ISSN 2229-5518

Dynamic Response of 3D Steel Frame

Sunil B1 and Prashanth Sungar2
1(PG Student, Department of Civil Engineering, M.s Ramaiah Institute Of Technology, Bangalore India) 2(Professor, Department of Civil Engineering, M.s Ramaiah Institute Of Technology , Bangalore India)

ABSTRACT: The main objective of this study is to investigate the dynamic response of 3D steel frame with bracings under dynamic loads. A 3D steel frame with bracings is fabricated as per the design. The steel frame is tested for seismic loads using shake table. Tests are conducted to evaluate the performance of the bare frame and also the frame with additional mass (placing 270kg and 540kg on the frame) under similar seismic conditions. Strain gauges are mounted on the steel structure in order to find out the strain at the respective point. The dynamic properties and response of the steel frame for different shake table tests are obtained and attempt has been made to compare the experimental shake table test results with the numerical results obtained from computer software package, ANSYS 11.0. From this software, response at any location or point on the structure can be obtained in addition to parameters such as modal frequencies and mode shapes. Further, ANSYS 11.0 is used to study the performance of the 3D steel frame for seismic loading conditions corresponding to different zones as per IS: 1893(part-1): 2002 by response spectrum analysis and response of the steel frame is compared for different zones.

Keywords: Steel frame, Dynamic analysis, ANSYS, Shake Table, Seismic Zones

———————————— ◆ ————————————

1. INTRODUCTION

Earthquakes are phenomena that result from the sudden release of stress in rocks that radiate dynamic waves. At the Earth's surface, earthquakes may manifest themselves by a shaking or displacement of the ground and sometimes tsunamis, which may lead to loss of life and destruction of property. Most naturally occurring earthquakes are related to the tectonic nature of the Earth. Such earthquakes are called tectonic earthquakes. The Earth's lithosphere is a patch-work of plates in the majority of tectonic earthquakes originates at depths not exceeding a few tens of miles. Earthquakes occurring at boundaries of tectonic plates are called intraplate earthquakes, while the less frequent events that occur in the interior of the lithospheric plates are called intraplate earthquakes. The consequences of earthquake events are well known to the public: thousands of persons are killed or injured each year, thousands are homeless, heavy damage to the building stock, complete disruption of the infrastructure, irreversible damage to the cultural heritage, very large indirect costs resulting from business interruption, loss of revenues, and interruption of industrial production. Recent Earthquakes have clearly demonstrated that the houses, bridges, public buildings constructed in many third world countries are not engineered to resist even moderate earthquakes. Recently in India, earthquakes caused huge economic losses and death toll, however not much attention is given in preventing such structural damages caused by earthquakes. Prediction of time of occurrence, location and intensity of future earthquakes are unfortunately not yet possible. Recent earthquakes have shown that effective prevention has to be based mainly on adequate design, construction and maintenance of new civil engineering structures, and retrofitting of existing structures and monuments lacking appropriate dynamic resistance characteristics. Since so many catastrophes caused by severe earthquakes were experienced in the past, it is essential that the construction industry, government and people should be aware of the danger and should be prepared against earthquakes by constructing earthquake resistant structures. However the recent earthquakes proved once again that no lesson was learnt from the past catastrophes. Many of the collapses or heavy structural damages were due to poor structural systems. Structural systems that do not have frames with enough shear and/or flexural strength may be one of the common reasons of damage due to earthquakes. Experience in past earthquakes has demonstrated that many common buildings and typical methods of construction lack basic resistance to earthquake forces. In most cases this resistance can be achieved by following simple, inexpensive principles

ISSN 2229-5518

of good building construction practice. Adherence to these simple rules will not prevent all damage in moderate or large earthquakes, but life-threatening collapses should be prevented, and damage limited to repairable proportions. The actual capacity of these structures and their ability to withstand moderate and strong earthquakes needs to be evaluated using accurate methods for predicting the behaviour of structures subjected to dynamic loads. Historically, several different methods have been used for the validation of the dynamic capability of structures that had been designed. Earlier methods usually involved some form of static calculations to estimate the forces generated during a dynamic event of a given ground acceleration, and then comparing this force to the capability of the structure, which may have been derived from calculations or from actual measurements. Extensive experimental and analytical research on steel frames is being carried out worldwide in the last 50 years to establish design procedures that would realistically predict structural behaviour during an earthquake. These methodologies have been verified mainly using static, cyclic or pseudo-dynamic tests.

