OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Investigation of Effects of impact loads on Framed Structures

Dhiraj R. Patil[1], U. R. Awari[2]
[1](*PG student AISSMSCOE, Pune (India)*
[2](*Assistant professor, AISSMSCOE, Pune (India)*

Abstract: *This research work consists of a general overview of numerical analysis and dynamic response of framed structures under impact loading. The purpose of the work is to introduce the Finite Element Method which is difficult while analyzing dynamic response to framed structures. Also to introduce the Ansys software and it will also explain and discuss particular model cases subjected to various impact loadings. With these models there will be understanding of the behavior of framed structures showing the clear results of stress, strain and deformation developed throughout the structures.*
Keywords: *Ansys, Deformation, Dynamic Response, Impact Loading, Numerical analysis.*

I. Introduction

This paper presents method of analysis for the framed structures of steel. The framed structure has the function of protecting important equipments, structures or specific areas from the damage of projectile impact. The affected structural member will undergo large deformation to absorb the energy brought by the projectile, without loss of its own integrity. The effect of an impact force on structure depends on the shape, mass, and velocity of the impacting body; the contact area; the structure's dynamic response; and the material type, etc. A significant analysis effort is required to evaluate the behavior of a structure under impact loading. To simplify the design, a methodology has been developed using an equivalent static load for a framed structure under impact load. This method has been used in structural design and has demonstrated satisfactory results in meeting design criteria. [1]

This paper work deals with behavior of framed structure under impact loading. In beginning there is over all study of Dynamic behavior of frame under Impact load. After that study, by using the analytical methods there is examination of the framed structure. Later by casting the model of frame structure there is experimental study by applying impact loads on it. Here impact load is created by dropping mass from desired height. In this work, impact load is applied on framed structure from certain height and observations are recorded by using FFT Analyzer and accelometer. Such successive readingsrecorded by gradually increasing height and the mass so that, impact will increases. Follow the same work as above for all readings. Finally, experimental and analytical readings are compared to draw conclusion.

1.1 Fast Fourier Transform (FFT)

The Fast Fourier Transform (FFT) is the powerful tool for analyzing and measuring signals from plug-in data acquisition devices. By using FFT, you can build a lower cost measurement system and avoid the communication overhead of working with a stand-alone instrument. Plus, you have the flexibility of configuring your measurement processing to meet your needs.

To perform FFT-based measurement, however, you must understand the fundamental issues and computations involved. This application note serves the following purposes.

1. Describes some of the basic signal analysis computations.
2. Discusses antialiasing and acquisition front ends for FFT based signal analysis.
3. Explains how to use windows correctly.
4. Explains some computations performed on the spectrum.
5. Shows you how to use FFT-based functions for network measurement.

The basic functions for FFT-based signal analysis are the FFT, the Power Spectrum, and the Cross Power Spectrum. Using these functions as building blocks, you can create additional measurement functions such as frequency response, impulse response, coherence, amplitude spectrum, and phase spectrum. [2]

1.2 The Finite Element Method

The finite element method (FEM) is the dominant discretization technique in structural mechanics. The basic concept in the physical FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points. The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements.

A general procedure for finite element analysis comprises of certain steps which are common to all such analyses, whether fluid flow, structural, heat transfer or some other problem. These steps are sometimes embodied in the software packages used for commercial purposes. The steps can be described as follows:

i) Preprocessing:

This step in general includes:

1. Defining the geometric domain of any problem.
2. Defining the type of element or elements to be used.
3. Defining the elemental material properties.
4. Defining the geometrical properties of elements such as length, area etc.
5. Defining the connectivities of elements.
6. Defining the boundary conditions.
7. Defining the conditions of loading.

