]$$

$$m_1 = \frac{W_{mC}+0.55W_D+W_{LL}+W_{LR}+0.33\times 2\times W_c}{g} \quad [3.21]$$

After substitutions, the values of $m2$ & $m1$ are as follows:

$$m_1 = 83\ t$$

$$m_2 = 58.4\ t$$

- *Stiffness*:

  The stiffness is given by the equations 3.22 & 3.23

$$k_1 = \frac{2EA_c}{h} \quad [3.22]$$

$$k_2 = \frac{1}{y_2} \quad [3.23]$$

Where,

$$y_2 = \frac{L^3}{96EI_b} \times \frac{2k+1}{k+2} \quad [3.24]$$

- *The values are as follows:*

$$k_1 = 6.18\ X\ 10^6 \frac{kN}{m}$$

$$y_2 = 3.34 \times 10^{-7}\ m^{-1}$$

$$k_2 = 3 \times 10^6\ kN/m$$

- *Limiting Frequencies and Mass Ratio:*

$$P_{L2} = \sqrt{\frac{k_2}{m_2}} \quad [3.25]$$

$$P_{L1} = \sqrt{\frac{k_1}{m_1}} \quad [3.26]$$

$$\gamma = \frac{m_2}{m_1} \quad [3.27]$$

Equations 3.25, 3.26 are the limiting frequencies that has to be determined. The natural frequencies of the frame 1 are determined using the equations 3.28 & 3. 29.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

$$P_1 = \sqrt{\frac{1}{2}\{[P_{L2}^2(1+\gamma) + P_{L1}^2] - \sqrt{(P_{L2}^2(1+\gamma) + P_{L1}^2) - 4P_{L2}^2P_{L1}^2}\}} \quad [3.28]$$

$$P_2 = \sqrt{\frac{1}{2}\{[P_{L2}^2(1+\gamma) + P_{L1}^2] + \sqrt{(P_{L2}^2(1+\gamma) + P_{L1}^2) - 4P_{L2}^2P_{L1}^2}\}} \quad [3.29]$$

Therefore, by Calculating all the values we get the natural frequencies of frame 1.

In the process of determining is safe for all the frames, hence the values of all three frames are tabulated in table 11. From section 11, we can know that the operating speed of the machine is around 50 Hz, Resonance check is something we divide our natural frequencies with the operating speed of the machine.

Table 11 Natural frequencies and resonance check for frame foundation

Frame number	**Natural frequency (P_1)**	**Natural frequency (P_2)**	**Resonance check w.r.t P_1**	**Resonance check w.r.t P_2**
Frame 1	26.96 Hz	38 Hz	0.53	0.76
Frame 2	22.4 Hz	33.46 Hz	0.45	0.66
Frame 3	25.7 Hz	38.7 Hz	0.51	0.77

As stated in section 3.1.1. i, there is no requirement for checking amplitude, if the resonance is under the specified limits.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER FOUR
FINITE ELEMENT MODELLING OF MACHINE FOUNDATIONS

➢ *Introduction:*

The finite element analysis is a very useful tool to solve problems mainly in the fields of Engineering and mathematics. The problems in the areas of Engineering and Mathematical physics can be solved using the finite element method mainly by dividing the problems into sub-domains of a finite number. In Engineering, particularly Mechanical problems, Structural analysis, Heat transfer, Fluid flow, Mass transport, and Electromagnetic potential are solved by creating respective elements.

For solving civil engineering problems, the inadequacy of traditional methods for finding the response of the structure is the reason for the adoption of the FEA and solving for the responses, especially in the cases of discontinuities such as geometry and loading. It is generally not possible to obtain a solution using the equations of classical mechanics which generate complex differential equations.

FEA also generates mathematical formulations by employing matrix algebra and can yield the values of the desired unknowns at nodes. Without the usage of FEA one requires to solve ordinary or partial differential equations. For example, if we want to analyse a beam in general, we use a double integral equation, which, because of the complicated geometries, loadings, and material properties, the solutions are not easy to obtain. Although the FEA doesn't give an exact solution it gives a solution which is near to the exact solution.

As discussed above the mathematical formulations is in the form of matrix algebra and the matrix in the case of FEA is a square matrix of a very huge size it would be practically impossible to apply inverse to this global stiffness matrix to obtain results such as displacement and rotations. Hence certain numerical methods are adopted for solving the field variables, these numerical methods give approximate values of the unknowns generally at nodes. In the case of a continuous body, the entire body is meshed down into elements. This process of modelling a body by dividing it into an equivalent system of smaller bodies or units, finite elements, is called finite element analysis. These elements are connected at nodes and common surfaces. In FEA, instead of solving the problem as a single entity as in the case of classical methods, we prepare matrices for each element and combine all of them to obtain the global matrix and by employing numerical techniques we arrive at a solution.

The solution for problems in structural engineering refers to determining the displacements at each node using these values stresses within each element found out and making up the structure that is subjected to a certain set of boundary conditions and for a given loading. And the above-mentioned stressed and displacements are only in the case of standard explicit which is the category of modelling in Abaqus. Field variable change with changing elements and changing the category of modelling.

➢ *Brief History of Finite Element Methods:*

The modern development in the field of finite element method began in the Mid- 20th century. The developments especially in the field of structural engineering came to light by the works of Hrennikoff in 1941 and later by McHenry in 1943, who employed one-dimensional elements such as bars and beams for the generation of lattice and deriving the solution of stresses in continuous structures. These papers were published in 1943 but were not many recognized for many years to come. Another person named Courant proposed the solution of stresses in a variational form. Then he introduced piecewise interpolation functions also called shape functions for triangular elements, over triangular sub-regions (within the triangle),

Making up the whole region as a method to obtain approximate numerical solutions. Another new type of method in analysing the finite element was proposed by Levy "the flexibility or force method", and later in the mid-1950s, his work suggested that another method (the stiffness or displacement method) could be a promising alternative for use in analysing statically redundant aircraft structures. The latter became a boon and is being used widely. However, his equations were cumbersome to solve by hand due to the numerous elements present, and thus the method became popular only with the advent of the high-speed digital computer.

Argyris and Kelsey developed a matrix method of structural analysis by employing the energy principles in the same time period. The introduction of the energy principles helped solve the problem of how to prepare the stiffness matrices and subsequently the important role that energy principles would play in the finite element method. The first usage of energy principles in formulating the stiffness matrices for two-dimensional elements was made by Turner et al. later in that decade. In this paper, the derivation of the stiffness matrices for truss elements, beam elements, and two-dimensional triangular and rectangular elements in plane stress was detailed and further outlined the procedure commonly known as the direct stiffness method for obtaining the global structure stiffness matrix. The opportune moment of the development of the high-speed digital computer in the early 1950s and the work of Turner et al. at the similar timeline of stiffness equations being written in matrix form. The phrase finite 20 elements was coined by a person named Clough in the 1960s at the same time both triangular and rectangular elements were used for plane stress analysis.

Today, the developments in mainframe computers and the availability of powerful microcomputers have brought this

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

method within reach of students and are being implemented in common academic regulations.

➢ *General Description of Finite Element Method:*

In finite element analysis, a body is divided into several sub-divisions called finite elements. The elements as discussed above depend upon the mass of the kind of structure under analysis for example the element in the case of analysing a fluid mechanics problem will be very different from that of a solid mechanics problem and similarly in electromagnetic analysis. The variation of the field variable (like displacement) inside the element is not known and hence to find out the field variables inside the element we assume that the variation of the field variable inside a finite element can be approximated by a simple function, in general polynomial and trigonometric expansions. These functions also called Interpolating functions are defined in terms of the nodal field variables, generally displacements, at the nodes. By solving the field equations, which are generally in the form of matrix equations by employing numerical techniques like gauss elimination, we will obtain these field variables. Once, these are known the approximating functions or interpolating functions define the field variable with in the elements.

The process to solve the finite element analysis is by series of steps. The step-by-step procedure for both static and linear or nonlinear structural problem can be described as follows:

- *Step 1: Discretization of Structure or Meshing*

We arrive to this step after the modelling of the structure. In this step we divide the entire structure into finite but numerous elements. The nearness to solution depends upon the number of elements and the type of element used.

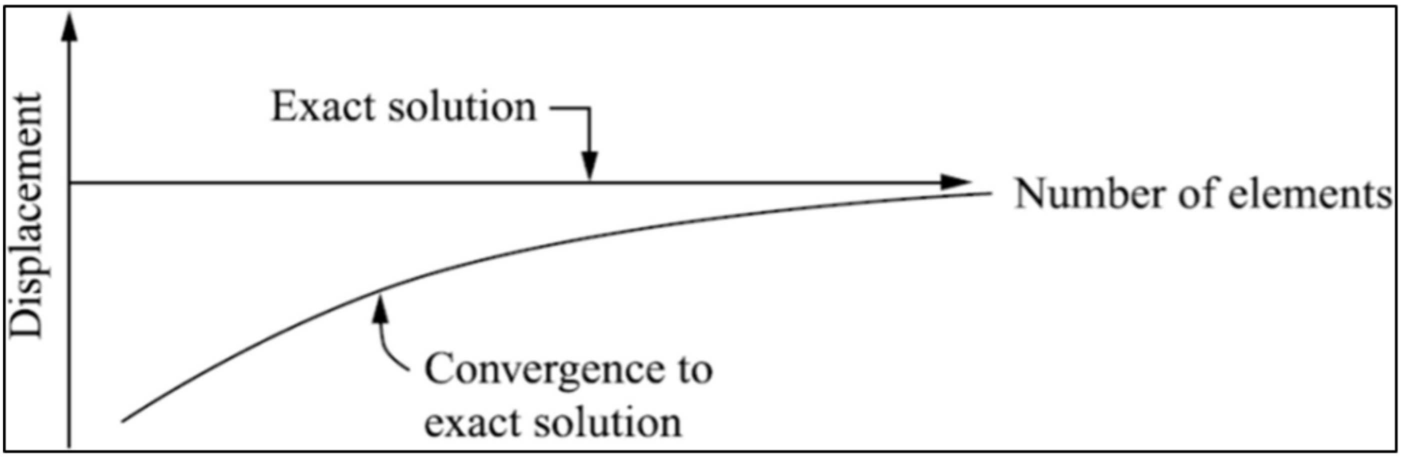

Fig 17 Convergence graph for the exact solutions

- *Step 2: Generating the proper interpolating functions:*

The displacement solution or field variables of a complex structure under any given loading conditions cannot be certain with absolute accuracy, instead, we assume some suitable solution within an element to approximate the unknown solution. The assumed solution must be simple from a computational point of view and it should satisfy certain convergence requirements. In general, the solution or the interpolation is taken in the form of a polynomial.

- *Step 3: Deriving the element stiffness matrices and load vectors The derivation is done in two forms*

✓ Strong Form
✓ Weak Form

In the case of the displacement model, the stiffness matrix and the load vector of the element can be derived by using either equilibrium conditions (strong form) or a suitable variation principle (weak form).

- *Step 4: Generation of global stiffness matrix:*

Now as the structure consists of finite but many elements and stiffness matrices for each element combining these matrices to form a giant global matrix is an attempt to bring the necessary interactions between these elements at least at the points of contact.

$$[K]\Phi=P$$

Here, [K] represents the assembled stiffness matrix, Φ represents nodal displacement vector or field variables and P is the force vector mainly consisting of moments and forces near nodes for the structure as a single entity for example fixed end moments in the case of stiffness matrix analysis.

- *Step 5: For finding the field variables:*

The equilibrium equations which we obtain after the formation of the global stiffness matrix should be further refined by taking into account the boundary conditions. In the case of hand calculations, we employ a technique called Statics condensation. After the incorporation of boundary conditions, the equilibrium equations can be expressed as:

$$[K]\Phi=P$$

After this step, the values of Φ are ascertained. Step 6: Computing the element strains and stresses Once we know the nodal displacements, the element strains and stresses can be computed by using the equations structural analysis equations. For example, strain displacement equation in case of stiffness matrix approach.

- *Step 6: Computing the element strains and stresses*

Once we know the nodal displacements, the element strains and stresses can be computed by using the equations structural analysis equations. For example, strain displacement equation in case of stiffness matrix approach.

➢ *Engineering Applications of Finite Element Method:*

The invention of the Finite element methods took place when the structural engineers were challenged to design a very economic design. In the case of the airline industry lesser payload means more economy in terms of fuel efficiency. Hence it is required to design sections which are more competitive than conservative sections. So now the values we obtain form the analysis should include the worst combinations of load cases and accurate analysis methodology. Here is where the finite element methodology came in handy for structural engineers, to incorporate all the possible loading conditions, especially in the field aerospace industry. However, the application of this methodology is not limited to the aerospace industry. Applications of the finite element method for various fields of engineering are as follow:

- Civil Engineering:

In Civil Engineering, FEM is used for various types of analysis. There are 4 types of analyses general linear static, linear dynamic, non-linear static, and nonlinear dynamic. Although there are hand calculation methods such as moment distribution, rotation coefficient and approximate methods there use is strictly restricted to the linear nature of the material. Analysis like non-linear static and non-linear dynamic are incorporated in software currently available software. General structural systems which are analysed using FEM are trusses, frames, folded plates, shell roofs, shear walls, bridges, and pre-stressed concrete structures. The inclusion of nonlinearity and performing some advanced analyses like a pushover analysis are done with ease using FEM. Software such as Etabs, Staad pro etc., are designed on the principles of FEM.

- *Nuclear energy:*

In this field, FEM is used for static analysis of nuclear pressure vessels and containment structures; and to find natural frequencies and stability of containment structures. FEM is used to find the response of reactor containment structures to dynamic loadings.

- *Nuclear design:*

FEM is used for stress analysis of pressure vessels, pistons, composite materials, and gears; and to find natural frequencies and stability of gears and machine tools. It is also used to solve crack and fracture problems under dynamic loads.

- *Biomedical Engineering:*

In this field, the analysis is done on elements inside the body such as bones, eyeballs, sockets etc. And the analysis of implants such as stunts in the heart are constantly subjected to forces and should be compatible with the body itself. It is used for impact analysis of the skull and to study the dynamics of anatomical structures. FEA is also employed in the field of archaeology for ascertaining the impact on the skull of the remains etc.

- *Aeronautical Engineering:*

In this field of engineering FEM is used for the analysis of aircraft wings, fuselages, fins, rockets, spacecraft, and missile structures; and to find natural frequencies of aircraft, rocket, spacecraft, and missile structures. It is used to find the response of aircraft structures to accidental loads and periodic loads.

➢ *Advantages of Finite Element Methods:*

The finite element method has several advantages that made it applicable to various fields of engineering. The following are the advantages of the finite element analysis:

- Modelling analysing irregular-shaped bodies becomes quite easy.
- Handling general load conditions.
- The components are analysed individually due to the generation of different sets of different components.
- Can handle unlimited numbers and kinds of boundary conditions

- The variations in sizes of the elements can be done by facilitating to use of small elements whenever necessary.
- We can include dynamic effects.

➤ *Disadvantages of Finite Element Method:*

The disadvantages of the finite element method are:

- To model a structure one needs to get familiar with the software and require a bit of experience in the area of modelling.
- The cost of such software is high due to its sophisticated design and acquiring the software is a difficult task since only a few outlets are available to procure and there is always a risk of falling for illicit online predators in case of online procurement.
- A computer with advanced analytical capabilities is required to run the program.
- The input (defining the problem) and the output (reading the results) becomes hard if not properly experienced.
- If the topography of the shape in not in the intended form the mesh formation can be un-recommended and hence the solution will also be away from the nearness of the exact solution.
- If the coding (generation of the FEM software) itself is wrong there is no way of identifying the errors except for engineering judgment.

➤ *About Ansys:*

Ansys, Inc. is an American company based in Canonsburg, Pennsylvania. It develops and markets CAE/Multiphysics engineering simulation software for product design, testing, and operation and offers its products and services to customers worldwide.

Ansys was founded in 1970 by John Swanson, who sold his interest in the company to venture capitalists in 1993. Ansys went public on NASDAQ in 1996. In the 2000s, the company acquired numerous other engineering design companies, obtaining additional technology for fluid dynamics, electronics design, and physics analysis. Ansys became a component of the NASDAQ-100 index on December 23, 2019. Many products are integrated into ANSYS software they are,

- Ansys Mechanical
- Ansys LS-DYNA
- Ansys Fluent
- Ansys CFX
- Ansys optiS LANG
- Ansys Sherlock and more

➤ *Advantages of Ansys Software:*

- ANSYS software can import all types of CAD geometries (3D and 2D) from different drafting software and can perform simulations.
- ANSYS has inbuilt drafting software like design modeller and space claim which allows it to make the flow of work smoother.
- ANSYS has the ability to perform advanced engineering simulations realistically and accurately with its variety of contact algorithms, time- dependent simulations.
- ANSYS also can integrate various physics combinedly into a single platform and performs the analysis. Just like integrating structural analysis with thermal and fluid flow analysis with both thermal and structural.
- ANSYS can optimise various features like boundary conditions, and geometric design, and can analyse the behaviour of the product under various criteria.

➤ *Ansys Sequence to Solution:*

Like any other analytical solution process, the analysis using the numerical techniques is the same except for the part of meshing in the software. Below Describes the steps involved in the process of obtaining results from ANSYS:

- *Solid Model Geometry: There are three ways to create a model in any finite element program for solid modelling.*

✓ Direct (manual) generation
✓ Specify the location of nodes
✓ Define which nodes make up an element
✓ Used for simple problems that can be modelled with line elements (links, beams, pipes)
✓ Not recommended for complex solid structures.
✓ Importing Geometry
✓ Geometry created in CAD systems like CATIA
✓ Saved as an import file as an STP file
✓ Inaccuracies occur during the import, and the model may not import correctly.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

- ✓ Solid Modelling approach
- ✓ The model is created from sile primitives (rectangles, Circles, Polygons, Blocks, Cylinders etc.,)
- ✓ Boolean operations are used to combine primitives.

However, when creating a solid model that contains over 20000 nodes, this approach is not recommended. Using a CAD program such as Autodesk Inventor to create the solid model was also investigated. Autodesk Inventor had a very good CAD Capability compared to creating the model in the ANSYS CAD environment.

The solid modelling approach was used to create the Block type of machine foundation. Whereas, Creating the machine foundation using solid geometry modelling is a bit complex because of its geometry which is complex compared to the block foundation modelled using solid geometry as shown in figure 4.2. Therefore, the frame type of the machine foundation is modelled in the CATIA and later imported into ANSYS as an STP file. The CATIA model of the machine foundation is shown in figure 18 Figure 19 shows the imported model of the frame foundation to ANSYS.