2. OBJECTIVES

The main objective of this project is to evaluate the response of steel frame under dynamic loading. The steel frame is fabricated and tested for dynamic loads. Tests are also proposed to evaluate the performance of the frame with additional loads under similar dynamic conditions. Strain gauge is mounted on the steel structure in order to find out the strain at the respective point. The dynamic characteristics of the test structure are determined by computational modeling before performing the shake table experiments. Computer software package like ANSYS is used to obtain the performance of structure under earthquake loading. From this software, response time histories at any location or point on the structure can be obtained in addition to parameters such as modal frequencies and mode shapes. The project is taken up to study the performance of 3D-steel frame structures designed as per relevant standards for dynamic loading condition corresponding to zone 4 and zone 5 as per IS-1893.

3. SCOPE OF THE WORK

A research project has been taken up at CPRI to study the response of 3D steel frame with bracings under varying loads to evaluate its earthquake resistance. In this inspection on 3D steel frame with bracings was fabricated. Tests are proposed to evaluate the performance of the frame with additional loads (placing 270kg and 540kg mass respectively on the frame) under similar dynamic conditions. The dynamic properties of the test structure are to be determined by computational modeling before performing the shake table experiments. Finally the test results are compared with analytical results. The most natural testing concept is the use of a shaking table. The test on a shaking table has the advantage of being dynamically similar to a real earthquake event. Earthquake Engineering Laboratory of CPRI has the state-of-the-art tri-axial shaker system for earthquake simulation.

4. METHODOLOGY

To evaluate the dynamic performance of 3D steel frame with bracings subjected to earthquake loading. A model of 3D steel frame is fabricated using channel and angle sections. In this thesis, the performance of steel frame is evaluated by both analytical and experimental methods.

Analytical methods using software ANSYS

- Natural frequencies and mode shapes are obtained from modal analysis.
- Stresses in structure are determined by carrying response spectrum analysis. (Acceleration in horizontal X and Y direction) with experimental damping values.

Experimental methods

- Resonance search tests.(Sine Sweep Test)
- Response spectrum tests.

1. In Resonance search test (sine sweep test), the frequency is varied from 1-50Hz under constant acceleration. The resonant frequencies within the frequency range of interest

ISSN 2229-5518

are obtained. This test is customarily used as an explanatory test, with low 'g' input level before dynamic testing. Damping values are obtained from this experiment.

2. In Response spectrum test, a typical response spectrum is played on the shake table.

3. Comparison of the analytical results with the experimental results.

5. DESCRIPTION OF STEEL FRAME

3D-Steel Frame with Bracings

The 3D-steel frame with bracings is fabricated for the analysis and testing. Beams and Columns are fabricated using channel section and the bracings using angle section. Length of the beam is 2.4m and the height of the column is 2m. The dimensions of the steel frame are given in meters is shown in fig 5.1 and fig 5.2. The boundary conditions of the supports are fixed. Members section properties and material properties are shown in table 5.1 and 5.2.