This step sometimes also referred as model definition step is critical. It can be said that a finite element problem which is perfectly computed is of no value if it corresponds to a wrong problem.

ii) Solution:

While solving the problem, FEM assembles the governing algebraic equations in matrix form and computes the unknown values of the primary dependent variables or field variables. These primary dependent variables can then be used by back substitution to compute additional, derived variables such as heat flow, element stresses, reaction forces etc.

iii) Post processing:

It comprises of the analysis and evaluation of the solution results. It contains sophisticated routines to print, sort and plot the selected results from a finite element solution. Operations that can be done include:

1. Checking equilibrium.
2. Animating the dynamic model behavior.
3. Plotting the deformed structural shape.
4. Calculating the factors of safety.
5. Sorting the element stresses in order of their magnitude. [3]

II. Experimental Analysis

2.1 Numerical Analysis

Finite element analysis/method (FEM) is very widely used method to assess the behavior of a variety of structural systems. FEM views the solution region built up of many small, interconnected sub regions or elements. It is designed to contain the structural properties and the material which specifies how the structure will react under specific loading conditions. ANSYS is a suite of powerful engineering simulation programs, based on the finite element method that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations.[4]-[5]

2.2 Modeling of the framed structure

To study this work, Ansys has been used to analyze the dynamic behavior of the Frame under the impact loadings. Thus the report successfully shows how a Frame Structure will resist impact loads or blast loads and bypasses the need to perform a physical experiment which is too costly and exceeds the factors of safety. In this experimental and analytical analysis steel frame with dimension of 0.3X0.3X0.3m and 0.3X0.3X0.6m are used. The same work can be done by varying dimensions of the model or by increasing number of storey or bays. Here, the results are shown only for the model of dimensions 0.3X0.3X0.3m. The Density used will be set as 1800 kg/m3, also specified from the journal above and is a value used in normal test data. An impact loading of certain amount is to be dropped on the framed structures from some height and as soon as it will fall on the structure, at the same instant observe FFT analyzer and record the reading. Then by varying height of drop and later mass of drop, numbers of observations are to be recorded. The intensity of falling amount of mass (force) applied on the top of the framed structure is considered as impact load.