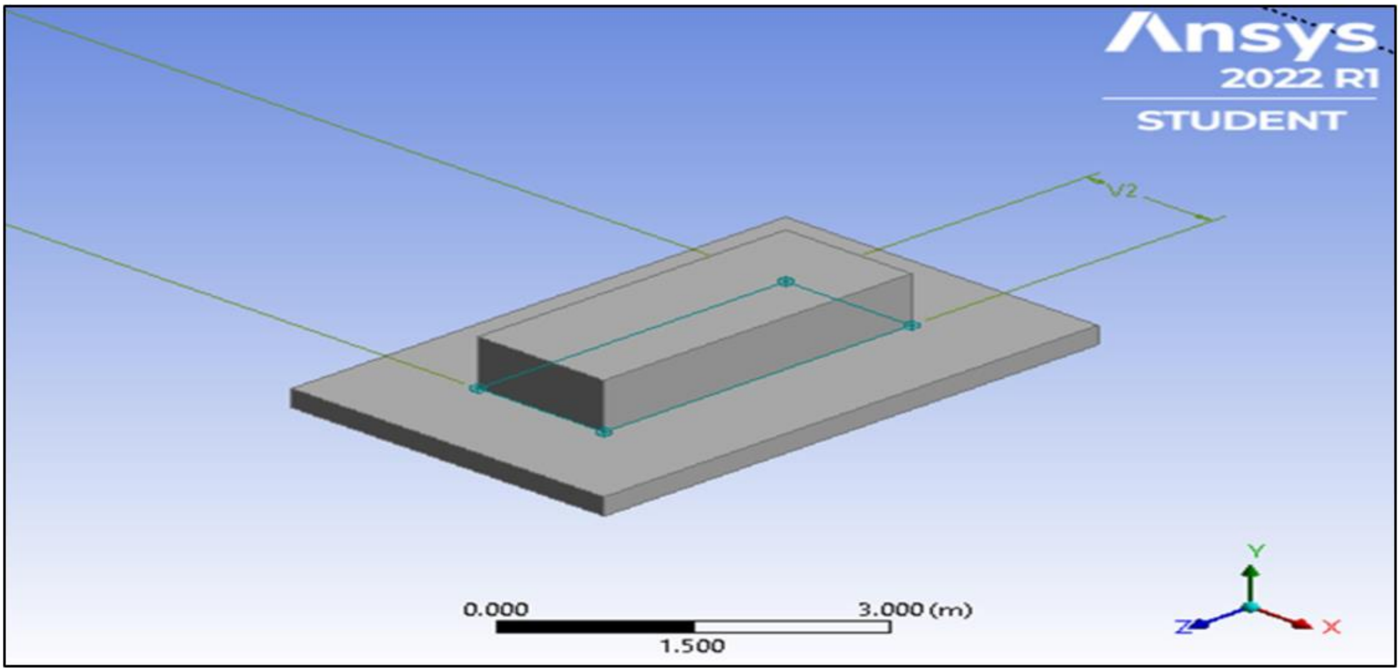

Fig 18 Solid geometry modelled Block Foundation

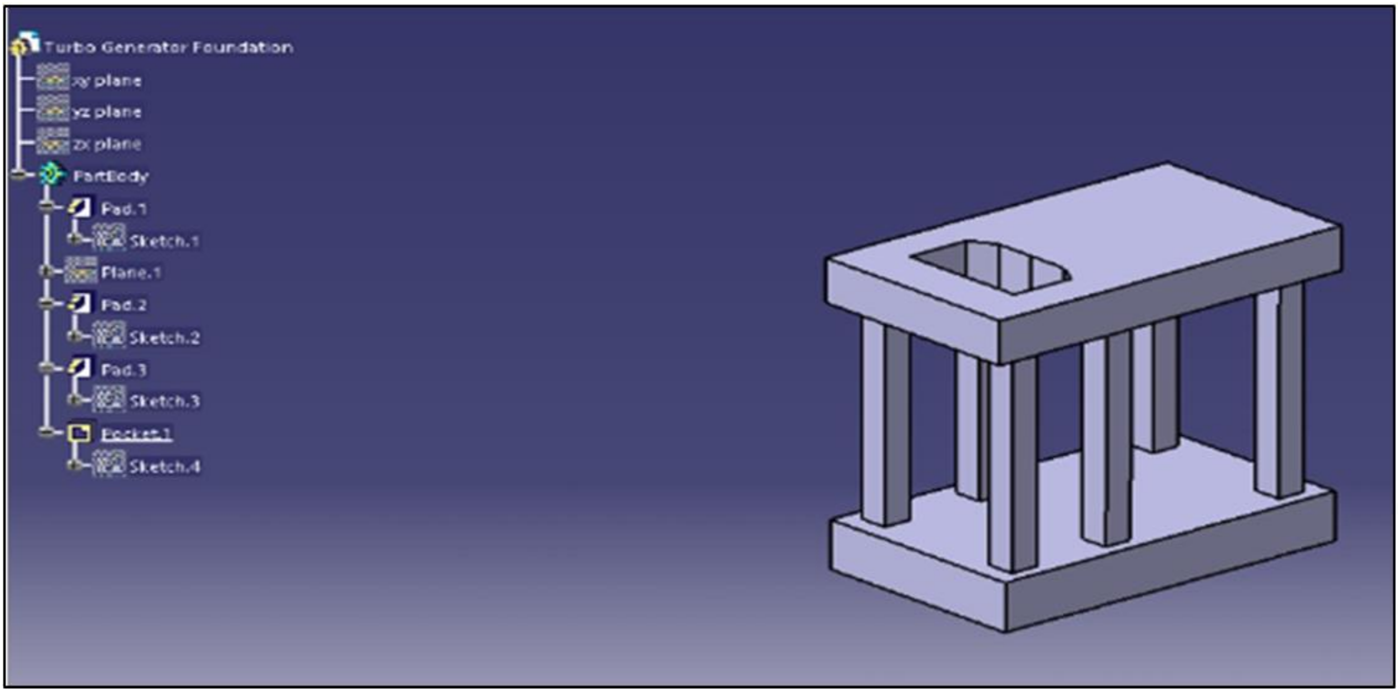

Fig 19 Frame foundation modelled in CATIA

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

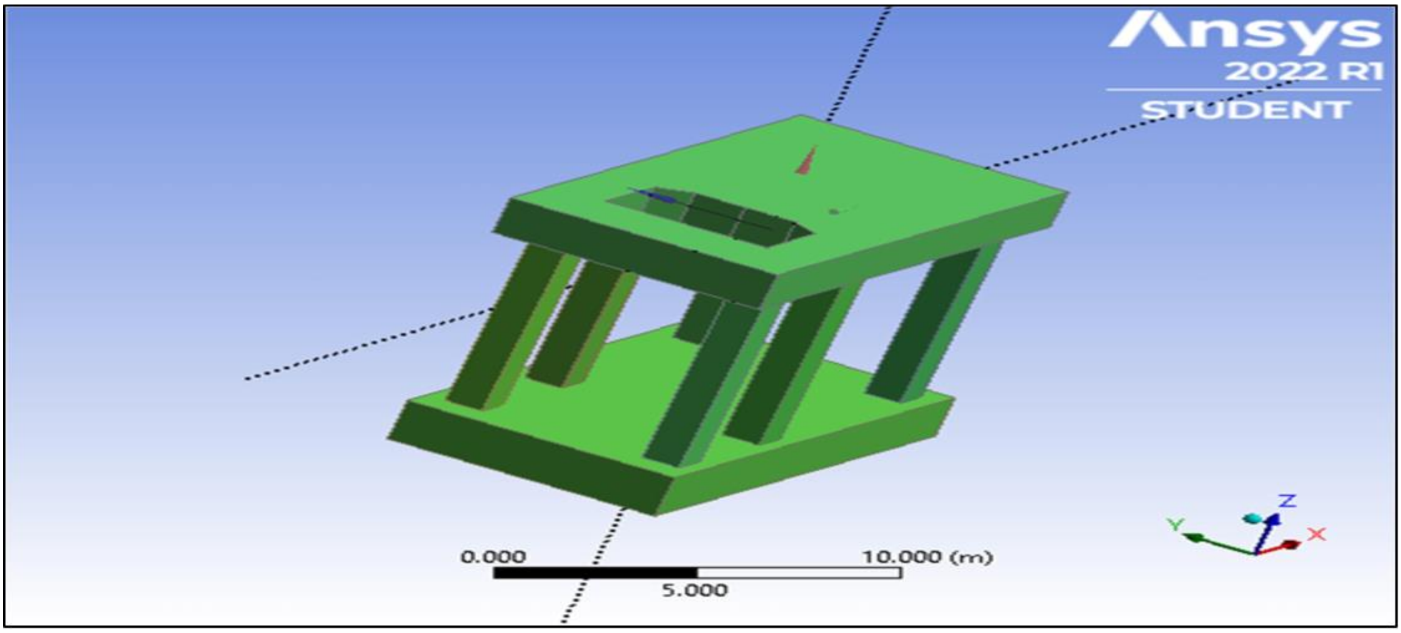

Fig 20 Imported frame foundation CATIA model into ANSYS

Define Material properties: ANSYS has an inbuilt list of various materials with their properties like density, and modulus of elasticity predefined under the module engineering data sources. We can choose the material of our type and continue the remaining process.

Both type of Machine foundations are assigned to have the same material property i.e., Concrete of density 2500kg/m3.

➢ *Finite Element Discretization or Meshing in ANSYS:*

Meshing is the process used to 'fill' the solid model with nodes and elements, i.e, to create the FEA model. Remember, you need nodes and elements for the finite element solution, not just the solid model. The solid model in CAD does not participate in the finite element solution.

Meshing in ANSYS can be applied annually or automatically. The element type selected(linear vs. Tetrahedral)and the mesh size can affect the accuracy of the results of the analysis. In automatic meshing, ANSYS automatically chooses a meshing size based on the shape of the model. This resulted in more elements than permitted by the University High option of ANSYS. Manual meshing allows the user to define the maximum size of the elements.

➢ *Meshing Methods:*

There are two main meshing methods: Free Meshing and Mapped meshing.

- *Free Meshing:*

✓ Has no element shape restrictions
✓ The mesh does not follow any pattern
✓ Suitable for complex shaped areas and volumes
✓ Volume meshes consist of high order tetrahedral(10 nodes), large dof

➢ *Mapped mesh:*

- Restricts element shapes to quadrilaterals (areas) and hexahedral (volume)
- Typically has a regular pattern with obvious rows of elements
- Suitable only for 'regular' shapes such as rectangles and bricks.

The finite element meshing of the block and frame type of machine foundations are as shown in figures 4.5, 4.6 & 4.7 In figure 4.5, we can see that the ANSYS software has automatically meshed the model according to its shape. Whereas, figure 4.6 shows the meshing done manually on the block type of machine foundation with a very fine mesh of element size of 0.001.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

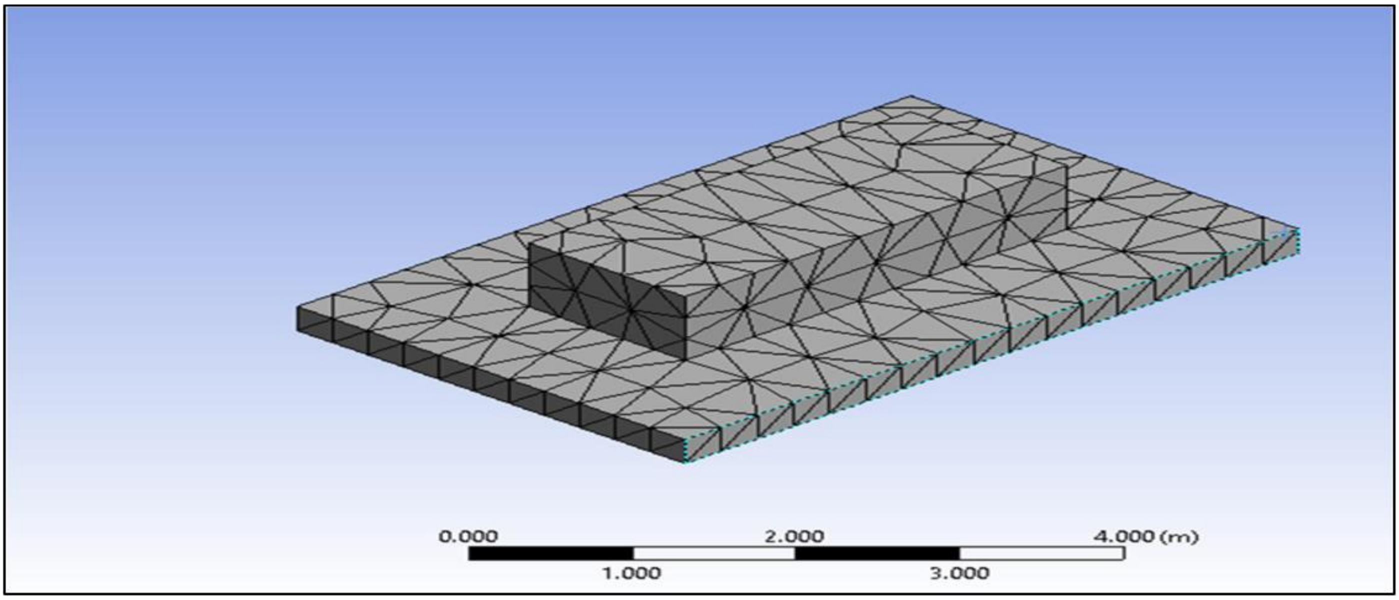

Fig 21 Automated mesh on Block foundation in ANSYS

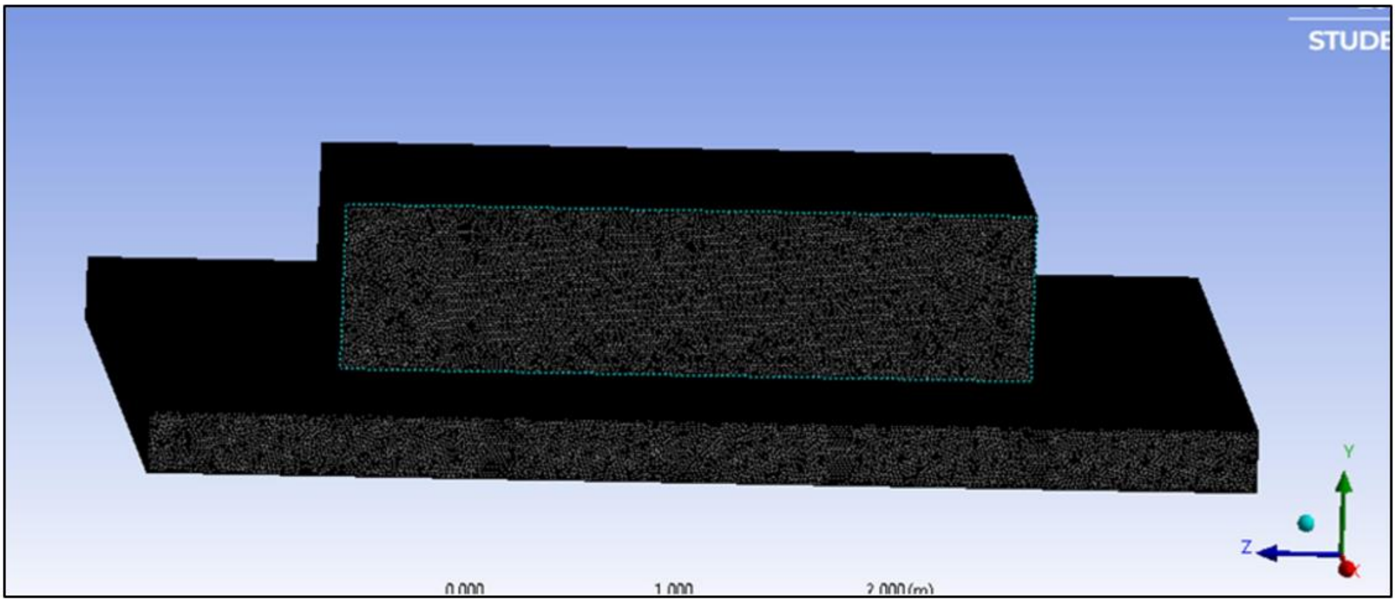

Fig 22 Manually meshed Block type of machine foundation

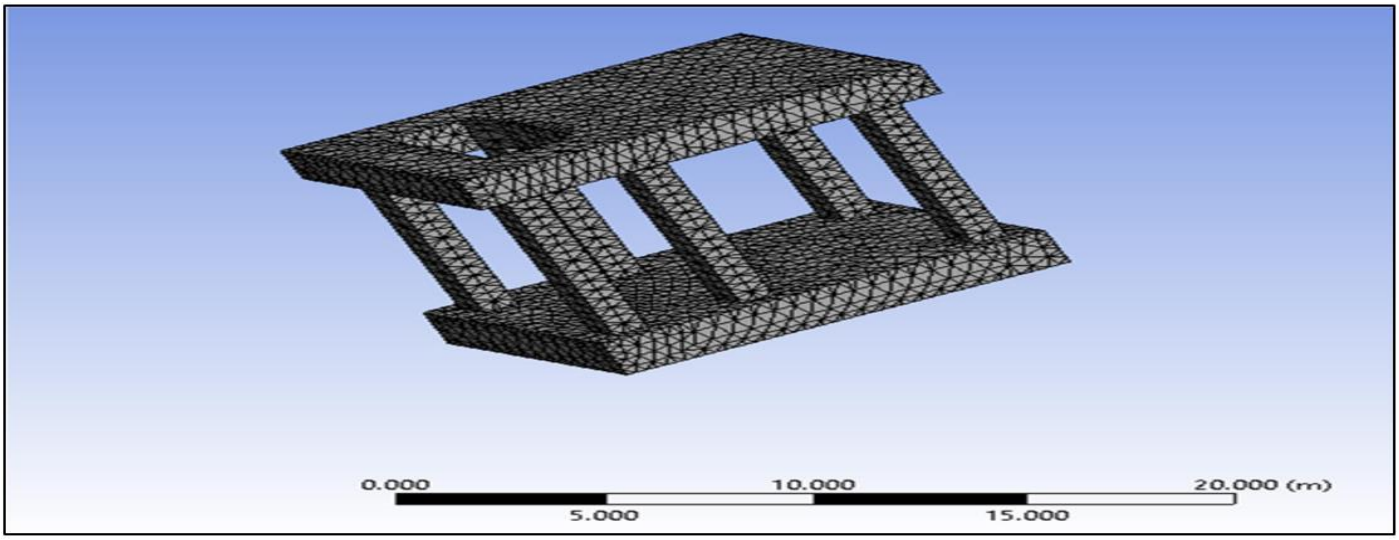

Fig 23 Finely meshed frame type of machine foundation

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

➢ *Applying Loads:*

The word loads in ANSYS includes boundary conditions (Constraints, Supports or boundary field specifications) as well as other externally and internally applied loads. Loads are divided into the following categories:

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

Most of these loads can be applied either to the solid model (key points, lines, and areas) or the finite element model (nodes and elements). The program uses the concept of time in the transient analysis as well as static (steady state) analyses. In transient analysis, time represents actual time, in seconds, in minutes, and in hours. In a steady state or static analysis, time simply acts as a counter to identify load steps and sub-steps. Firstly, fixed support is assigned to both type of foundations as shown in figures 4.8 & 4.9. The blue-coloured surfaces indicate the assigned fixed supports to the foundations.