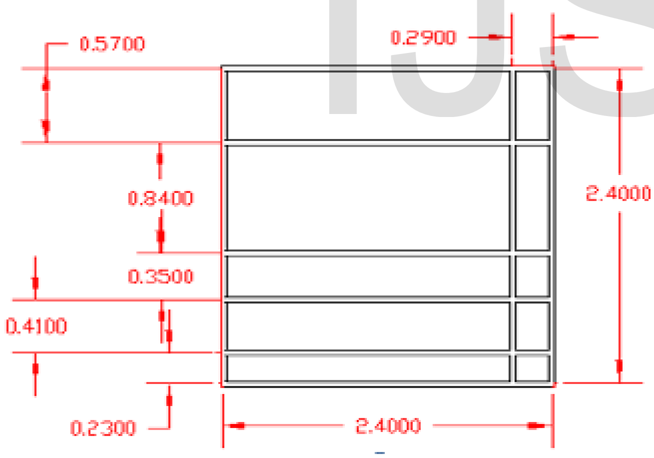


Figure 5.1: Top view of 3D-Steel frame with bracings

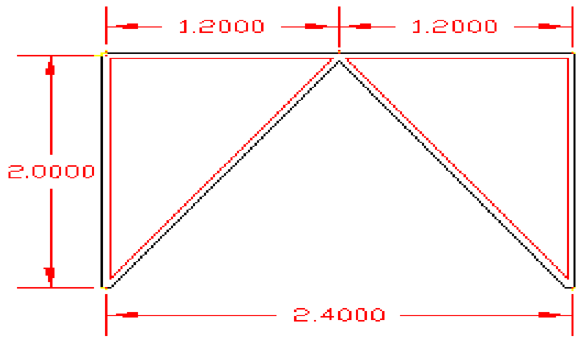


Figure 5.2: Front view of 3D-Steel frame with bracings

Material	Concrete	Steel
Modulus of elasticity N/m^2	2.5 x 10^{10}	2 x 10^{11}
Poisons ratio	0.15	0.3
Density KN/m^3	25	78.6

s

Table 5.1: Materials properties

SL NO.	Member	Dimensions
1	Column	ISMC-75
2	Beam	ISMC-75
3	Bracings	ISA-50X50X6

Table 5.2: Member Section Properties

6. FE ANALYSIS OF STEEL FRAME

INTRODUCTION

The Finite Element Analysis (FEA) rapidly grew as the most useful numerical analysis tool for engineers because of its natural benefits over prior approaches. The main advantages are that it can be applied to arbitrary shapes in any number of dimensions. The shape can be made of any number of materials. The material properties can be non-homogeneous (depend on location) and/or anisotropic (depend on direction). The FEA provides a standard process for converting governing energy principles or governing differential equations in to a system of matrix equations to be solved for an approximate solution. For linear problems such solutions can be very accurate and quickly obtained.

Finite Element Analysis (FEA) can be used to study the structural dynamic characteristics and seismic performance of vibrating mechanical systems, the understanding of which is paramount to any root-cause failure study involving excessive vibrations. The Finite Element technique can be used for existing equipment or to evaluate the dynamic response of structures prior to fabrication.

A finite element model is typically much more detailed than an experimental model. Finite element models contain thousands of degrees of freedom (DOFs) while most experimental models include hundreds of DOFs or less.

STEPS IN FINITE ELEMENT METHOD

1. Discretisation of the continuum (Structural system)
2. Selection of displacement model
3. Derivation of element stiffness matrix
4. Assembly of the algebraic equations for the entire continuum
5. Applying the boundary conditions
6. Solution for various displacements

In this dissertation work, an attempt is made to find the dynamic characteristics by modal analysis for different conditions and response of the steel frame subjected response spectra loading corresponding to Zones IV and V as per IS: 1893(part-1): 2002 by response spectrum analysis using the FEA software package ANSYS.

ANSYS

ANSYS is a commercially available software package for both computer aided design (CAD) and Finite Element Analysis (FEA) made by ANSYS Inc, Canonsburg. It is a general-purpose finite element analysis computer program. “General purpose” means that the software addresses a wide range of engineering problem-solving requirements as compared to specialized programs, which concentrate on particular types of analysis. ANSYS offers a comprehensive range of engineering simulation solution sets providing access to virtually any field of engineering simulation that a design process requires. Companies in a wide variety of industries use ANSYS software. The tools put a virtual product through a rigorous testing procedure (such as crashing a car into a brick wall, or running for several years on a tarmac road) before it becomes a physical object.

Finite element analysis involves three stages of activity,

1. Pre-processing
2. Processing and
3. Post-processing.

- **Pre-processing** involves the preparation of data, such as nodal coordinates, connectivity, boundary conditions, and loading and material information.
- **Processing** stage involves stiffness generation and solution of equations, resulting in the evaluation of nodal

ISSN 2229-5518

variables. Other derived quantities, such as stress or strain, may be evaluated at this stage.

- **Post-processing** stages deals with the presentation of results. Typically, the deformed configuration, mode shapes, accelerations and stress distribution are computed and displayed at this stage. A complete finite element analysis is a logical interaction of these three stages.