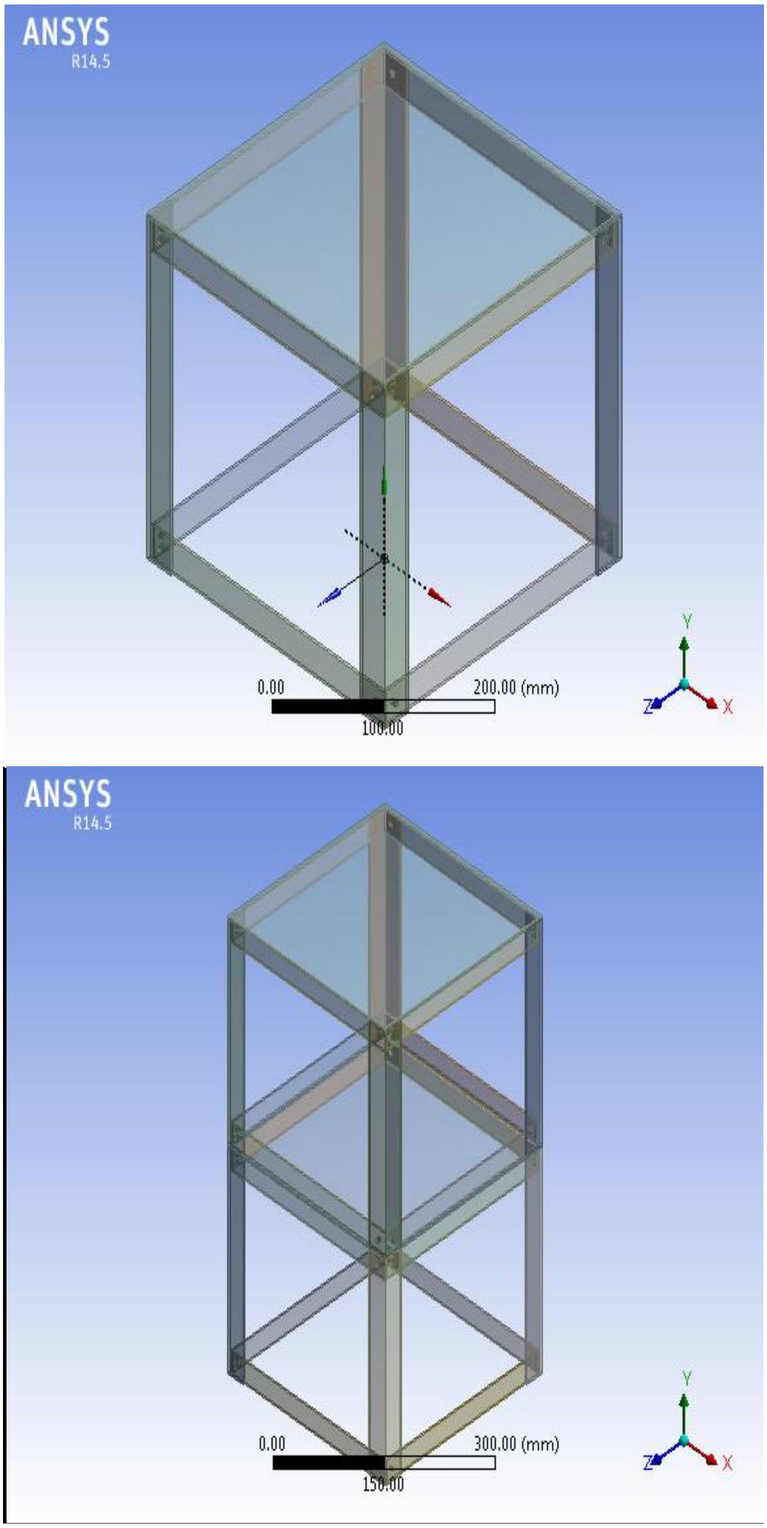


Fig. 1- Ansys Modeling of Framed Structure

Fig. 2- Model used for experimental work (connected with FFT analyzer)

2.3 Details of loading

Table 1: Total impact on the frame

Impact Load ⇨ Mass in Kg ⇩	Impact load (N) for height of 0.05 m	Impact load (N) for height of 0.15 m	Impact load (N) for height of 0.20 m	Impact load (N) for height of 0.30 m	Impact load (N) for height of 0.50 m
0.1	49	147	196	294	490
0.2	98	294	392	588	980
0.3	147	441	588	882	1470
0.5	245	735	980	1470	2450
1.0	490	1470	1960	2940	4900
1.5	735	2205	2940	4410	7350
2.0	980	2940	3920	5880	9800
3.0	1470	4410	5880	8820	14700
5.0	2450	7350	9800	14700	24500

III. Result And Discussion

Table 2: Amplitudes recorded in ANSYS and FFT for various loads

Amplitudes in nm ⇨ Mass in Kg ⇩	For height of 0.05 m		For height of 0.15 m		For height of 0.20 m		For height of 0.30 m		For height of 0.50 m	
	Ansys	FFT	Ansys	FFT	Ansys	FFT	Ansys	FFT	Ansys	FFT
0.1	-	-	75	110	100	110	150	172	250	242
0.2	-	-	150	172	200	200	300	312	501	521
0.3	75	110	225	229	300	312	451	434	752	757
0.5	125	115	376	380	501	521	752	757	1254	1230
1.0	250	242	752	757	1000	1120	1505	1480	2509	2510
1.5	376	380	1129	1130	1505	1480	2258	2240	3764	3420
2.0	500	521	1505	1480	2007	2180	3011	2750	5018	5330
3.0	752	757	2258	2240	3011	2750	4516	4490	7528	6520
5.0	1254	1230	3764	3420	5018	5330	7528	6520	12547	-

The above table shows the amplitudes for the loadings given in table 1 as above, by varying height of drop, for the model 0.3X0.3X0.3 m.

The following graphs show the variation in amplitudes for the various drops of heights and various loadings shown in tables 1 and 2. The variation is between ANSYS results and FFT results recorded in table 2.