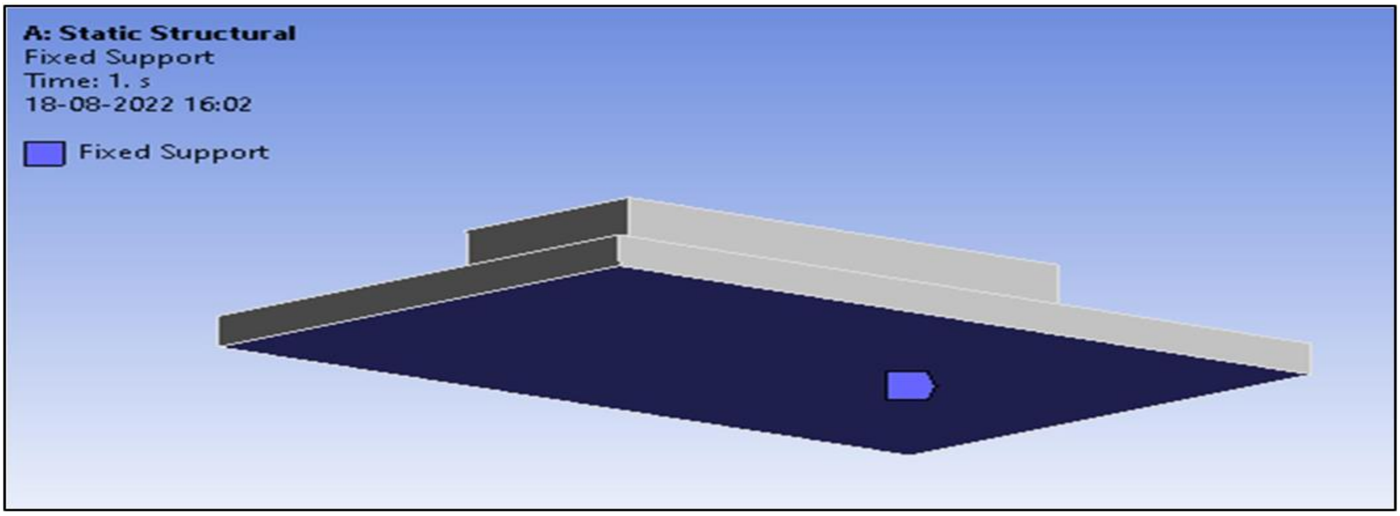

Fig 24 Fixed Support of Block foundation

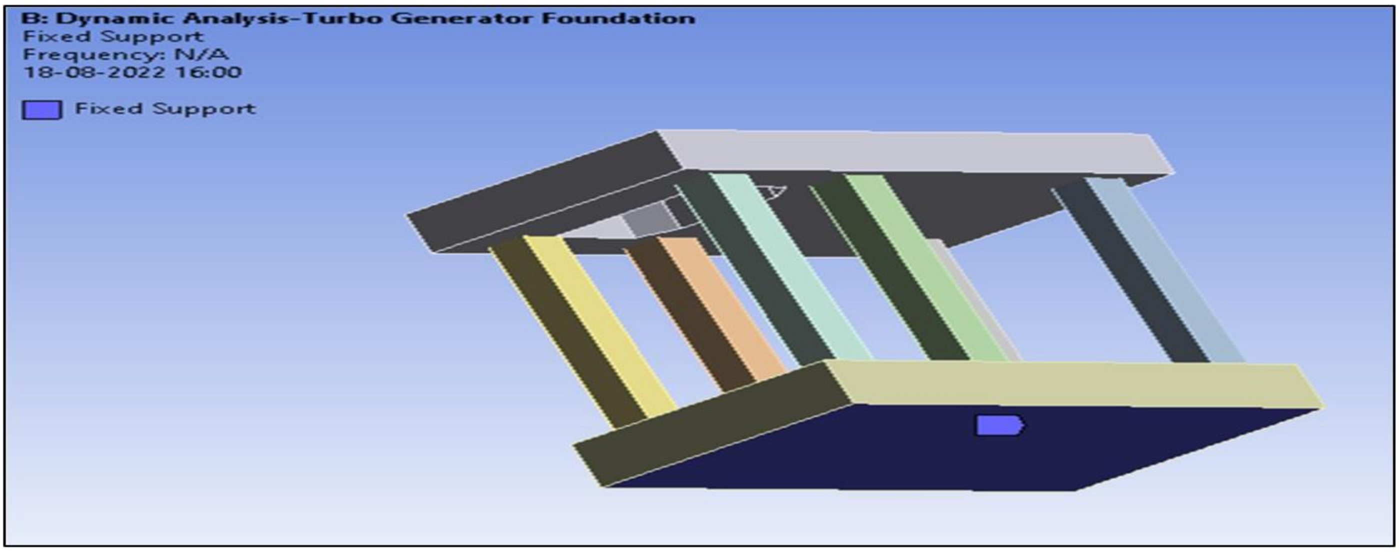

Fig 25 Fixed support for Frame type of foundation

➢ *Applying loads on block foundations:*

Three remote forces are applied on the block foundation according to section 3.1.1.d, where all the forces are calculated according to the code and considering all the unbalanced forces from the machine. Below figures 26,27,28 are about assigning loads on the block foundation.

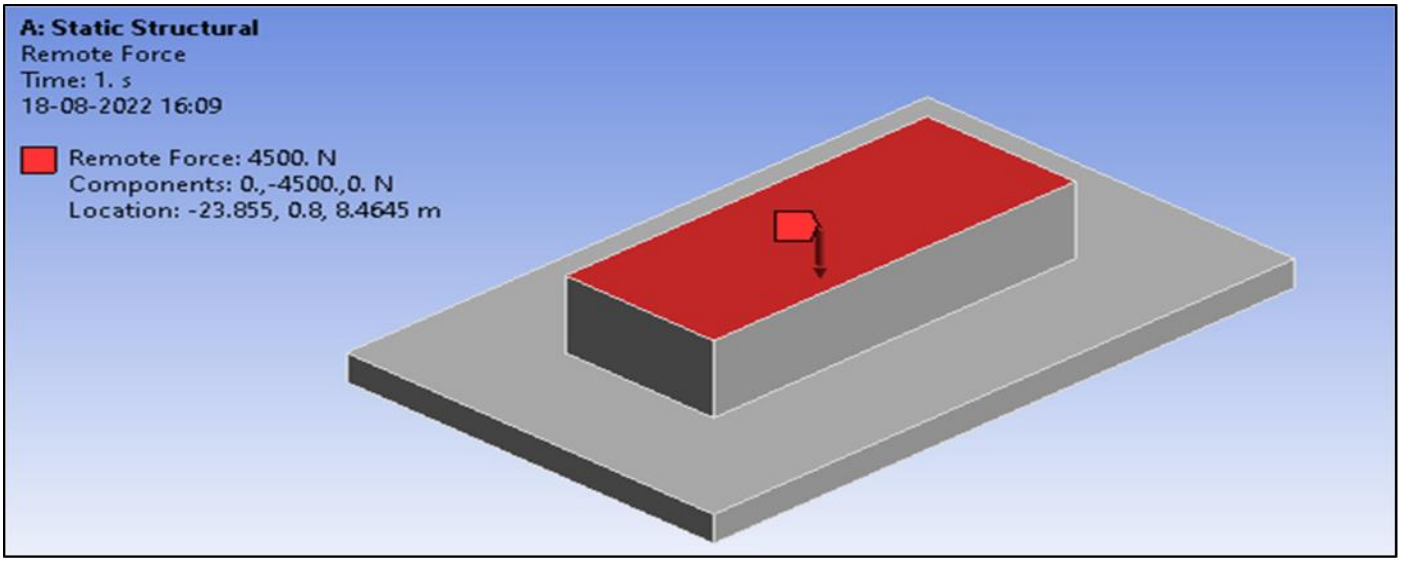

Fig 26 Remote force(Fy) on block foundation

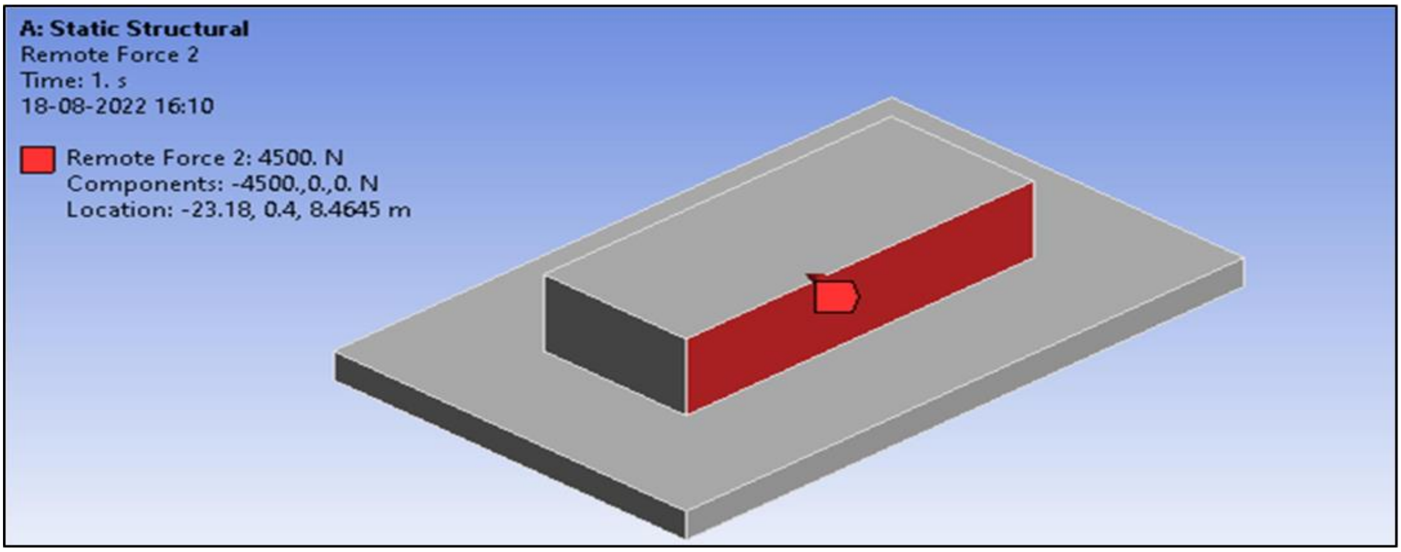

Fig 27 Remote force (Fx) on block foundation

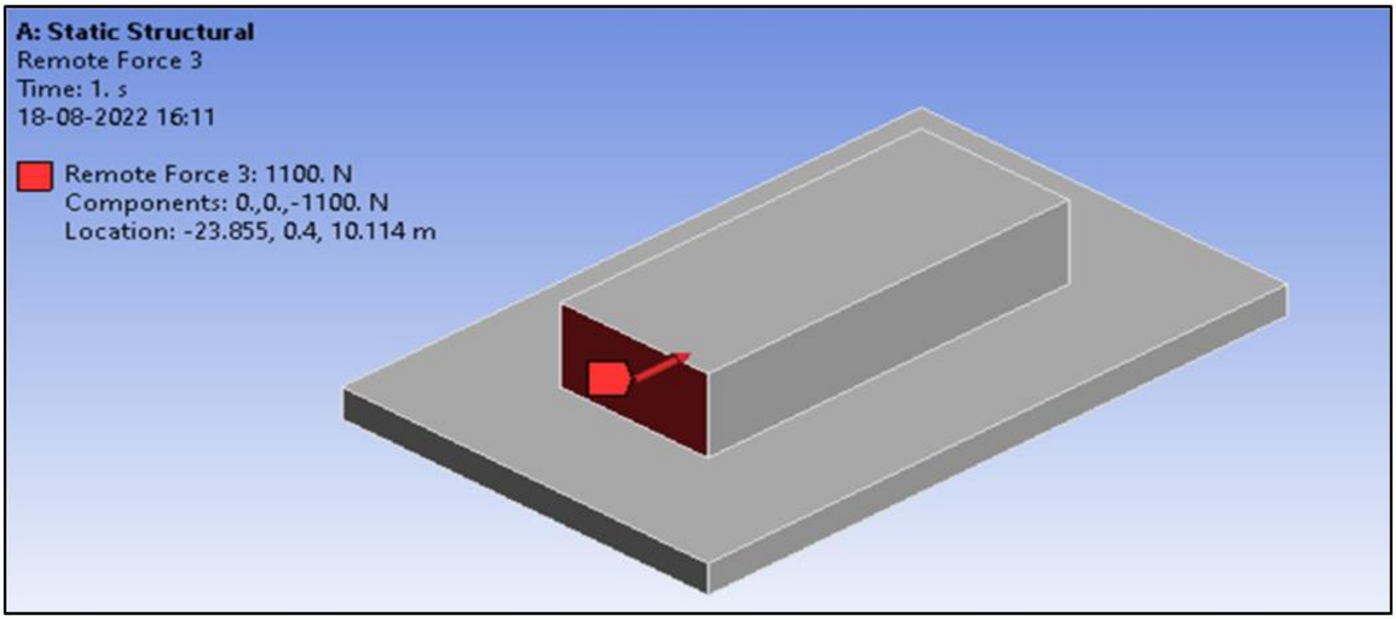

Fig 28 Remote force (Fz) on block foundation

➢ *Applying loads on frame foundations:*

Same as the block foundations, three remote forces all in the Y-direction are acted upon the foundation as described in section 3. Figures 29&30 shows the application of remote forces on three frames as described in section 3. on the frame foundation.