MODELING IN ANSYS (PRE-PROCESSING)

ANSYS provides a number of element types for modeling steel frame in its element library. **Beam 188 2 noded element** is used for modeling beam and column and for bracings, **LINK8 elements** are used**. Structural Mass** (3D Mass 21) is used for modeling added mass on the steel frame. The geometry of the model is developed as per the dimensions. Element properties are assigned to the steel frame as in **Table 6.1 & 6.2**. The boundary condition is, fixed at the ends. The geometrical model of Steel Frame is as shown in **Figure.6.1**.

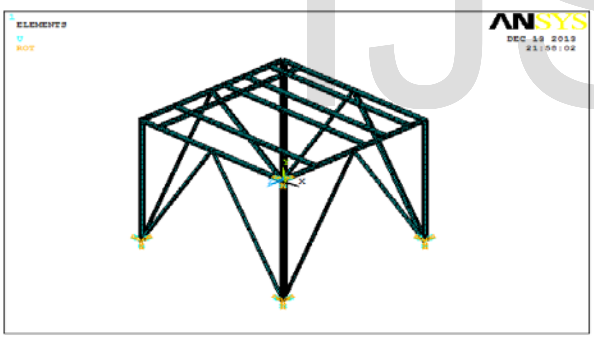


Figure 6.1: Geometrical model of 3D Steel frame

MASS21 Element Description

MASS21 is a point element having up to six degrees of freedom: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. A different mass and rotary inertia may be assigned to each coordinate direction.

ANALYSIS IN ANSYS (PROCESSING)

The developed piping system model is taken to the processing stage for analysis. Using ANSYS as the processor, a series of analysis are conducted simulating different field conditions by gathering all specified information about the problem using different solvers available within the ANSYS software for different types of analysis.

POST ANALYSIS IN ANSYS (POST-PROCESSING)

After the analysis of the structure/piping system in ANSYS, the results of the analysis are retrieved in the post-processor of ANSYS. In general post processor (POST1), results can be graphically visualized or listed the results in tables. All the results of static analysis, modal analysis and response spectrum analysis (nodal displacements, stresses, natural frequencies, mode shapes, accelerations etc) can be retrieved using the general post processor.

Material	**Concrete**	**Steel**
Modulus of elasticity N/m^2	2.5 x 10^{10}	2 x 10^{11}
Poisons ratio	0.15	0.3
Density KN/m^3	25	78.6

Table 6.1 Element Properties

SL NO.	**Member**	**Dimensions**
1	Column	ISMC-75
2	Beam	ISMC-75
3	Bracings	ISA-50X50X6

Table 6.2: Member Section Properties

MODAL ANALYSIS

Modal Analysis is the basic study in the dynamic characteristics of structures. This analysis characterizes the dynamic properties of an elastic structure by identifying its mode of vibration. The response of the structure is different at each of the different natural frequencies. These deformation patterns are called mode shapes. Both the natural frequency (which depends on the mass and stiffness distributions in structure) and mode shapes are used to help the design of structural system mainly for vibration applications. Modes are inherent properties of a structure and are determined by the material properties (mass, damping, and stiffness), and boundary conditions of the structure. Each mode is defined by a natural (modal or resonant) frequency, modal damping, and a mode shape (i.e. the so-called "modal parameters"). If either the material properties or the boundary conditions of a structure change, its modes will change. For instance, if mass is added to a structure, it will vibrate differently.

Modal analysis is a method or a process or a technique to describe a structure in terms of its natural characteristics which are (its dynamic properties),

- Natural frequency
- Mode participation factors
- And Mode shapes

The modal analysis calculates the natural modes of the discretised model, not those of the real continuous system. However the discretised modes are close to the continuous ones and for a mode number the accuracy improves as more and more elements are used to model the system. For any given level of discretisation the accuracy is better for the lower modes and progressively worsens as we go to higher and higher modes. The highest numbered modes are unlikely to be realistic since they are oscillations whose wavelengths are of the same order as the segment length.

Modal analysis uses the overall mass and stiffness of the structure to find various periods at which it will naturally resonate. These periods of vibration are very important to note in earthquake engineering, as it is imperative that a building's natural frequency does not match the frequency of expected earthquakes in the region in which the structure is to be constructed. If a structure's natural frequency matches the earthquake's frequency, the structure may continue to resonate and experience structural damage.