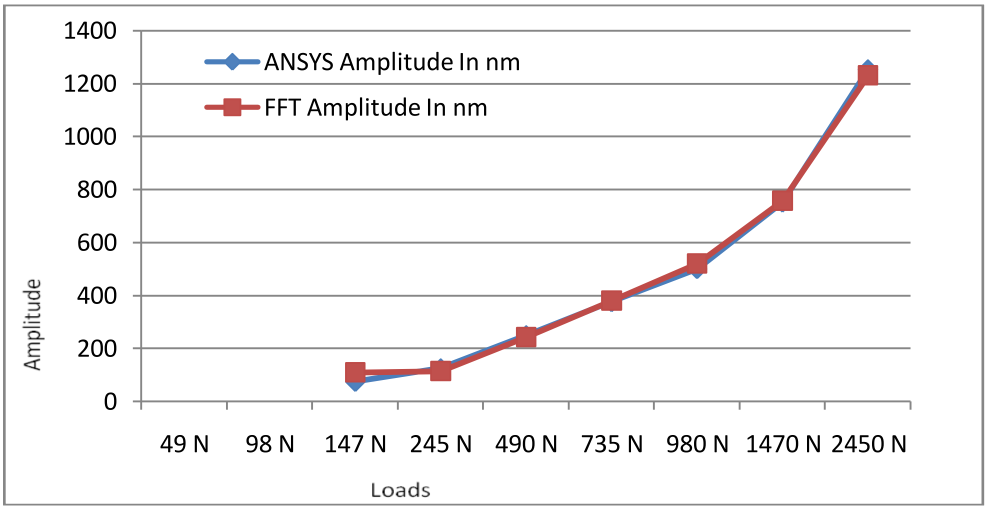


Fig.3: Graphical variation for the amplitude by FFT and ANSYS for model 0.3x0.3x0.3 for height 5 cm

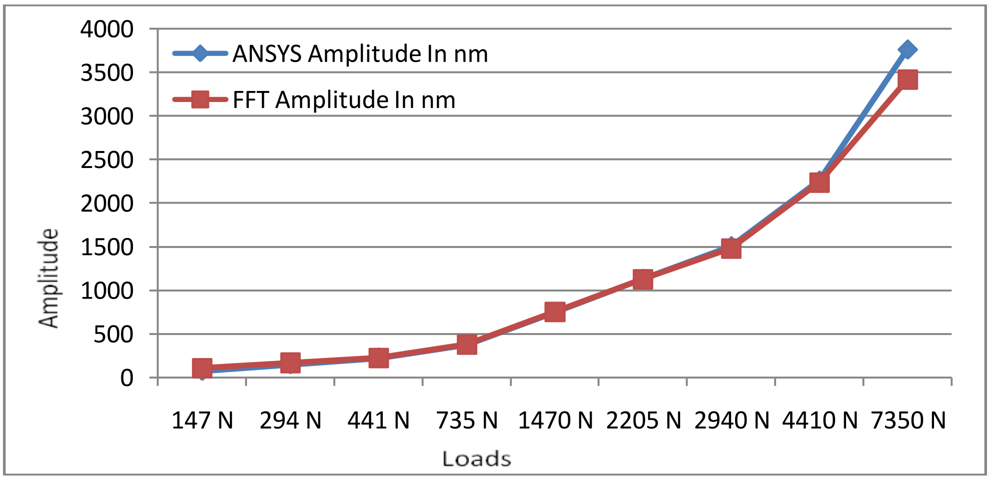


Fig.4: Graphical variation for the amplitude by FFT and ANSYS for model 0.3x0.3x0.3 for height 15 cm

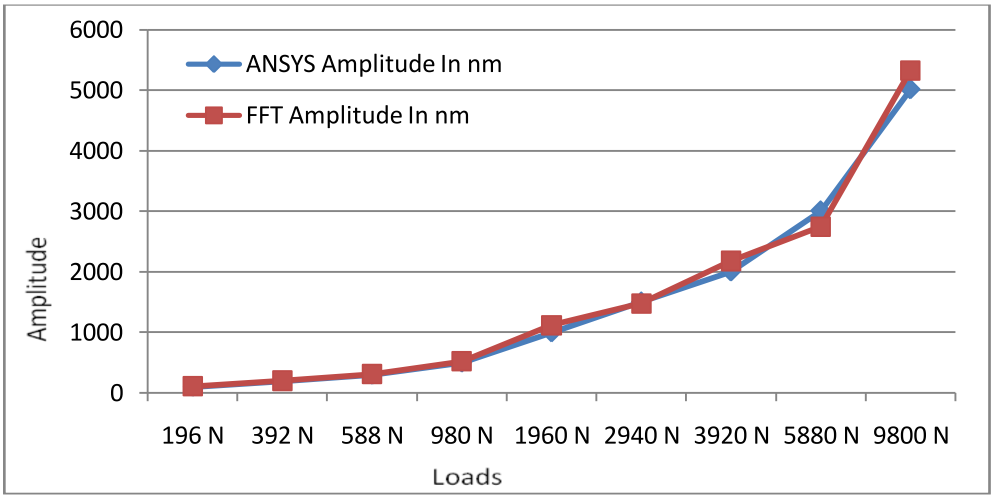


Fig.5: Graphical variation for the amplitude by FFT and ANSYS for model 0.3x0.3x0.3 for height 20 cm