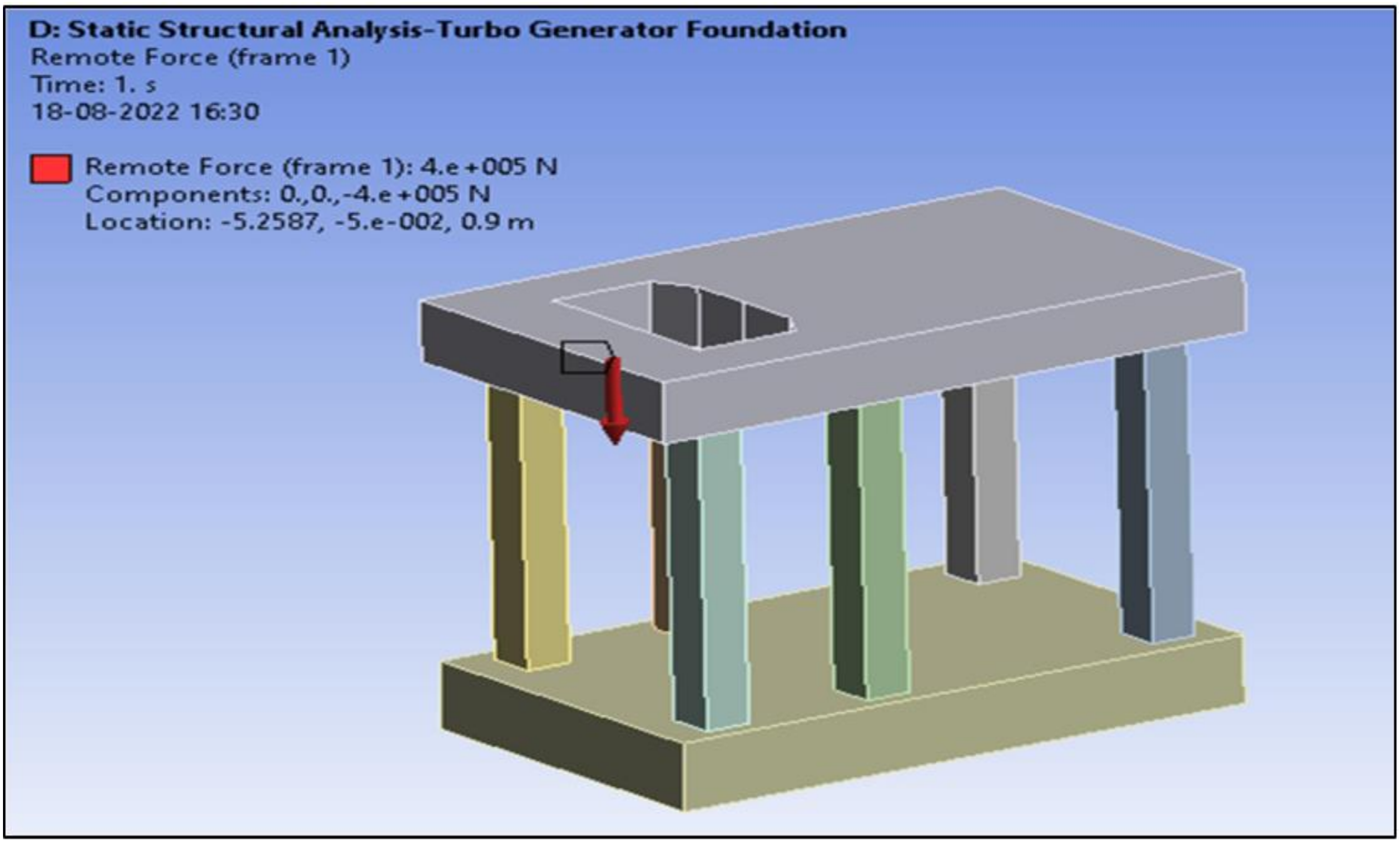

Fig 29 Remote force on frame 1 of Frame Foundation

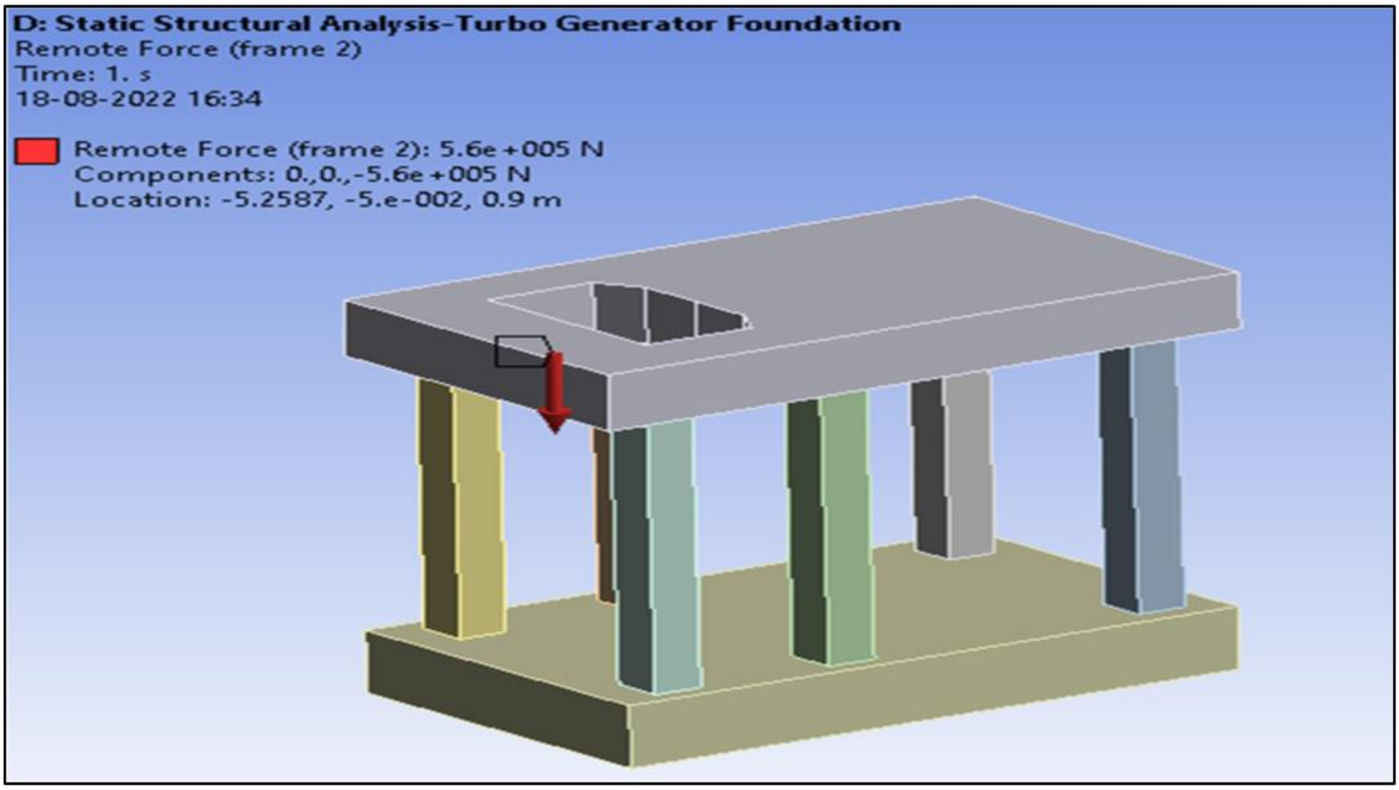

Fig 30 Remote force on frame 2 of frame foundation

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

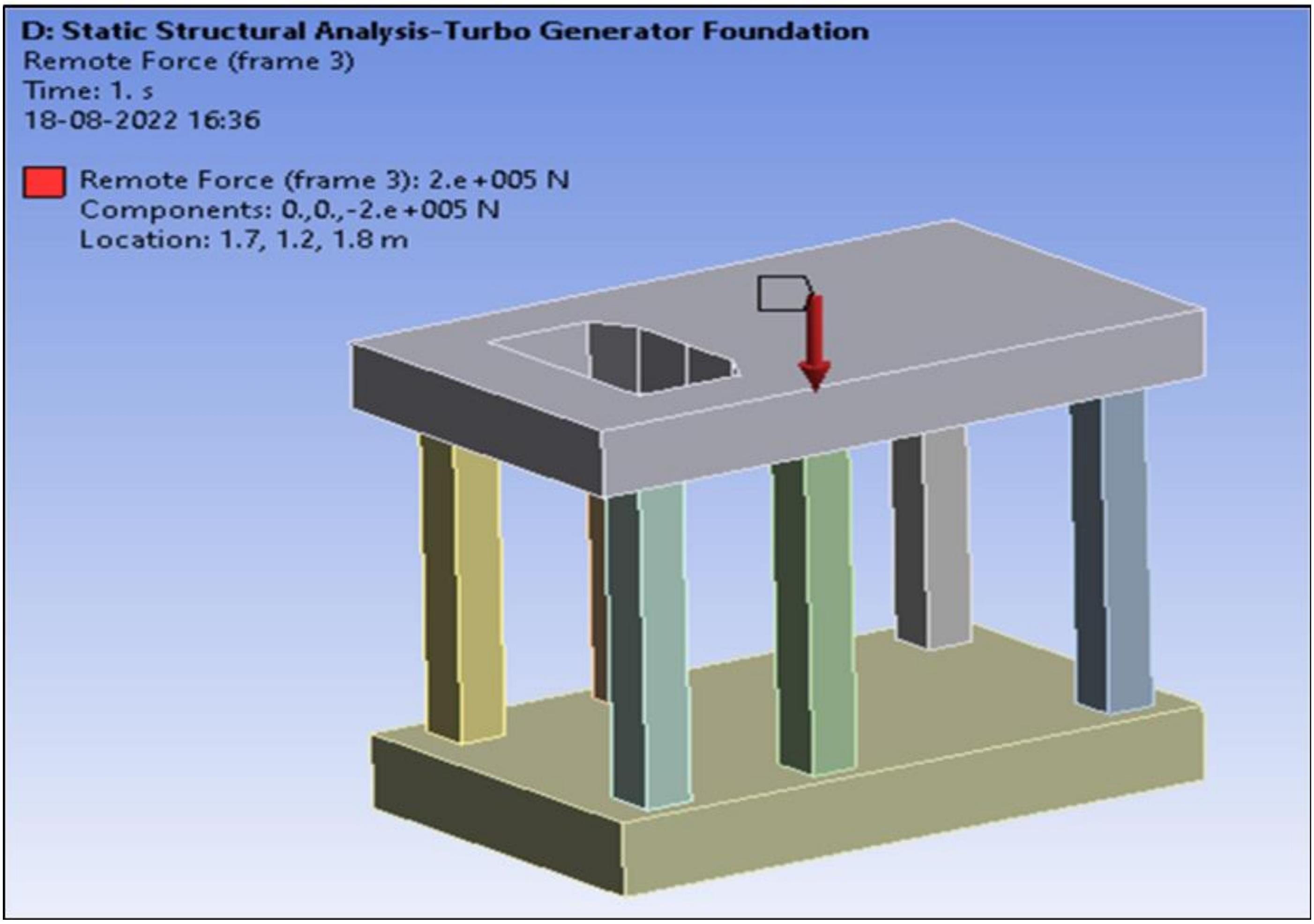

Fig 31 Remote force on frame 3 of Frame foundation

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER FIVE
OPTIMIZATION

➢ General

Optimal design of any structure has advantages as the structural design engineers focus on several factors that are to be satisfied for the functional requirements and economically feasible model. Thus optimisation of trusses provides one such solution where all the parameters in the design are combined with optimisation to develop a design that meets all the required criteria.

- Tremendous resources are consumed in the architecture, engineering and construction industries, thus sustainability and efficiency have been a prime concern recently.
- Steel trusses are widely used in industrial buildings and bridges. It is a necessity to consider both structural and economic requirements.
- Truss optimization is an important class of engineering problems, with better solutions allowing for reduced material use and construction cost.
- The consideration of dynamic effects is of great relevance for designing structures subjected to earthquakes or structures subjected to random vibrations.

➢ *Optimization*

The rise of research applications in developing algorithms for optimisation has paved the way to utilise those effective techniques in structural problems. The optimisation is importance in finding economical solutions while there are limited resources. The primary goal of designing any structure is in terms of strength, serviceability, safety and economic considerations. Optimisation methods can be broadly classified into classical methods and metaheuristic optimisation. The classical methods which are also called analytical methods to obtain the maximum and minimum of a given problem are unsuitable for large-scale applications. Thus metaheuristic algorithms are used for tackling complex problems.

➢ *Metaheuristic Optimization*

Practical optimization problems are non-linear and also multi-modal. The objectives and the constraints might conflict with each other as the complexity grows. Sometimes, an optimum may not even exist. Even if there is an optimum, finding the optimum becomes a tedious task to accomplish. In cases where classical optimisation methods pose issues in finding derivatives, metaheuristics with their very random nature of generating solutions and slowly converging to the optimum are computationally feasible. There are several metaheuristic algorithms available which include Genetic Algorithm, Particle Swarm Optimisation, Ant Colony Optimisation, Teaching Learning Based Algorithm,

Simulated Annealing, Cuckoo Search Algorithm, Differential Evolution Algorithm, Heat Transfer Search Algorithm and others. All these algorithms have a unique approach to producing optimal solutions. Most of these algorithms are inspired by nature.

➢ *Topology Optimization:*

This type of optimization helps to optimize the stiffness by reducing the weight of the system. It is faster, and simpler compared to the parameterised geometry studies/ Design Assessment analyses. This is especially useful for additive manufacturing (i.e., the process of creating an object by building it one layer by another layer as 3D printing). It also requires simplification in CAD.

➢ *Limitations of Topology Optimization:*

- *Element types:*

  All elements which are not a shell, solid or plane are ignored.

- *Constraint/ Objective function:*

✓ The compliance objective is not compatible with both force-based and displacement-based loading.
✓ Extrusion work only for hexagonal mesh.
✓ Minimum member size requires member density 4 times finer than the specified member size.

➢ *Topology Optimization Procedure:*

Drag the topology optimization module on to the analysis system as shown in the figure 32

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

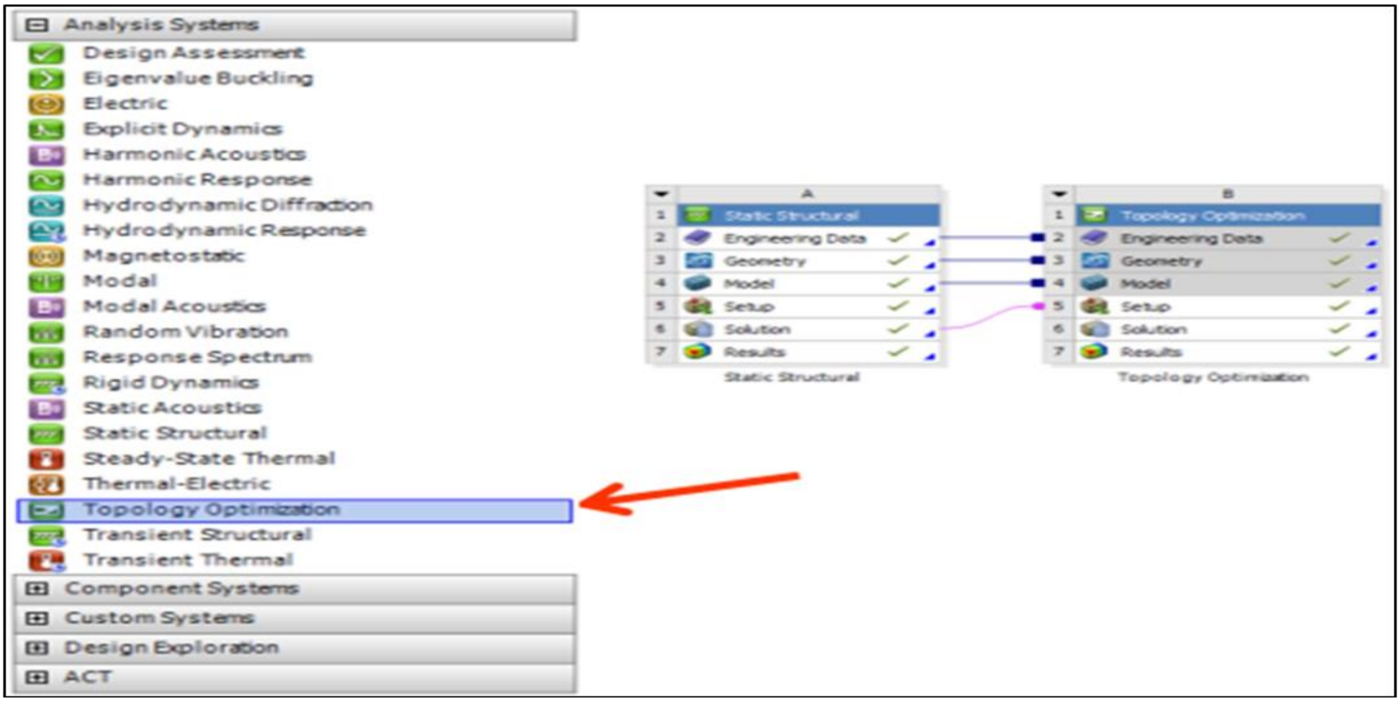

Fig 32 Topology optimization analysis module

All linked analyses must be linked to the topology optimization system as shown in figure 33

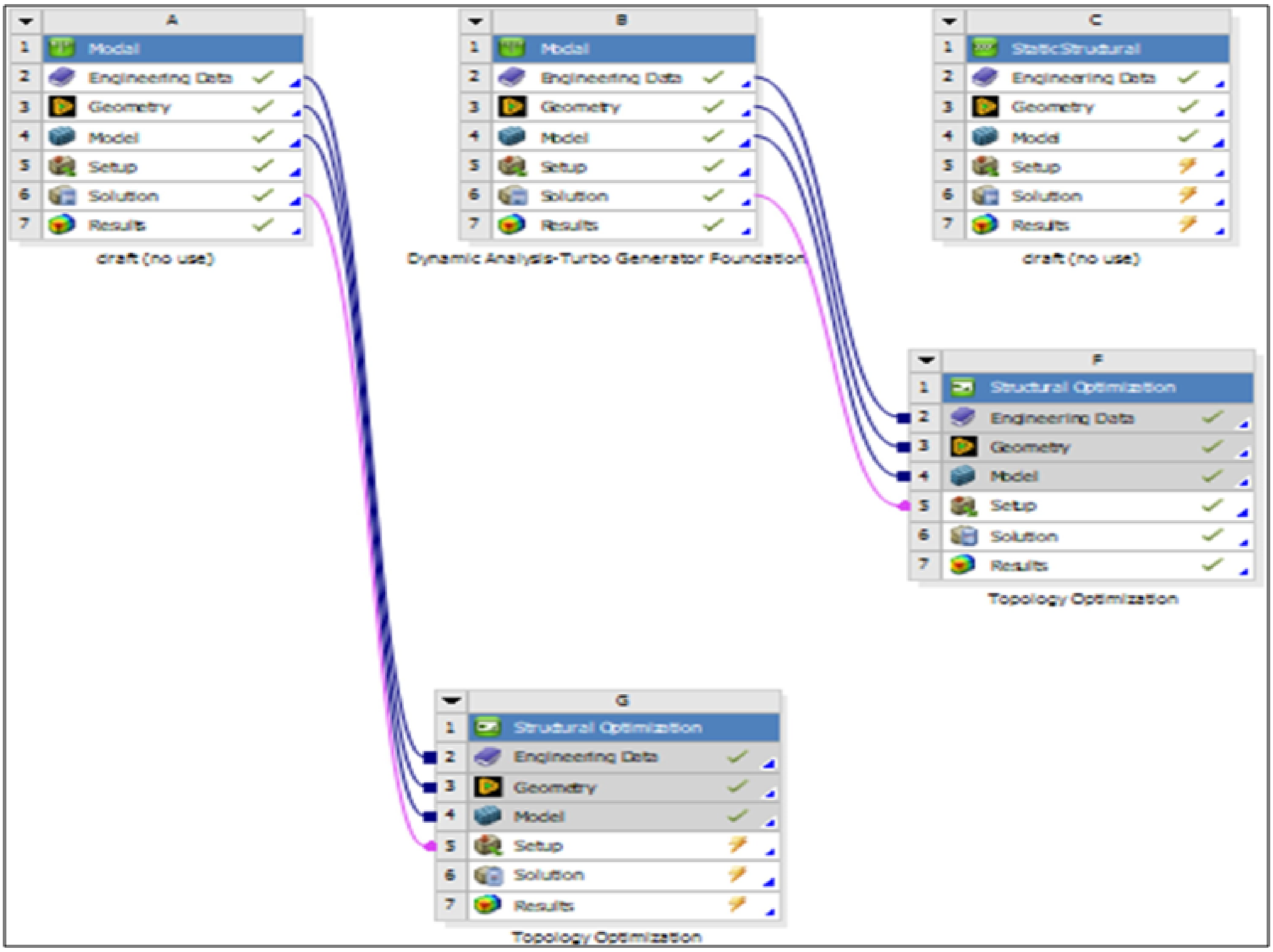

Fig 33 Linking all analysis systems to topology optimization.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

- *Optimization region:*

Select bodies which are to be optimized, If required we can select regions which should be remained unchanged as exclusion regions. In this investigation, we have chosen columns as the optimization region as shown in figure 34

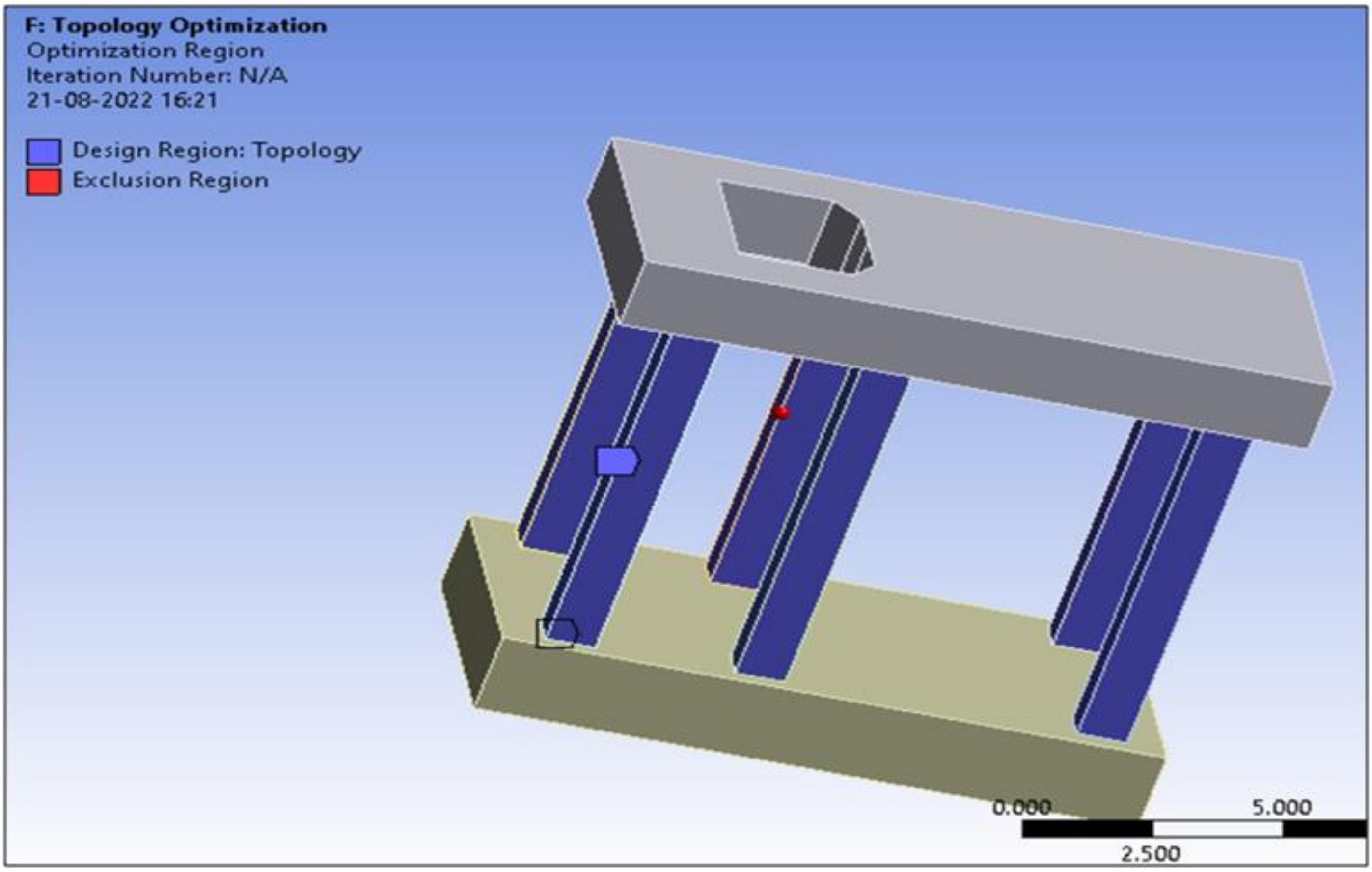

Fig 34 Selection of Optimizing regions.