Following are the benefits of modal analysis, It allows the design to avoid resonant vibrations or to vibrate at a specified frequency. It gives engineers an idea of how the design will respond to different types of dynamic loads. Because a structure's vibration characteristics determine how it responds to any type of dynamic load, it is always mandatory to perform modal analysis first before performing any other dynamic analysis.

7. SHAKE TABLE TESTING

TRI-AXIAL SHAKER SYSTEM AT CPRI

Earthquake engineering laboratory housing the tri-axial shaker system with six degrees of freedom, capable of performing a diverse range of seismic qualification test requirements on equipment, sub-assemblies and components as per National / International standards has been established at Central Power Research Institute CPRI, Bangalore in the

ISSN 2229-5518

year 2003. The tri-axial shaker system consisting of a shaking-table is a unique facility that can strictly simulate the earthquake ground motion without any distortion. The shaking table can vibrate in one axis to three axes with six degrees of freedom. The advanced control system allows the reproduction of earthquake ground motions with high fidelity and little distortion. Table shows salient features of high-performance shaker system at CPRI, Bangalore. The seismic qualification tests are being conducted using the tri-axial earthquake simulation system, which features a 10-ton payload capacity shake table of all-welded steel construction. An advanced control system allows the reproduction of earthquake ground motions with high fidelity.

Salient Features of Shaking Table Facilities of CPRI, Bangalore

Sl.No	Item	Performance
1.	Maximum payload	10 tons
2.	Table dimension	3m × 3m
3.	Exciting direction (Simultaneous / Sequential)	X, Y, Z
4.	Degrees of Freedom	Six, 3 translational and 3 rotational
5.	Max. Height of the specimen	10 m
6.	Displacement/ Max. Stroke	
	X &Y Direction	± 150 mm
	Z - Direction	± 100 mm
7.	Velocity (X, Y&Z direction)	1000 mm/s
8.	Acceleration	±1 g (X, Y&Z direction)
9.	Maximum specimen channels	128
10.	Frequency range	0.1 to 50 Hz
11	Yawing moment	10 ton. per m
12	Overturning moment	40 ton. per m
13	Actuators	
	Vertical	4 nos. of 180 KN
	Horizontal	4 nos. of 150 KN

Shake Table Actuator (a) Horizontal, (b) Vertical

ISSN 2229-5518

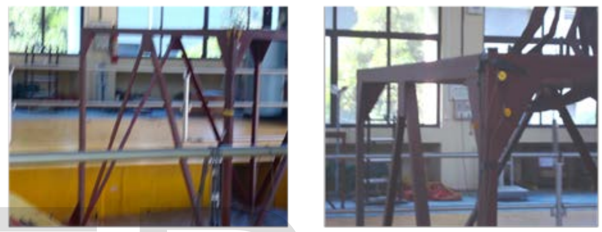
OVERHEAD CRANE PLACING THE STEEL FRAME ON THE SHAKE TABLE

TEST SETUP

FITTING THE 3D-STEEL FRAME ON THE SHAKE TABLE USING BOLTS

TEST SET UP

The Steel frame structure is fabricated outside the laboratory and suitable arrangement had been made to move the frame structure to the shake table. Precautions are taken such that no structural damage occurs during transportation and placing of the structure on the shake table. 50KN forklift is employed to carry the steel frame into the laboratory and then the overhead crane is used to place the specimen on the shake table as shown in figure and then steel frame is fitted on the shake table using bolts as shown in figure. At the specified locations accelerometers are mounted to measure the response of the structure in terms of acceleration and strain gauge is fixed at the critical location in order to measure strain which have been discussed below.



At the specified locations on the scaled model accelerometers are mounted as shown in the figures. During testing the output of accelerometers are recorded as acceleration response. Three accelerometers are fixed in the three mutually perpendicular directions.