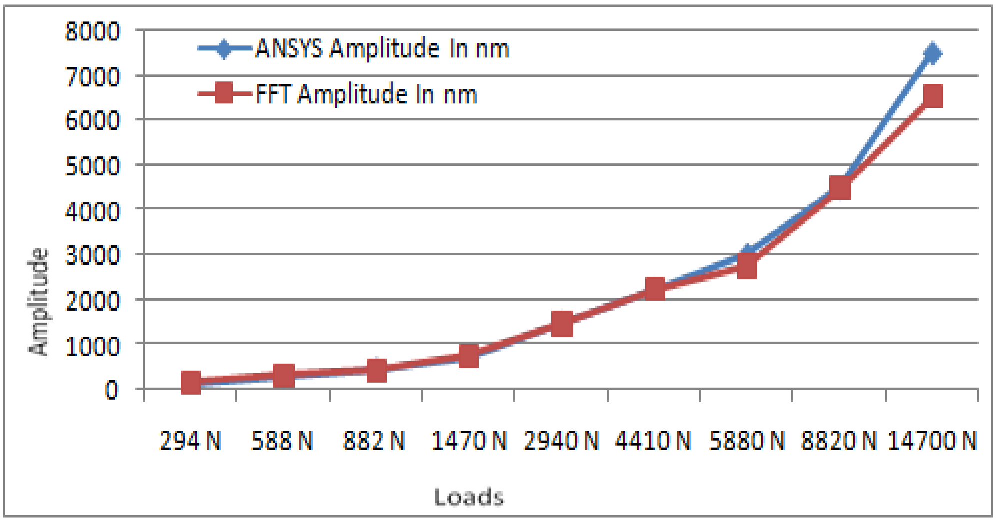


Fig.6: Graphical variation for the amplitude by FFT and ANSYS for model 0.3x0.3x0.3 for height 30 cm

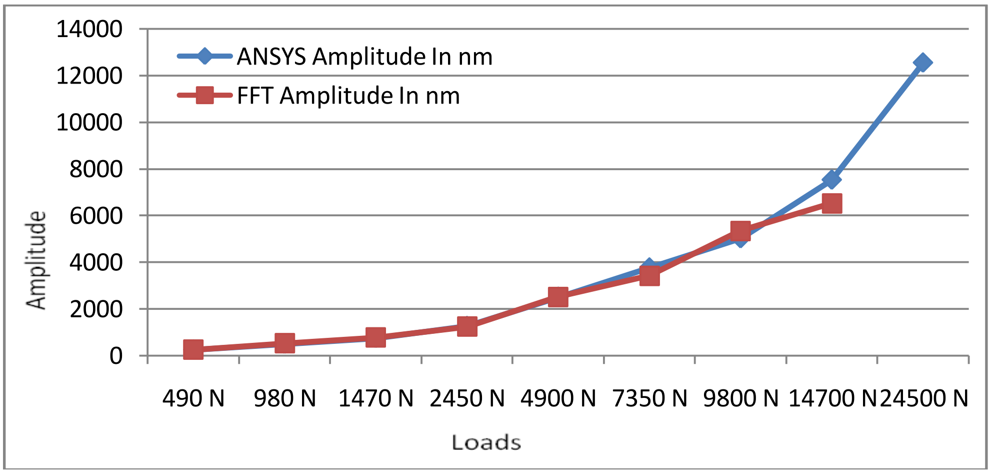
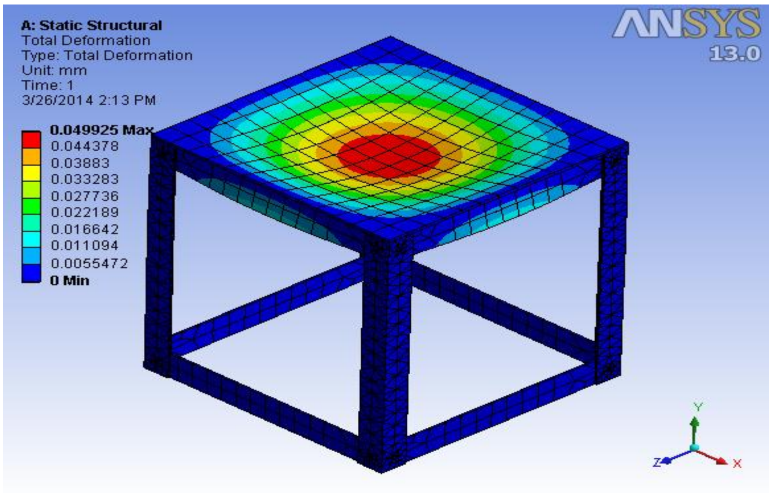