- *Objective:*

The default objective function is minimisation compliance. Other objectives include maximizing mass/ volume. We can use multiple objective functions in a single system. The objective function is as shown in figure 34.

- *Response Constraints:*

The default response constraint is 50% mass retention, which applies the same to the volume type of constraint also. After the completion of specified iterations for optimization, the final optimized model is presented with animated results to watch the real-time optimization process.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER SIX
RESULTS AND DISCUSSIONS

This chapter deals with the presentation of results obtained from finite element analysis of machine foundations and a brief discussion on the behaviour of block and frame types of foundations. The results obtained due to static and dynamic analysis of machine foundations are presented below.

➢ *Static Analysis of Machine Foundations:*

For the static analysis of machine foundations two types of machine foundations are modelled using 'ANSYS workbench 2022' are:

- Block type of machine foundation
- Frame type of machine foundation

Both these models are subjected to the same meshing and loading conditions. The same analysis procedure is followed for two of them. Stresses and deflections are determined for both foundations under static loading. The maximum stresses and deflection induced in both types of machine foundations under static loads are shown in table 6.1.

Table 12 Representing Maximum deflection and stresses induced in the foundations.

Type of Machine Foundation	Block Type of Machine Foundation	Frame Type of Machine Foundation
Maximum Stress (MPa)	2.47×10^{-3}	6.3
Maximum Deflection (mm)	1.096×10^{-4}	1.8291×10^{-8}

➢ *Results of Stresses and Defections of block type of machine foundation under static loading:*

The stresses and deflections induced in the Block type of machine foundation are graphically represented by figures 35 & 36 respectively.

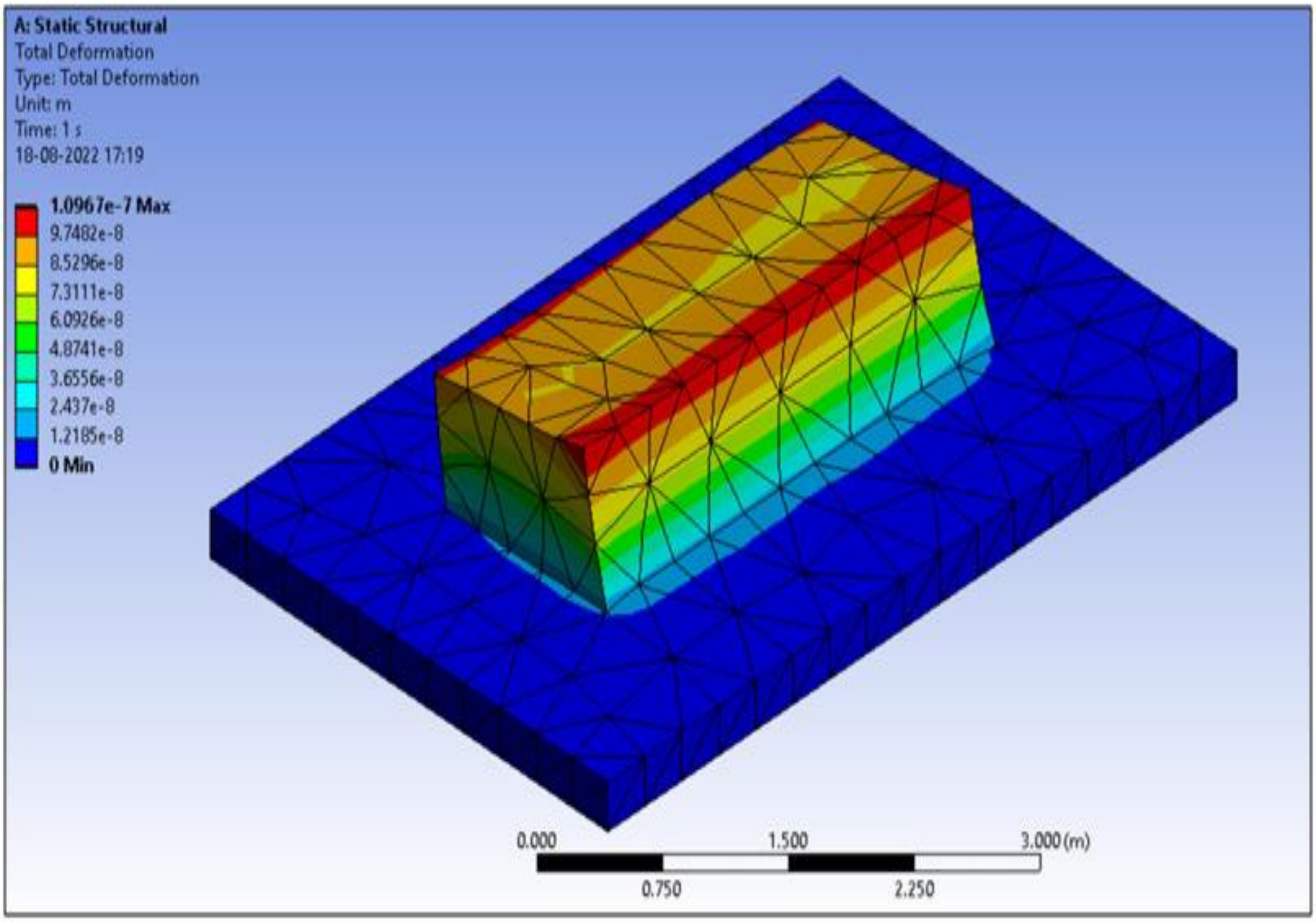

Fig 35 FEA model of Block foundation representing the total deformation (m)

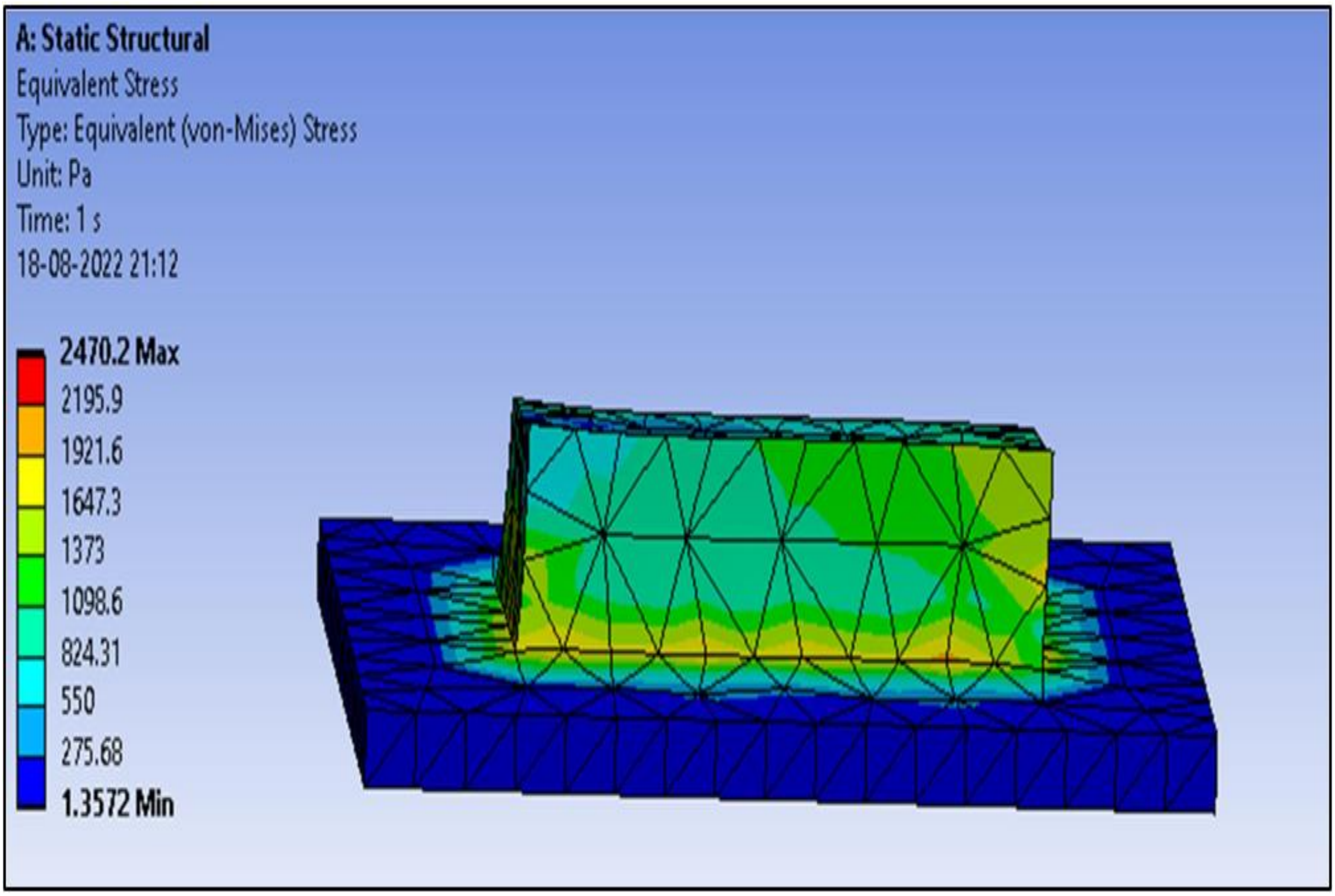

Fig 36 FEA model of Block foundation representing the Equivalent stresses (Pa)

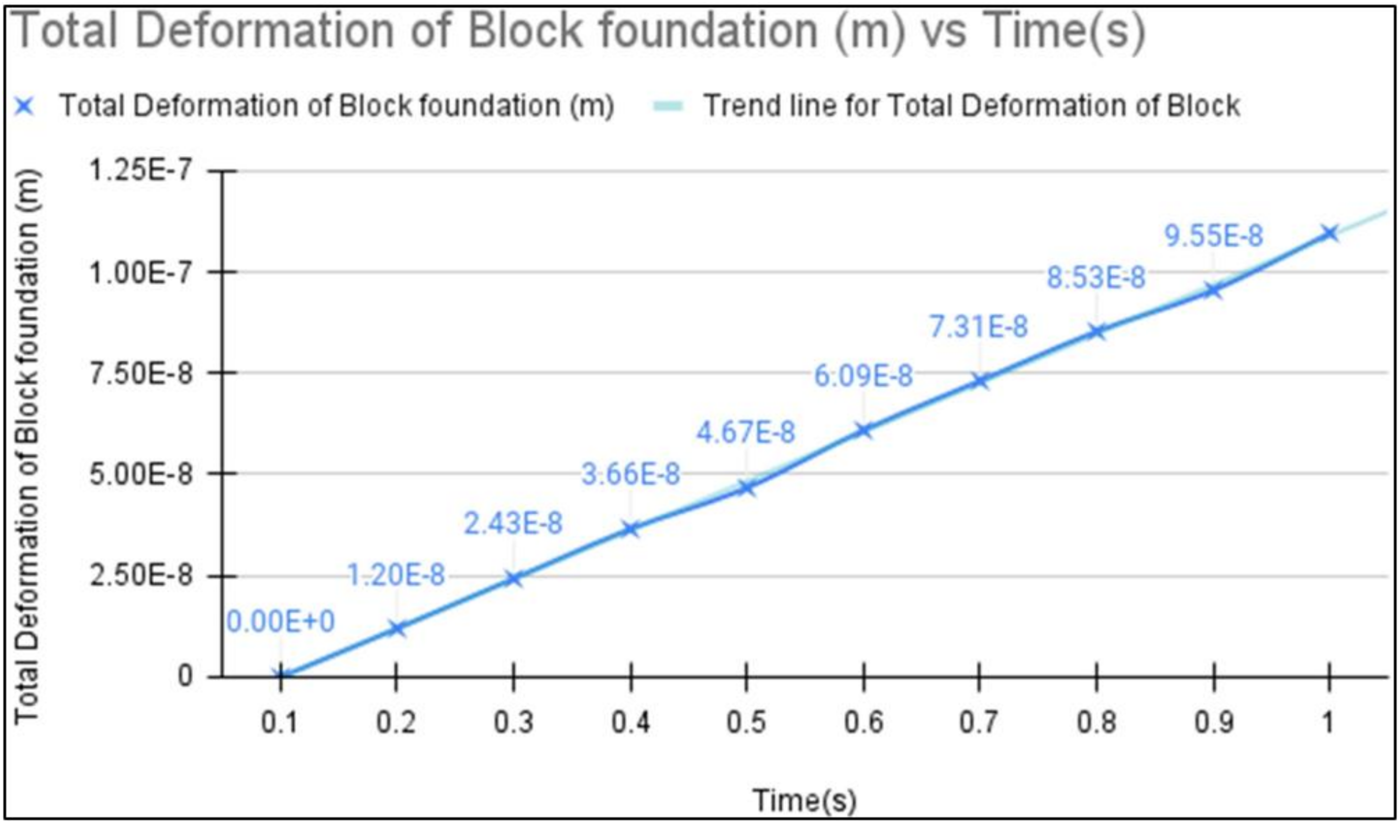

Fig 37 Graphical representation of the variation of deformation in block foundation w.r.t time.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

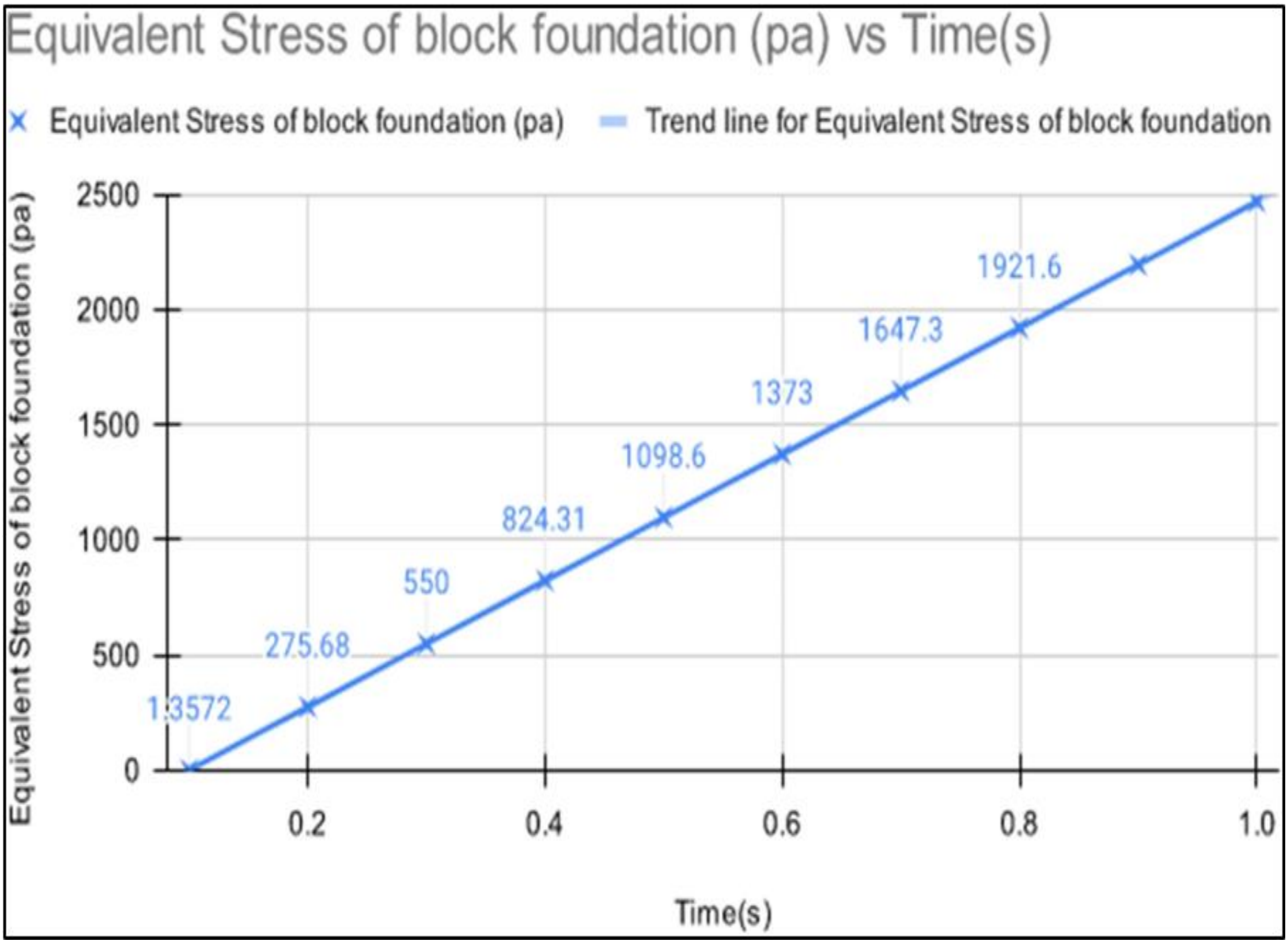

Fig 38 Graphical representation of Equivalent stress in block foundation w.r.t time

Table 13 Total Deformation and Equivalent stresses of Block foundation under static loads.

Time(s)	Total Deformation of Block foundation (m)	Equivalent Stress of block foundation (pa)
0.1	0	1.38E-06
0.2	1.2E-08	2.75E-05
0.3	2.43E-08	5.50E-05
0.4	3.66E-08	8.24E-05
0.5	4.67E-08	1.10E-03
0.6	6.09E-08	1.37E-03
0.7	7.31E-08	1.65E-03
0.8	8.53E-08	1.92E-03
0.9	9.55E-08	2.20E-03
1	1.10E-07	2.40E-03

Figure 35 represents the variation of total deformation of the block foundation with a minimum value of 0 (m) to a maximum deformation of 1.096e-7 (m) foundation under static loads. Figure 36 shows the distribution of Equivalent stress induced in the block foundation under static loads.

Figure 37 shows the graphical variation of total deformation of the block foundation concerning time and figure 38 represents the variation of equivalent stress induced in the foundation concerning time. Both graphs show a linear variation with respect to time, which represents the linear elastic nature of the block foundation.

➢ *Results of stresses and deflections of frame type of machine foundation under static loading:*

The stresses and deflections induced in the Frame type of machine foundation are graphically represented by figures 39 & 40 respectively.

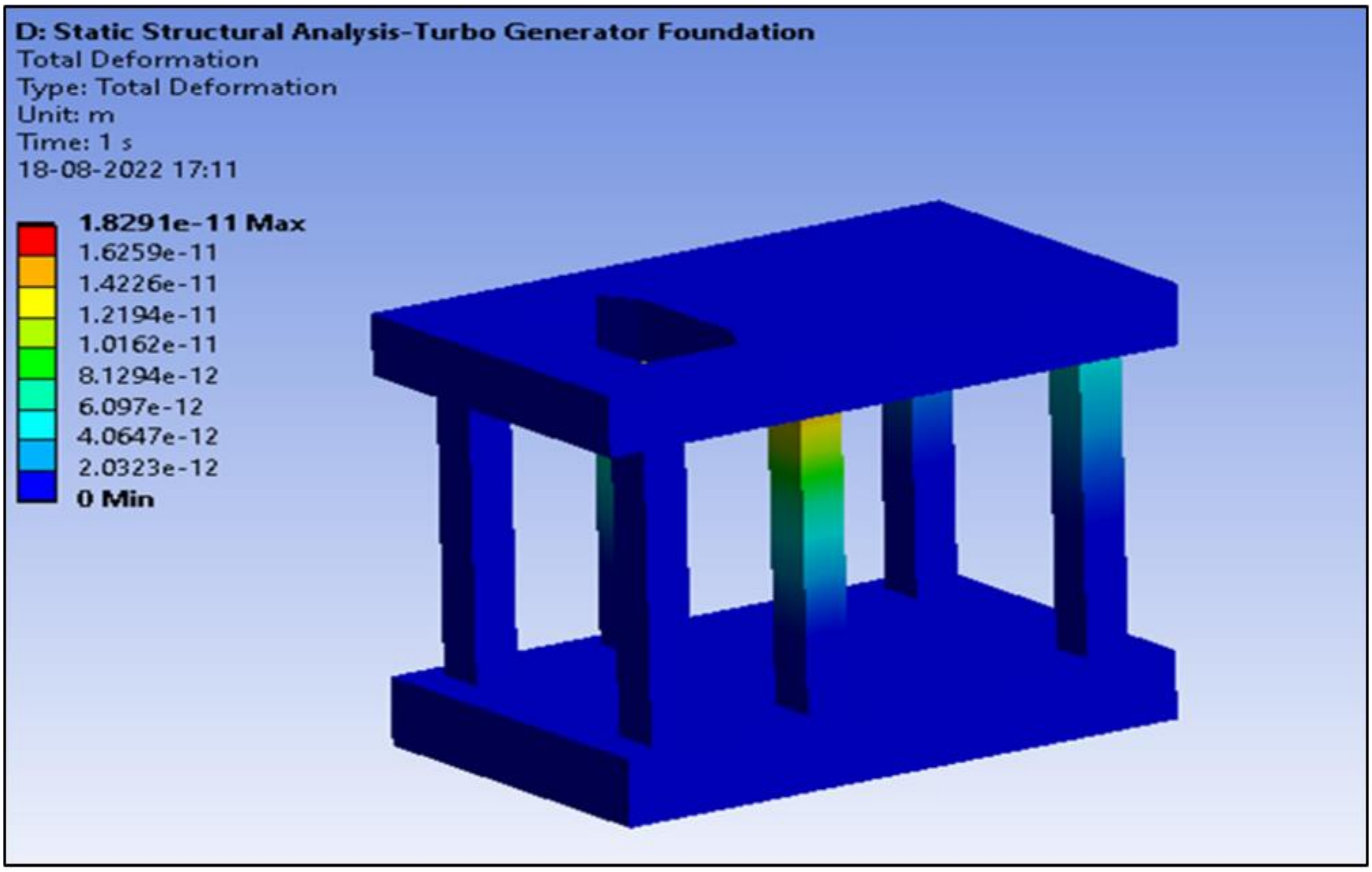

Fig 39 FEA model representing total deformation of frame type of machine foundation under static loads.

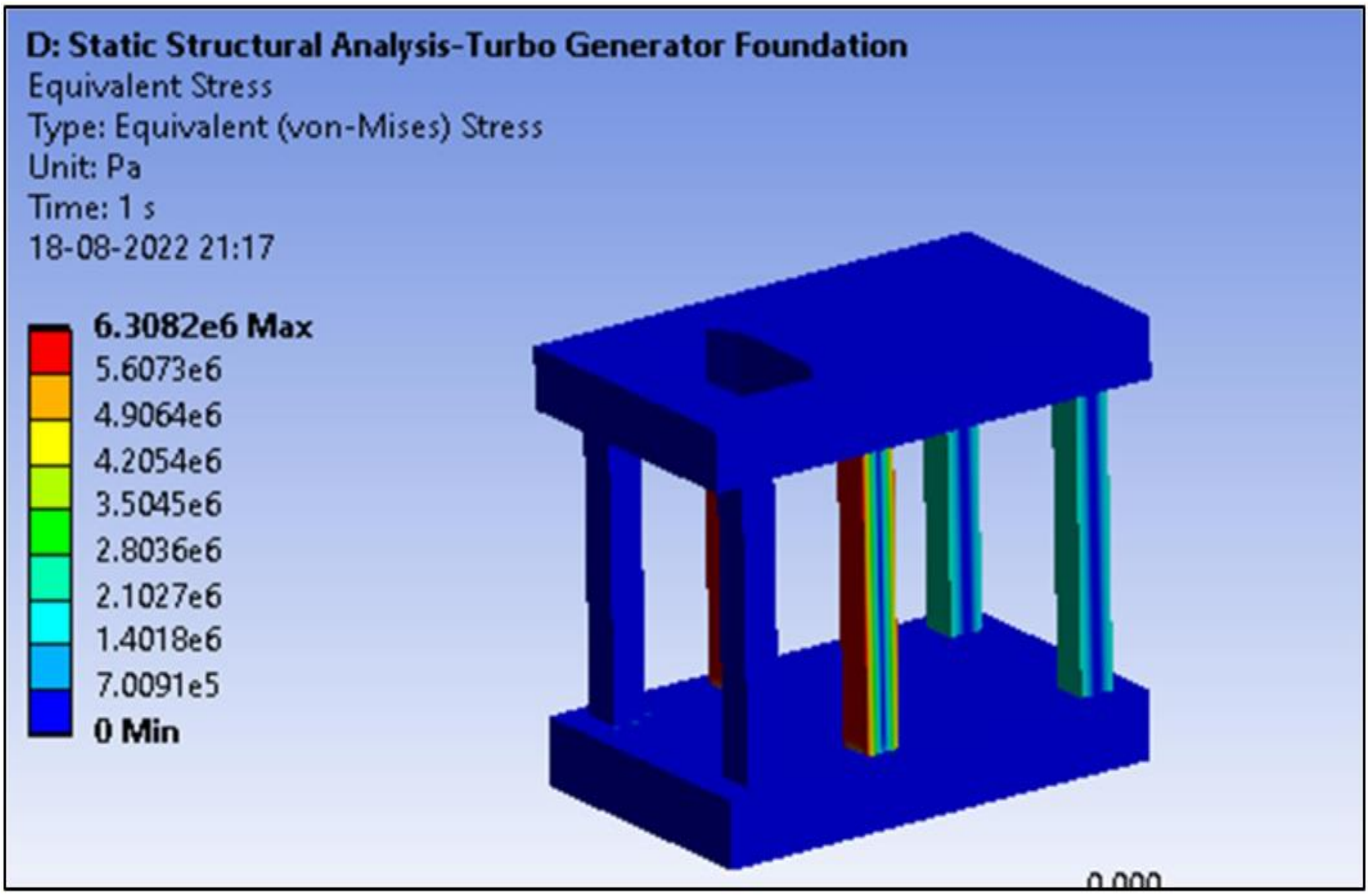

Fig 40 FEA model representing Equivalent stress distribution of Frame type of Foundation under static loads

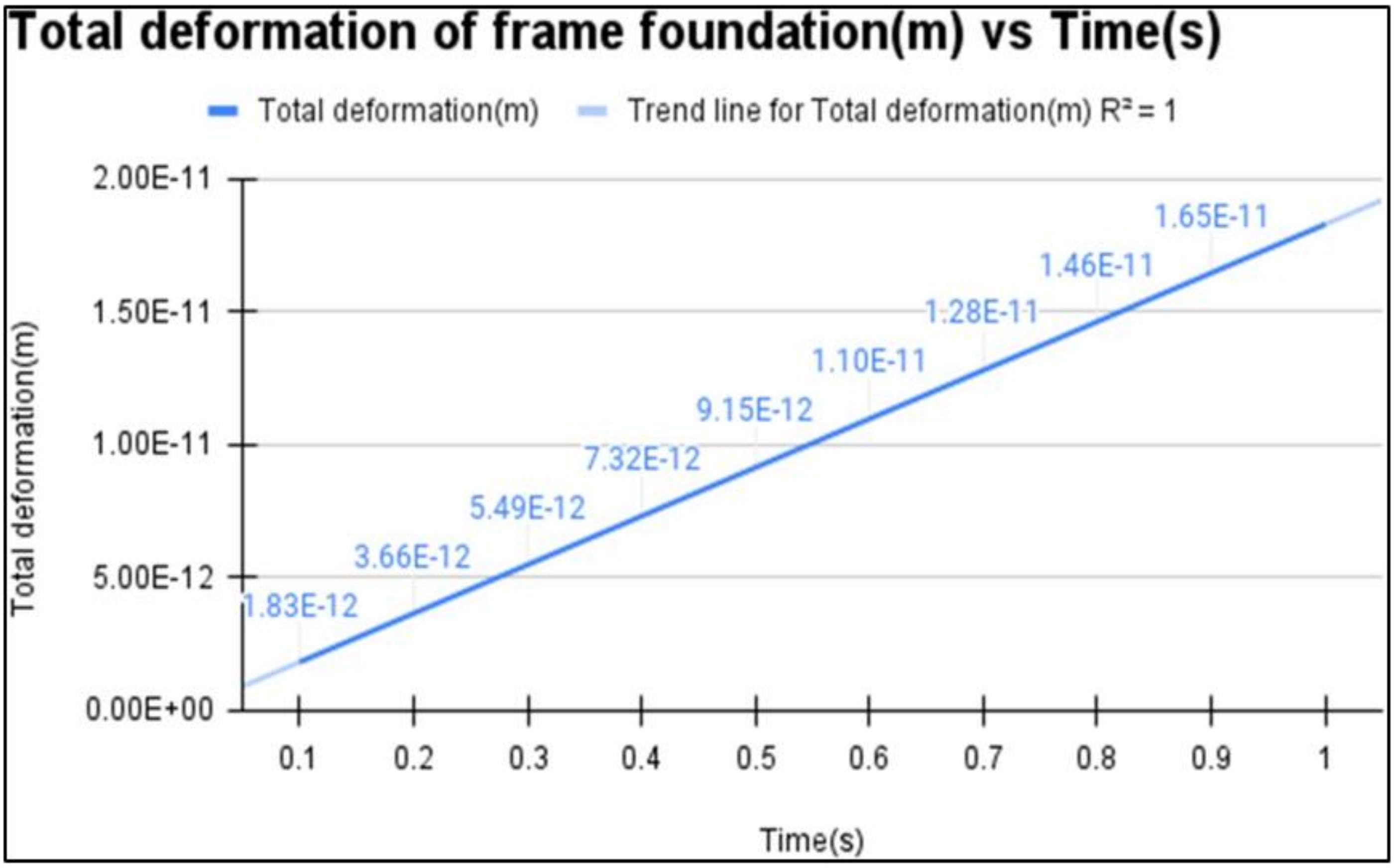

Fig 41 A graphical representation of the variation of total deformation of frame foundation concerning time.

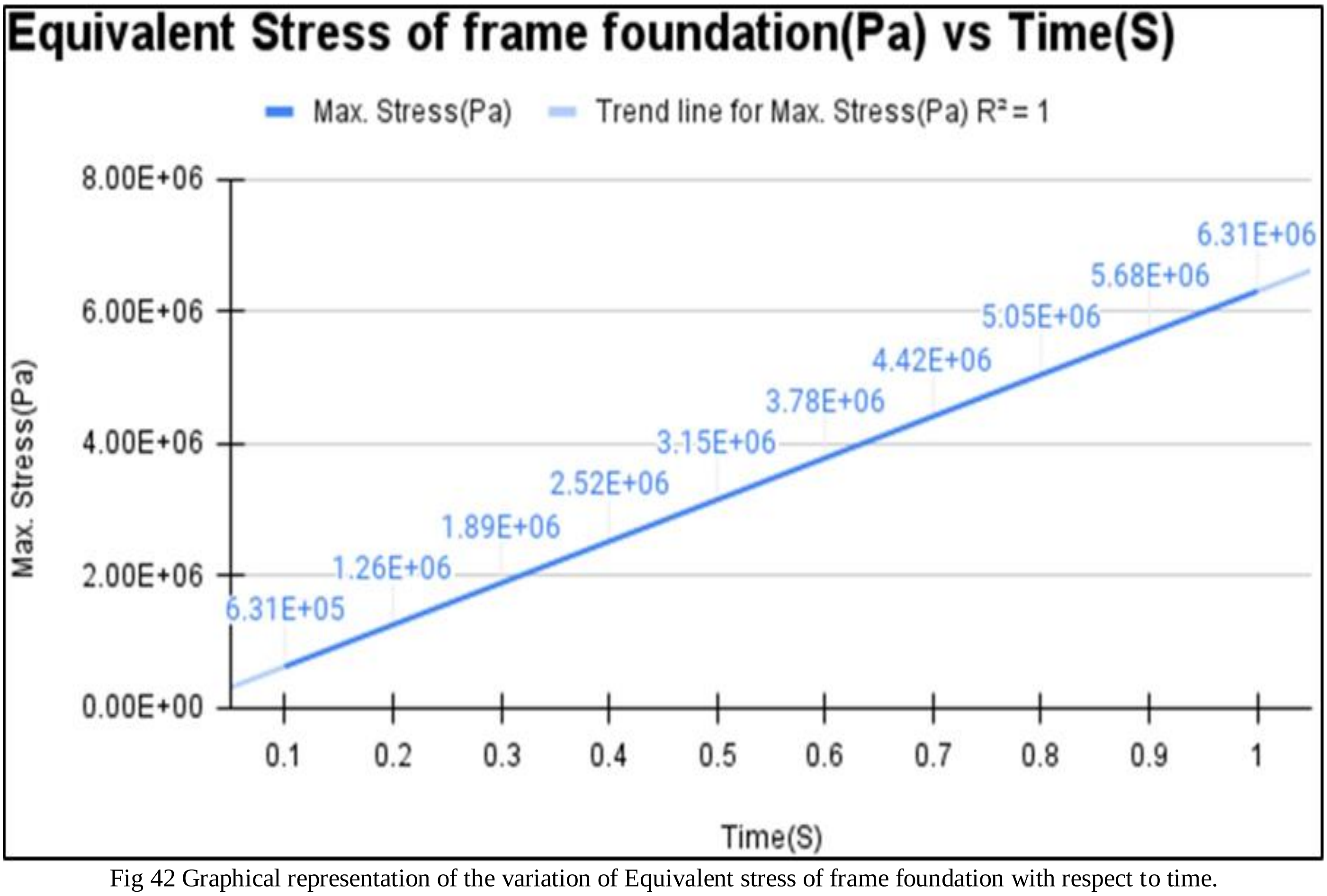

Fig 42 Graphical representation of the variation of Equivalent stress of frame foundation with respect to time.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

Table 14 Equivalent stress and Total Deformation of Frame type of foundation under static loads.

Time(S)	Equivalent Stress(Pa)	Total deformation(m)
0.1	6.31E+05	1.83E-12
0.2	1.26E+06	3.66E-12
0.3	1.89E+06	5.49E-12
0.4	2.52E+06	7.32E-12
0.5	3.15E+06	9.15E-12
0.6	3.78E+06	1.10E-11
0.7	4.42E+06	1.28E-11
0.8	5.05E+06	1.46E-11
0.9	5.68E+06	1.65E-11
1	6.31E+06	1.83E-11

Figure 39 shows the total deformation of the frame type of the machine foundation. Figure 40 represents the distribution of equivalent stress of frame foundation under static loads. Figure 41 shows the graphical variation of deformation with respect to time and figure 42 shows the variation of the Equivalent stress of foundation with respect to time. Both graphs figure 43&44 represents the linear variation of deformation and equivalent stress of frame foundation with respect to time, which shows the linear elastic nature of the foundation.

➢ *Dynamic Analysis of Machine Foundations:*

The maximum stresses and deflection induced in both types of machine foundations under dynamic loads are shown in table 6.4.