Fig.7: Graphical variation for the amplitude by FFT and ANSYS for model 0.3x0.3x0.3 for height 50 cm

The results below show the deformation, maximum principal stresses and the stress across the ZZ direction for the model 0.3X0.3X0.3 m in the ANSYS.



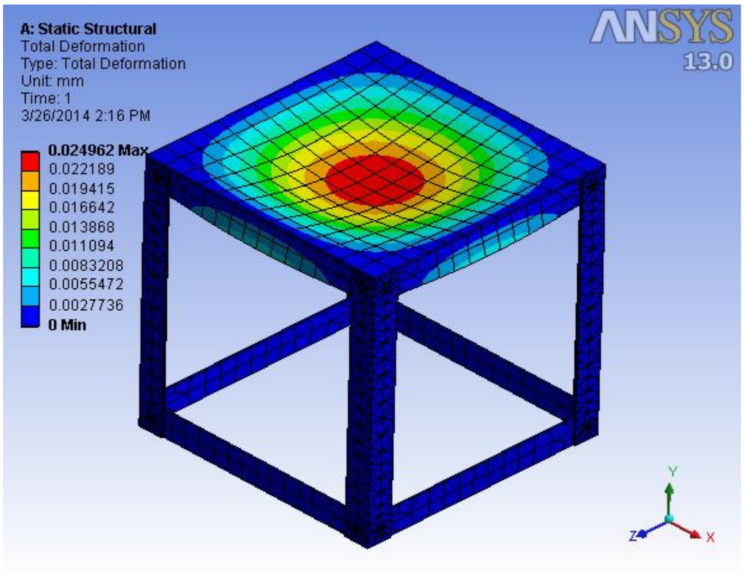


Fig.8 -Amplitude/deformation under 1.0 kg and 2.0 kg impact load for height 0.05 m by ANSYS