Table 15 Maximum Stresses and Deformation of Frame and Block Foundation under Dynamic loads

Type of foundation	Block Foundation	Frame foundation
Maximum deformation(mm)	18.79	8.54×10^{-2}
Maximum Stress (Mpa)	3.8×10^{2}	3.7489

- *Results of stresses and deformation of block foundation under dynamic analysis:*

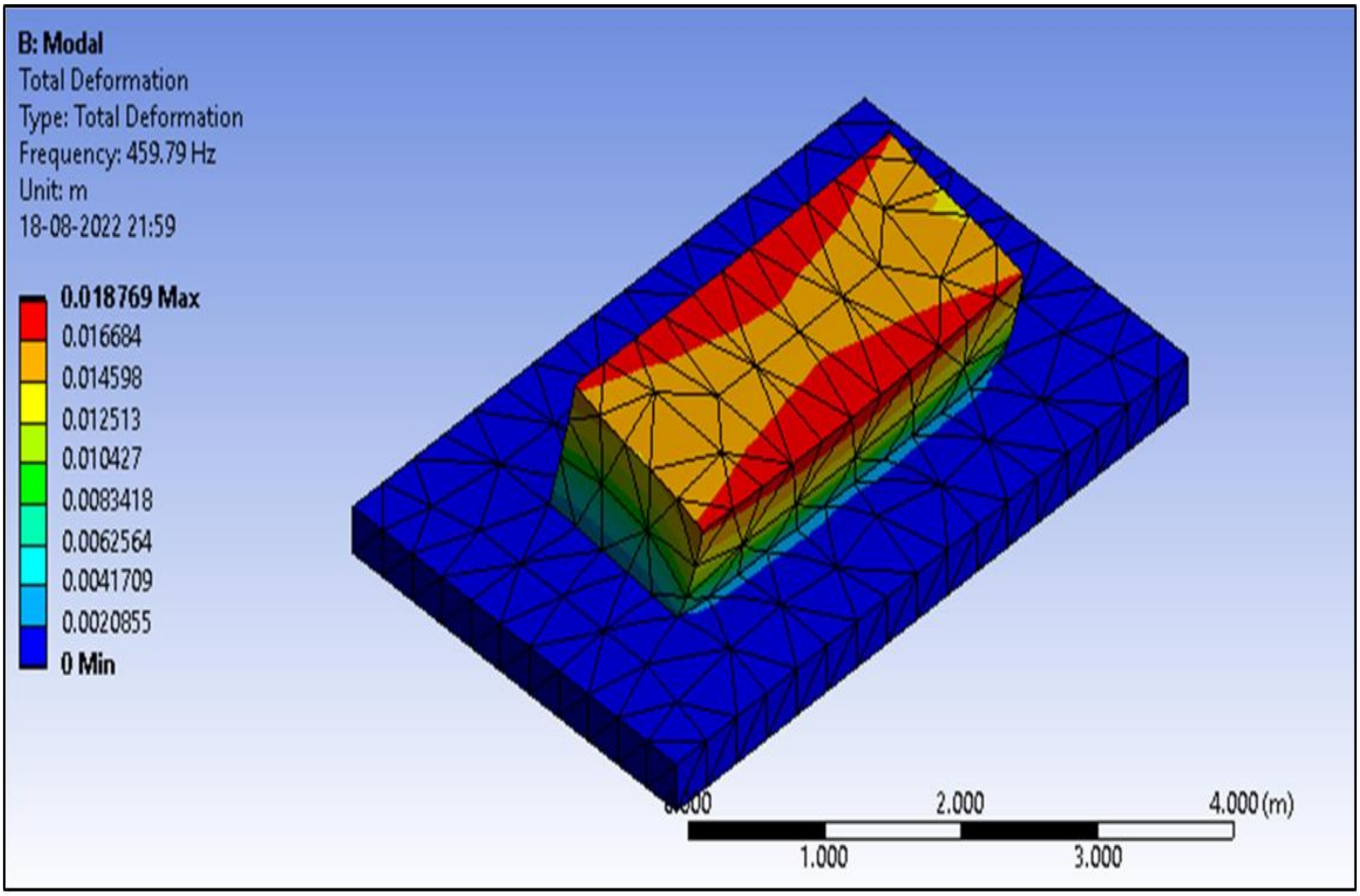

Fig 43 FEA model of Total Deformation of block foundation under dynamic loads.

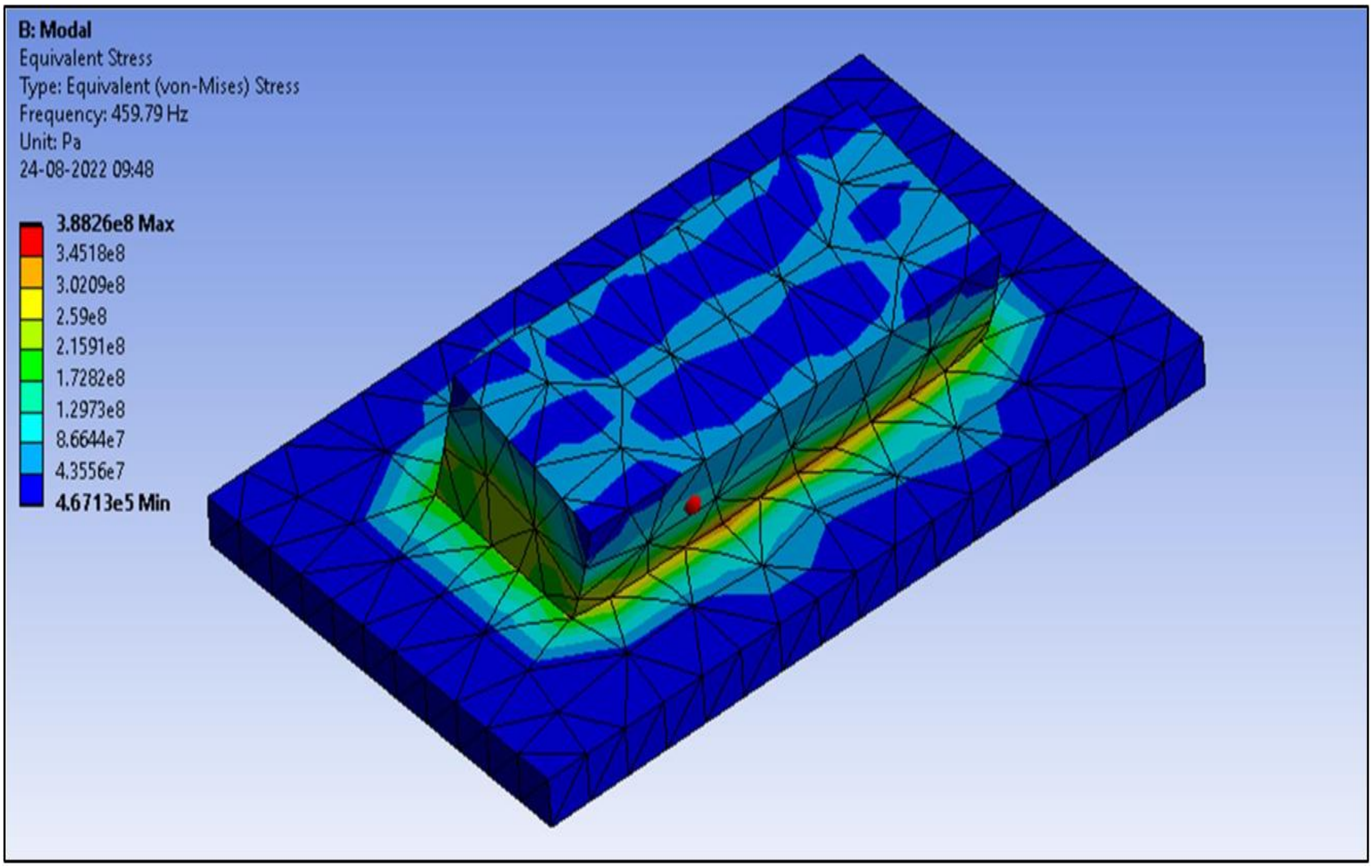

Fig 44 FEA Model of Distribution of Equivalent stress in block foundation under dynamic loads.

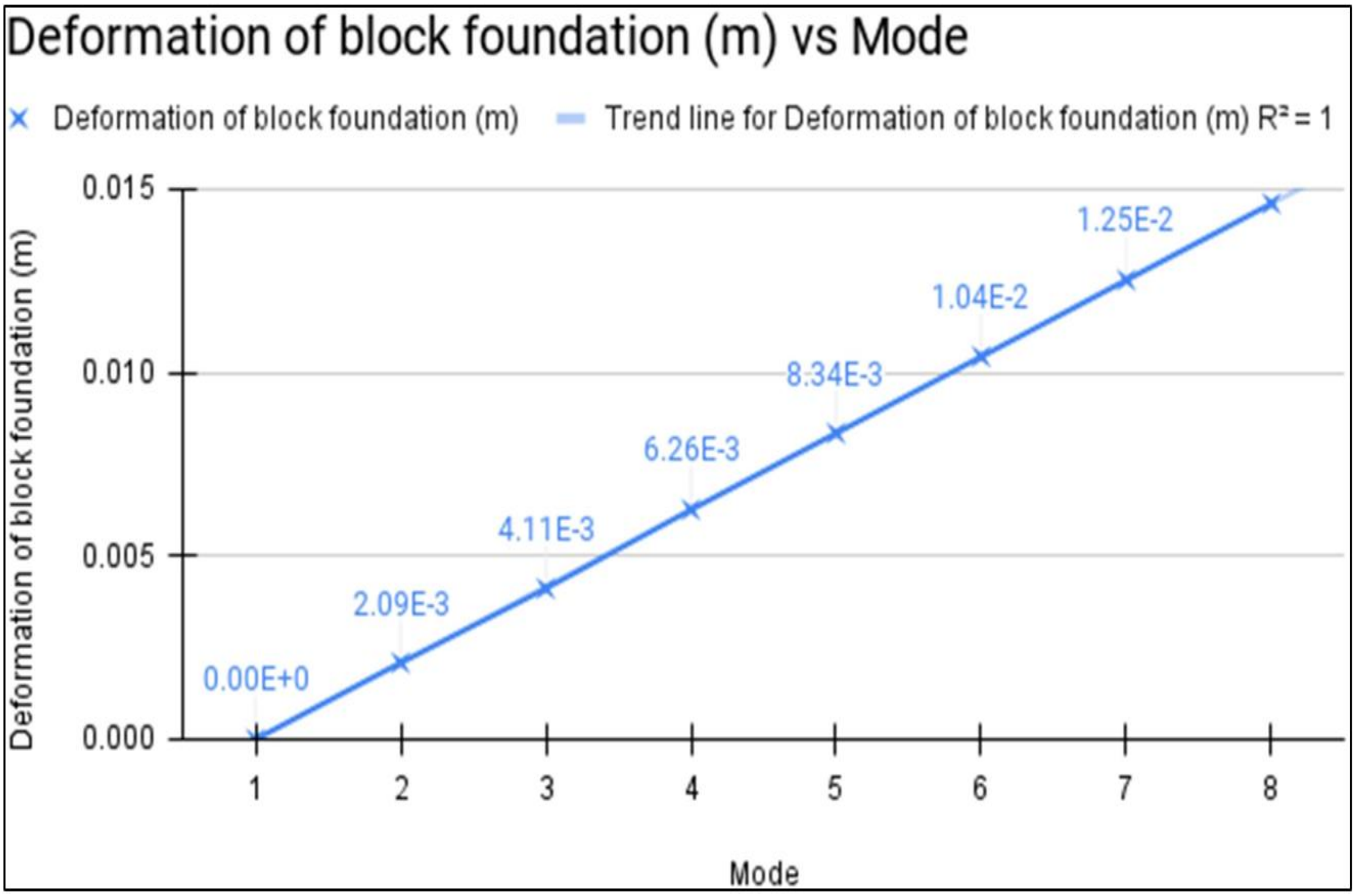

Fig 45 Graphical representation of Total deformation of block foundation under dynamic loads.

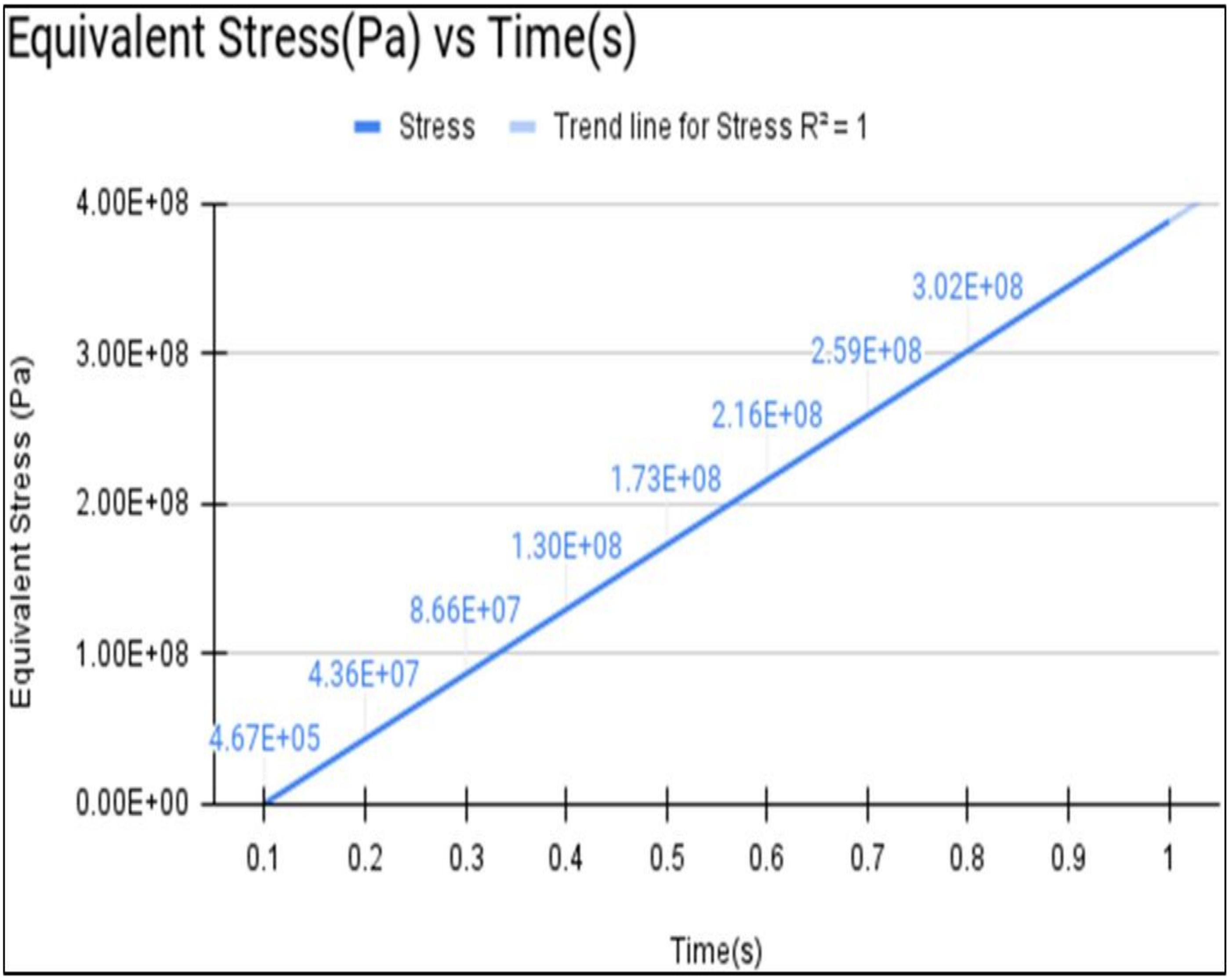

Fig 46 Graphical representation of Equivalent stress of block foundation under dynamic loads.

Table 16 Total Deformation of block foundation in terms of frequency and meters under dynamic loads.

Mode	Total deformation (Hz)	Total deformation(m)	Equivalent stress(Pa)
1	459.79	0	4.67E+05
2	509.46	0.0020855	4.36E+07
3	523.42	0.0041079	8.66E+07
4	647.01	0.0062564	1.30E+08
5	704.12	0.008341	1.73E+08
6	833.99	0.010427	2.16E+08
7	876.82	0.012513	2.59E+08
8	905.56	0.014598	3.02E+08

Figure 45 shows the Total deformation of the block foundation under dynamic loads, Figure 46 shows the distribution of Equivalent stress induced in the block foundation due to dynamic loadings. Figures 47 & 48 represents the graphical variation of Deformation and Equivalent stresses induced in foundations with respect to time under dynamic loads.

Both graphs show a linear variation of deflection and stresses with respect t time, which represents the rigid body behaviour of the block foundation. Because of this rigid body behaviour, the natural frequencies in different modes are higher.

➢ *Results of Dynamic analysis deformations and stresses on Frame Foundation:*

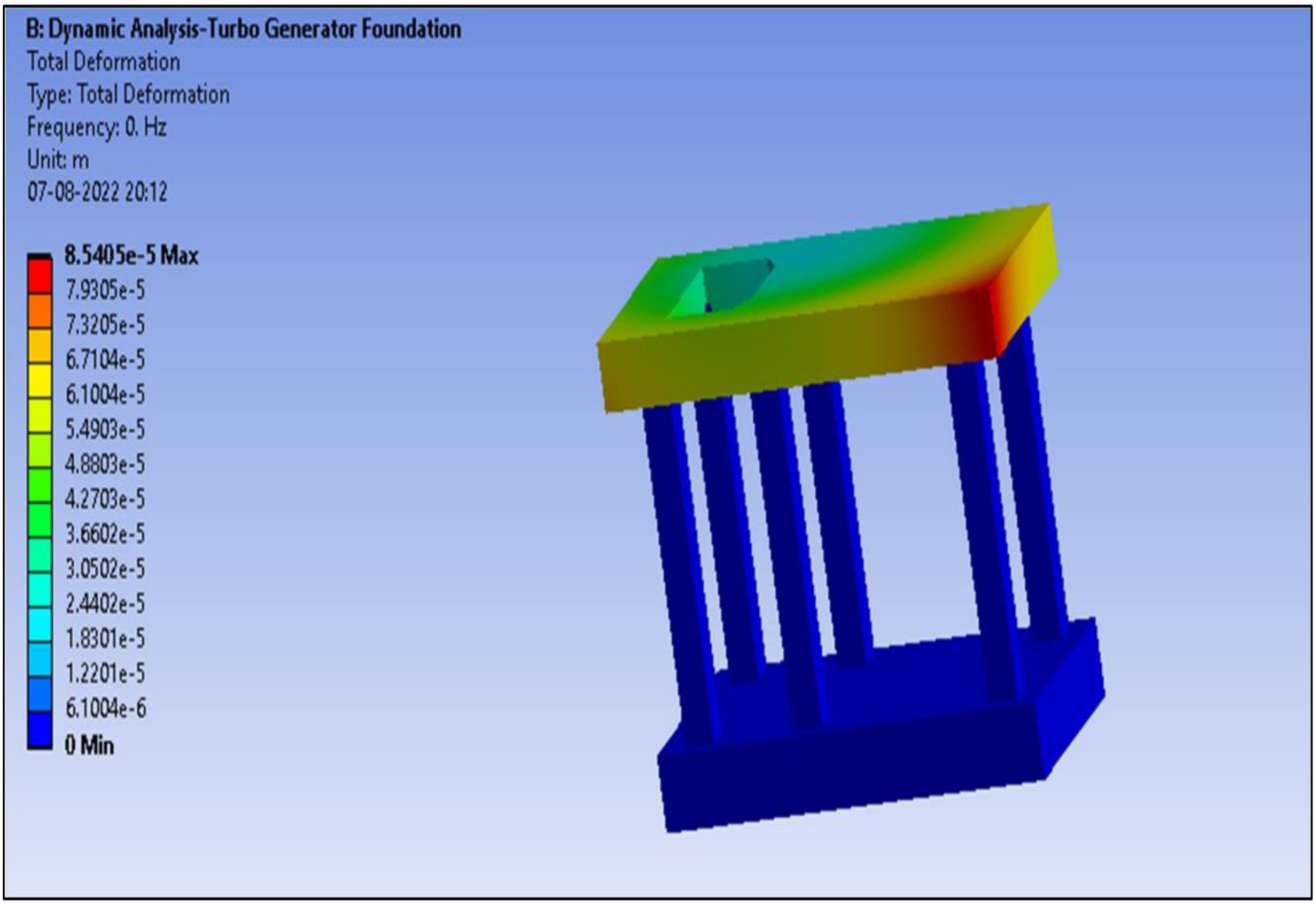

Fig 47 Dynamic analysis Total deformation of frame foundation.

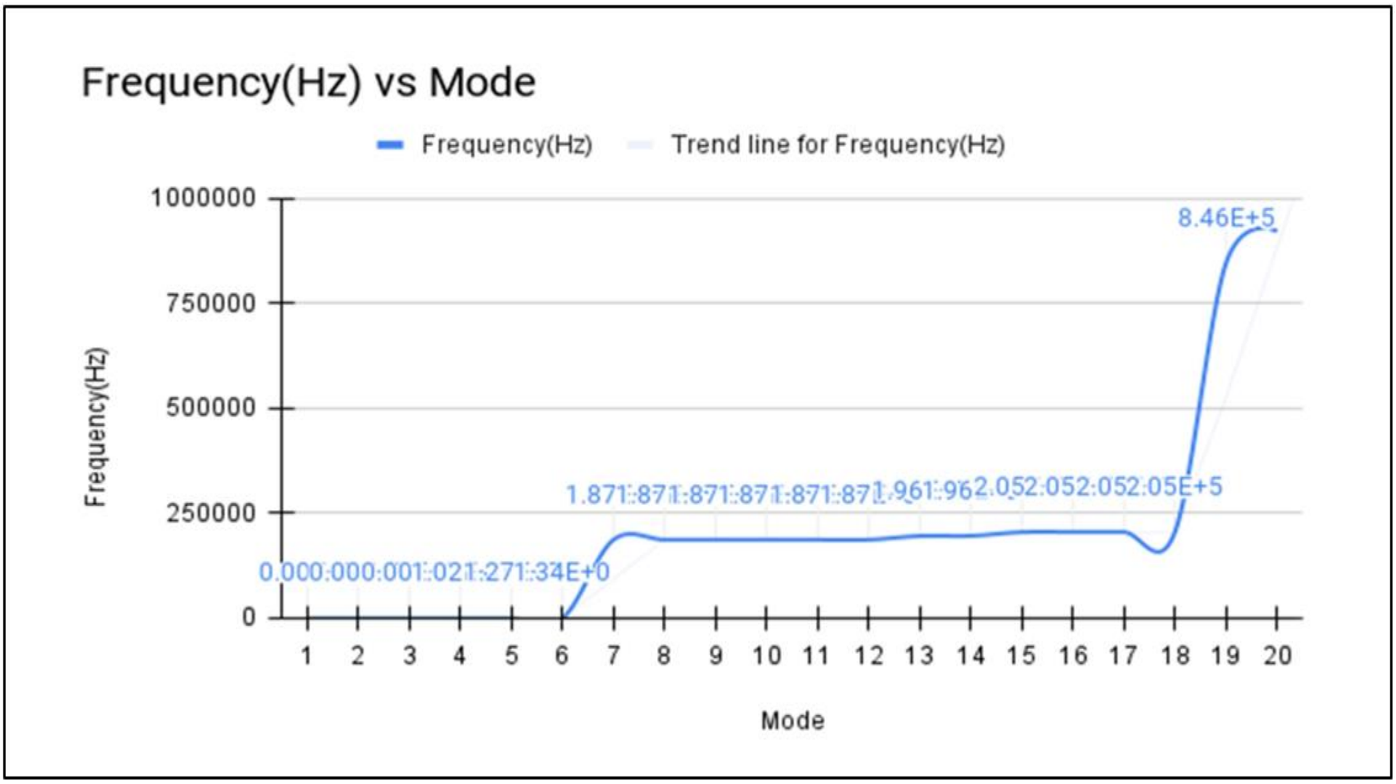

Fig 48 Graphical representation of the frequency of Frame foundation under dynamic loads.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

Table 17 Natural Frequency(Hz) of frame foundation for different modes.

Mode	Frequency(Hz)
1	0
2	0
3	0
4	1.0211
5	1.2716
6	1.3404
7	1.87E+05
8	1.87E+05
9	1.87E+05
10	1.87E+05
11	1.87E+05
12	1.87E+05
13	1.96E+05
14	1.96E+05
15	2.05E+05
16	2.05E+05
17	2.05E+05
18	2.05E+05
19	8.46E+05
20	9.22E+05

Figure 47 shows the Deformation of the frame foundation, figure 48 graphically represents the variation of deformation of frame type of machine foundation concerning different modes. There is an abnormal pattern in the deformation of the frame foundation as in figure 49, there is a sudden rise in the deformation after the 18th mode, this sudden rise is due to the slenderness of the machine foundation compared to the block foundation.

Whereas, the natural frequency of a particular body is dependent on the mass and tension of the body. In the case of dynamic analysis of frame type of machine foundation, there is no variation of mass concerning time, hence, the mass of the foundation does not affect the change in frequency. But, as the number of modes increases, there is a significant rise in tension induced in the frame foundation, which leads to an increase in the value of the natural frequency of the frame foundation.

- *Topology Optimization:*

Based on the previous analysis investigations, there are some components in the structure which are either under-stressed or overly-stressed, these components lead to the optimization where the wastage of material is reduced and thus obtaining a stable structure with all the strength and safety requirements.

In this case of optimization, the main aim is to avoid resonance in the structure. Hence either the whole structure or its components are to be modelled separately and analysed for dynamic loadings. The top deck and bottom raft slabs of the frame foundation are similar to that of the block foundation exhibiting rigid mass behaviour with a higher natural frequency than the operating frequency of the machine which is 50Hz. Hence, there is a requirement to analyse the column due to its slender nature.

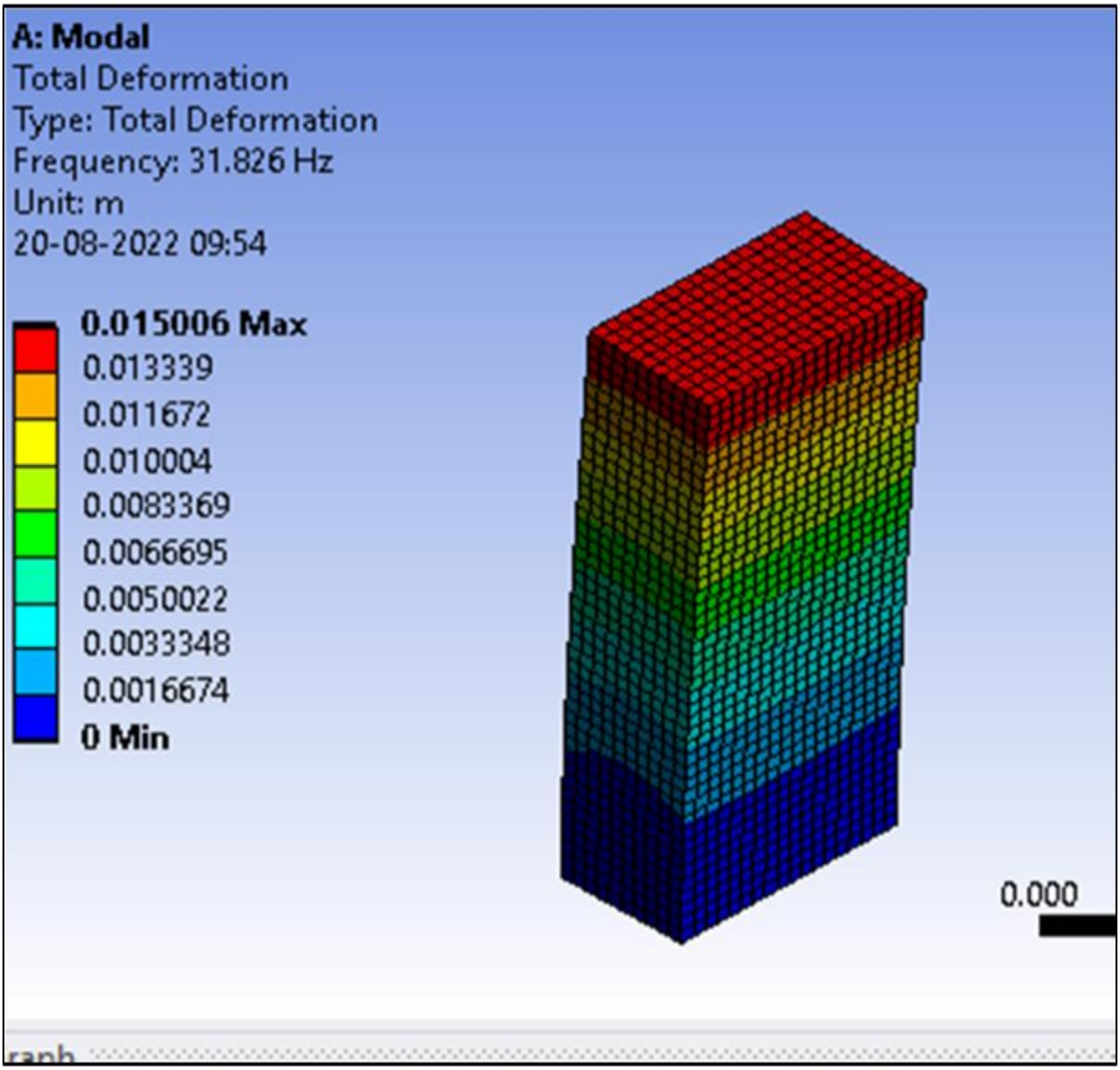

Fig 49 Total deformation of the single column

Table 18 Total Deformation of Column under dynamic loading

Mode	Total deformation(Hz)	Total deformation (m)
1	31.826	0.001667
2	53.009	0.003334
3	113.24	0.005002
4	165.55	0.0066695
5	215.05	0.00833689
6	221.06	0.01004

Figure 49 show the total deformation of a single column under dynamic loading. Table 17 represents the values of deflection of a single column obtained due to dynamic loads. From table 18, it can be seen that the frequency of the column in the 2nd mode of vibration(53Hz) is nearly equal to the operating frequency of the machine(50Hz), this leads to the resonance in the structure, failing the machine foundation in resonance.

This leads to the optimization of the column. Therefore, topology optimization is carried out in the ANSYS workbench.

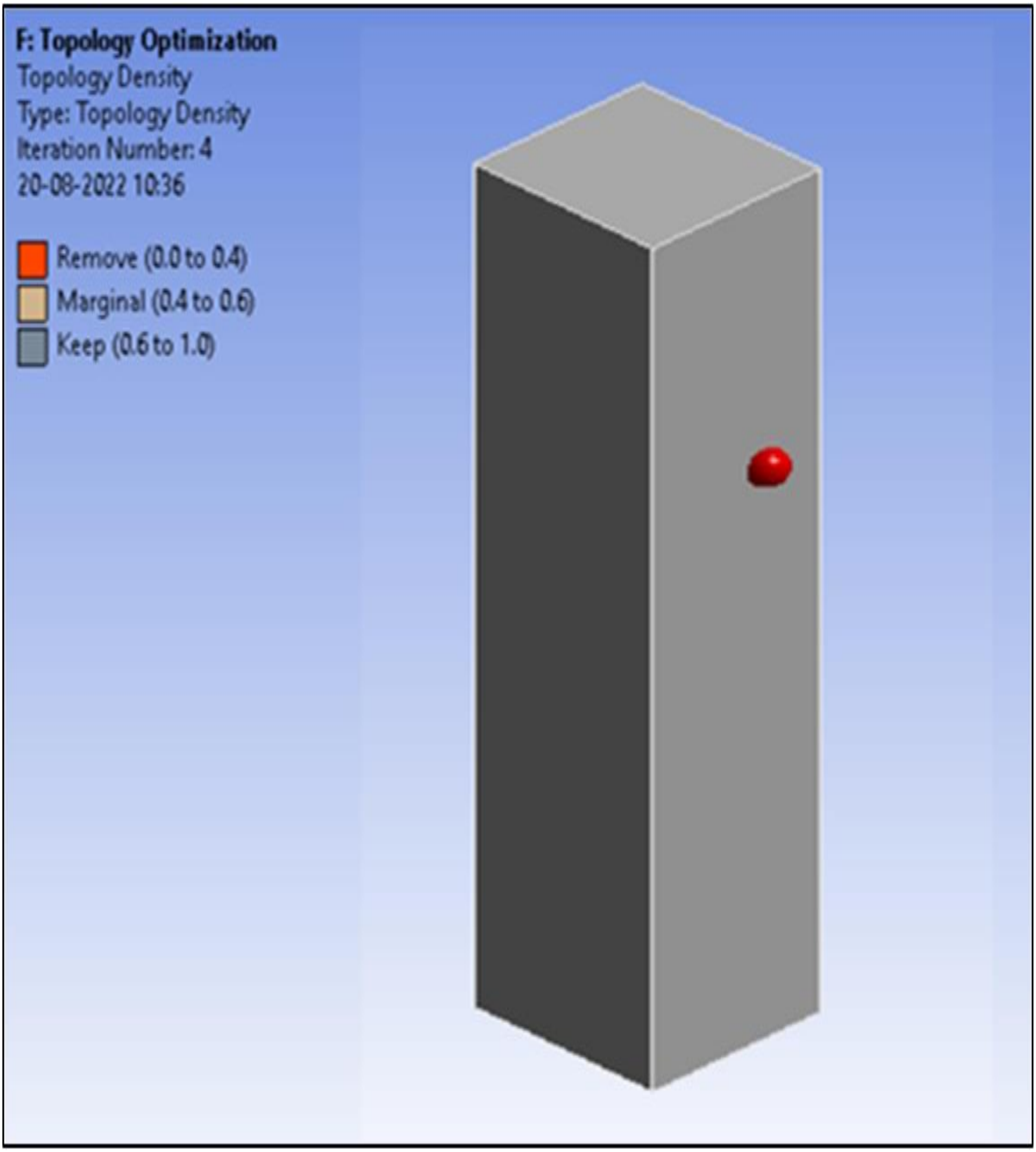

Fig 50 Optimized Column

Table 19 Dimensions and frequency values of the column before and after topology optimization

Term	Before Topology Optimization	After Topology Optimization
Dimensions of column (breadth X width) (m)	0.9 × 1	0.7 × 0.8
Frequency value at 2^{nd} mode(Hz)	53.00	44 Hz

Figure 50 represents the final optimized column with a 30% reduction in mass and volume. Table 19 shows the dimensions and frequency values of the column before and after topology optimization. With the results obtained, the dimensions of the column are reduced which resulted in a lower value of natural frequency, thus avoiding resonance in the structure.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/24nov077

CHAPTER SEVEN
CONCLUSIONS & SCOPE OF FUTURE WORK

- *Conclusions***:**

Based on the finite element analysis of the foundation types block and frame machine foundations, subjected to static and dynamic loads, the conclusions are drawn as follows:

The maximum deformation and maximum stress induced in the block foundation due to static loads are 1.096 × 10–4 mm and 2.47 × 10–3 Mpa. The maximum deformation and stress induced in the frame foundation subjected to static loads are 1.8291 × 10–8 mm and 6.3 Mpa.

The maximum deformation and stresses induced in the block foundation due to dynamic loads are 18.79 mm and 3.8 × 102 Mpa. The maximum deflection and stresses induced in the frame foundation due to dynamic loads are 8.54 × 10–2 mm and 3.75 Mpa.

With the lower deformations in the frame foundation, these are used for high- speed machines and larger machines.With the higher deformations, the block foundations are used for compressor machines.

An individual column component, when analysed dynamically resulted in a resonance condition, therefore, a topology optimization carried out resulted in the 30% mass reduction and avoided resonance, satisfying all the strength and serviceability criteria.

The resonance condition in the block foundation can be neglected due to its rigid body mass behaviour resulting in higher natural frequencies. Hence, low-speed machines can be used.

Whereas, the frame foundation due to its slender components columns, there is a chance for resonance in the structure, with its lower natural frequencies. Therefore, IAt is safe to use high-speed machines.

- *Scope of Future Work:*

The present work is carried out by modelling two types of machine foundations, block and frame. Future endeavours may analyse different other types of machine foundations dynamically.

Future studies may include the usage of vibration isolation devices for the foundations and the usage of different materials like steel frame foundations.

REFERENCES

[1]. Giorgetti Stefano, Alessandro Giorgetti, & Gabriele arcidiacona, "Machine Foundations dynamical analysis: A case study on reciprocating compressing foundation", Journal of Machine MDPI, 2021, vol.9, P.1-18
[2]. Goranka Stimac, Sanjin Braut & Roberto Zigulic, "Optimization of machine foundations using frequency constraints", springer- Verlag berlin, 2014, P.1-11
[3]. Jayaraman P & Kouzer K.M. "Dynamic analysis of Turbo generator machine foundation", Journal of civil engineering and engineering technology, 2014, vol.4,P.45-53
[4]. K.G. Bhatia, "Foundations for industrial machines and earthquake effects", ISET Journal of Earthquake Technology, 2008, vol.45, P.13-29
[5]. Mohammed Adel, Yassr Alzeni & Ahmed Ghalab, "Modelling of turbo generator foundation using FEM to study isolation effect on the system", International Journal of Innovations in Engineering and Technology 2019, vol.9, P.31-41
[6]. Mohammed Yousif, Mohammed J Hamood, "Finite Element Analysis of piled machine foundation", Proceeding of the Institution of civil engineering- Structures and Buildings, 2015, Vol. 5, P.1-14.
[7]. Nawrotzki P, Huffman G, & Uzunoglu T, ' Static and Dynamic analysis of concrete turbine foundations, structural engineering international, 18(3), 2008, vol.3, P.265- 270
[8]. P.M.Shimpalle & Sindhu panapalli, " Dynamic Analysis of machine foundations using Ansys", International Journal of Engineering Research and Technology, 2018, vol.4, P.73-79
[9]. Prakash S, Puri V.K, " Foundations for vibrating machines", Journal of structural engineering, 2006, 33(1), vol.1, P.13-29
[10]. Song F. Jan & Sheng C., "Dynamic analysis and design large compressor foundations in the high seismic zone", ASCE structures congress, 2010, vol.3, P.2702- 2713
[11]. Zhong Liu, " Design of foundations for large dynamic equipment in a high seismic region", ASCE structures congress, May 2013, P.1403-1414.
[12]. ANSYS user manual 2022
[13]. Bathe K. "Finite Element Procedures in Engineering Analysis", Englewood Cliffs, NJ: Prentice Hall, 1981.
[14]. Jasbir S. Arora, "Guide to Structural Optimization", ASCE, 1997
[15]. K.G. Bhatia, " Handbook on Design and construction of foundations for industrial machines", D-CAD, 2008
[16]. IS 2974-1992 part 3, "Code of practice for Analysis and design of machine foundations for Rotary machine types (Medium and High frequencies)"