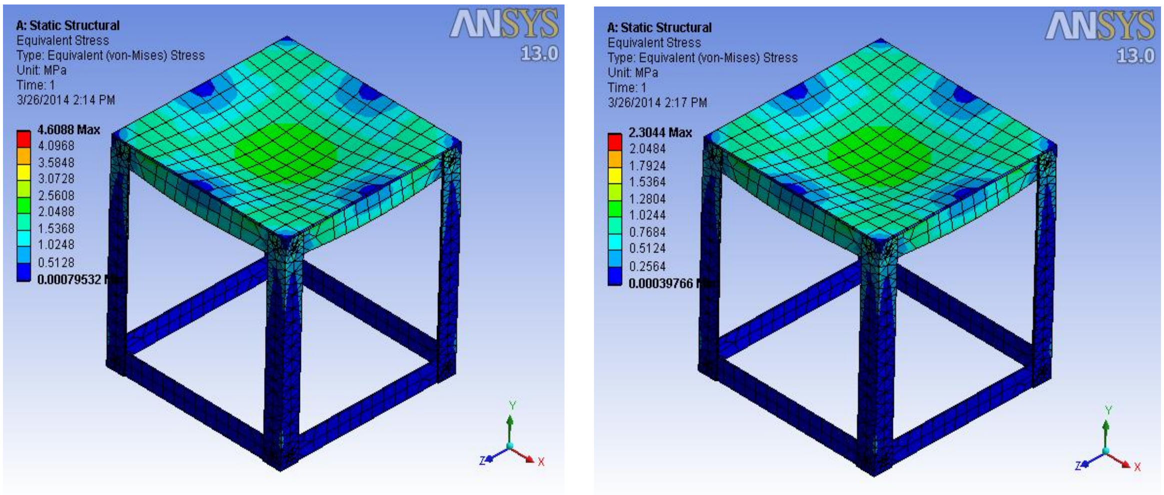


Fig.9-Equivalent elastic stresses

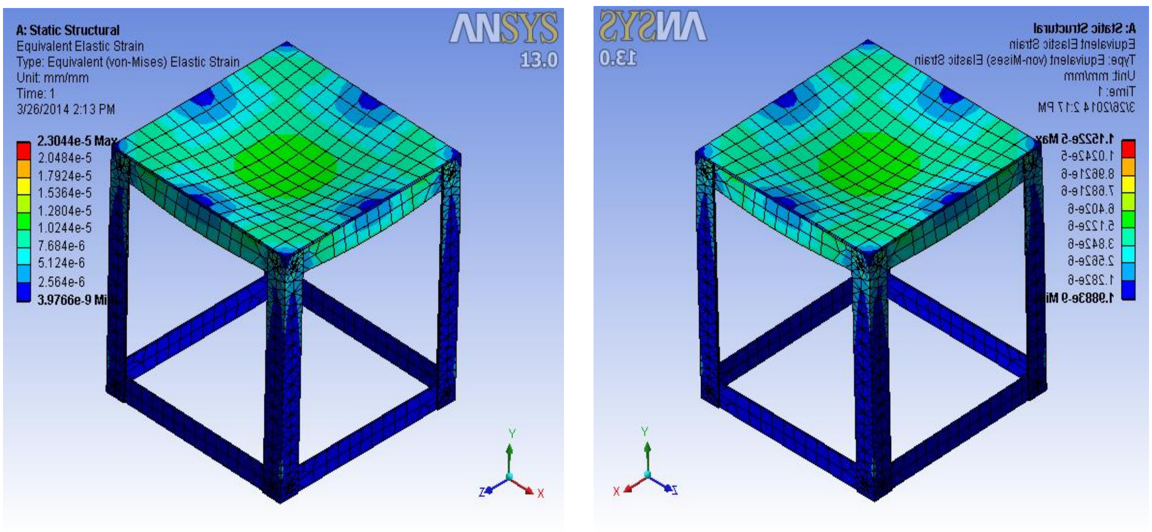


Fig.10-Equivalent elastic strains

IV. Conclusion

1. The impact pressure develops larger kinetic energy in the structure which ultimately produces large amount of displacement in the structure.
2. The duration of the impact load, the maximum amount of impact and the maximum midspan deflection are directly proportional to the height of drop of load.
3. The flexural rigidity of the structure affects the duration of the impact load, the maximum amount of impact and the maximum midspan deflection.
4. The results produced by using Ansys software are too closed to the experimental results.
5. Finite element method by using Ansys software is capable of investigating the possible damage modes of RC frame subjected to impact loading.

V. Future Scope

There is a scope of carrying out same project work for the frame made up of different material. The drop of height and the number of bays and storeys may also vary. One can apply the lateral loading to the structure as it may suppose to expose earthquake loading, wind loading etc.

REFERENCES

Journal Papers:

[1] Yimin Zhu,Engineering Method Design Ductile Structure under Impact Load, Beijing, China, August 7-12, 4896P; pp. 267-273.

Books:

[2] John G. Proakis and Dimitris G. Manolakis, Digital signal processing (3rd edition, Pearson Prentice Hall publication)

Thesis:

[3] Eshan Verma and G. Ashish, Vibrational analysis of framed structures, National Institute of Technology, Rourkela, India. May 2011.

Proceeding Papers:

[4] Wahid Arif, Numerical Analysis of the Dynamic Response of Masonry Structures Subjected to Impact Loading, School of Civil Engineering, Faculty of Engineering.

[5] Norman Jones, The Response of Structures Subjected to Large Impact Loads, Ship structures symposium, Arlington, Virginia,Nov.1993.