3D STEEL FRAME WITH BRACINGS

Table Time History of Accelerometers in X-direction:

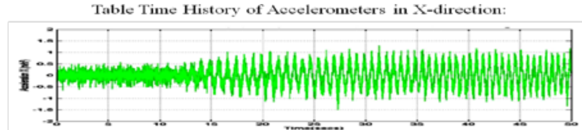


Table time history of accelerometer in X-direction for 0.1g

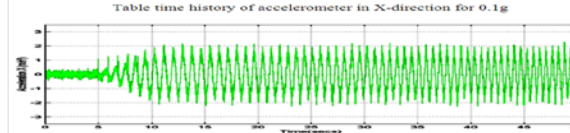


Table time history of accelerometer in X-direction for 0.2g

Table Time History of Accelerometers in Y direction

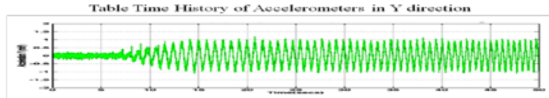


Table time history of accelerometer in Y-direction for 0.1g

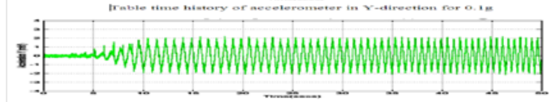
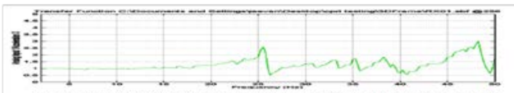
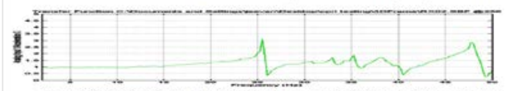


Table time history of accelerometer in Y-direction for 0.2g

Typical transfer functions in X- direction:

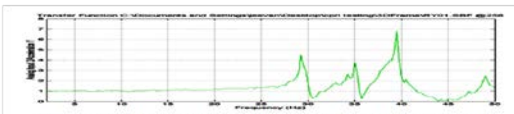


TF of Accelerometer in X-direction at 0.1g, Resonance Frequency =25.50Hz

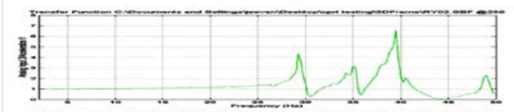


TF of Accelerometer in X-direction at 0.2g, Resonance Frequency =25.50Hz

Typical transfer functions in Y- direction:

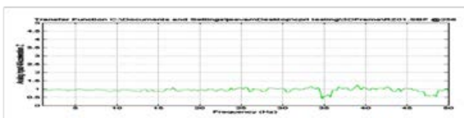


TF of Accelerometer in Y-direction at 0.1g, Resonance Frequency =29.25Hz

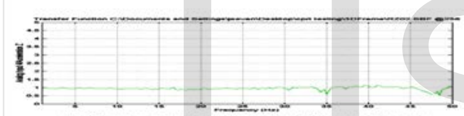


TF of Accelerometer in Y-direction at 0.2g, Resonance Frequency = 29.25Hz

Typical transfer functions in Z- direction:



TF of Accelerometer in Z-direction at 0.1g, No Resonance Frequency



TF of Accelerometer in Z-direction at 0.2g, No Resonance Frequency

3D STEEL FRAME WITH 270KG MASS

Table Time History of Accelerometers in X direction

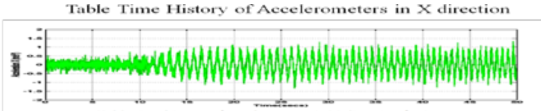


Table time history of accelerometer in X-direction for 0.1g

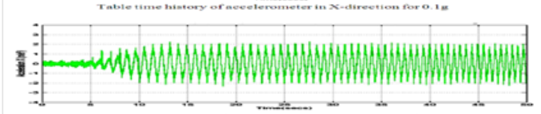
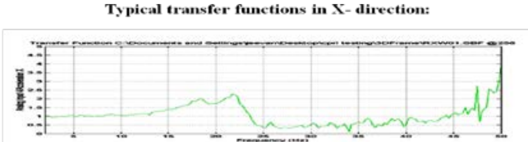
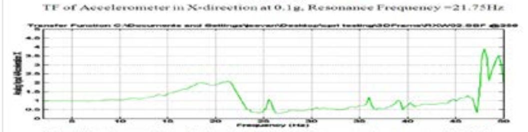


Table time history of accelerometer in X-direction for 0.2g

Typical transfer functions in X- direction:



TF of Accelerometer in X-direction at 0.1g, Resonance Frequency =21.75Hz



TF of Accelerometer in X-direction at 0.2g, Resonance Frequency =21.25Hz

3D-STEEL FRAME WITH 540KG MASS

Table Time History of Accelerometers in X direction

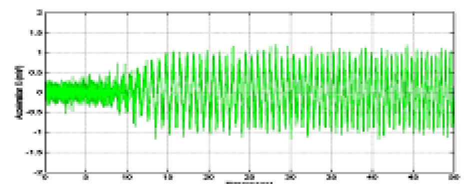
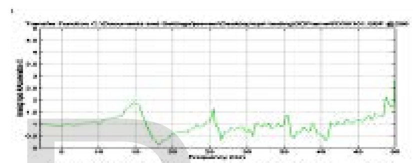


Table time history of accelerometer in X-direction for 0.1g

Typical transfer functions in X- direction:



TF of Accelerometer in X-direction at 0.1g, Resonance Frequency = 14.75 Hz

8. RESULTS AND DISCUSSION

The present study has been carried out in connection with the performance investigation of 3D-Steel frame with and without mass and to evaluate the performance of the frame with additional loads (placing 270kg and 540kg mass respectively on the frame) under the similar seismic conditions. Modal analysis of 3D-Steel frame with bracings is carried out using the software ANSYS software. From these analyses the modal parameters such as natural frequencies and mode shapes are obtained. Resonance search tests are conducted on the scale down models using shake table to evaluate the natural frequencies, magnification factor and damping values experimentally.

FREQUENCY COMPARISONS:

Resonance frequencies obtained using software and shake table tests for the model are compared in table 1 and 2.

Table 1: Natural frequencies obtained from FEA packages and Sine sweep test in X-axis

ISSN 2229-5518

Models	Natural Frequencies (Hz)	
	ANSYS	SINE SWEEP TEST
3D-steel frame without mass	23.920	25.5
3D-steel frame with 270kg mass	20.868	21.75
3D-steel frame with 540kg mass	17.958	14.75

Table 2: Natural frequencies obtained from FEA packages and Sine sweep test in Y-axis

Models	Natural Frequencies (Hz)	
	ANSYS	SINE SWEEP TEST
3D-steel frame without mass	27.422	29.25

9. CONCLUSIONS

- Literature review on the works done by earlier researchers in evaluating the seismic performance of steel frame is carried out. The details of their findings are brought out in this report.
- A brief introduction on earthquakes and the philosophy of structural design are explained.
- FE model of steel frame is developed using ANSYS software using beam and link elements for modeling beams, columns and bracings.
- Dynamic characteristics of 3D steel frame with bracings for different conditions are evaluated using software ANSYS. The mode shapes and the resonance frequencies obtained from this software are tabulated
- The experimental and numerical results are compared. The natural frequencies of the 3D-Steel frame obtained from shake table tests are closely matching with the values obtained using the software ANSYS 11.0. It can be concluded from the experimental and numerical results that the steel frame's dynamic properties change with additional mass. Natural frequency decreases as the mass on the structure increases.

10. REFERENCES

- **Praveen.K.Malhotra (2001), "Response spectrum of incompatible acceleration, velocity, and displacement histories", Earthquake engineering and structural dynamics; Vol 30; pp 279-286.**
- **Stefano Sorace (1998), "Seismic damage assessment of steel frames", Journal of structural engineering, Vol 124, No 5, pp 531-540.**
- **Stephen.P.Schneider and Ala Amidi (1998), "Seismic behavior of steel frames with deformable panel zones", Journal of structural engineering, Vol 127, No 1, pp 35-42.**
- **Kihak Lee and Douglas.A.Foutch (2002), "Performance evaluation of new steel frame buildings for seismic loads", Earthquake engineering and structural dynamics; Vol 31; pp 653-670.**
- **Sudip Paul, C.V.R. Murty and Sudhir.K.Jain (2000), "State-of –the-art review of steel moment resisting frames-Part I: general consideration and**

ISSN 2229-5518
stability provisions", Journal of structural engineering, Vol 27, No 1, pp 23-32.

- **Sudip Paul, C.V.R. Murty and Sudhir.K.Jain (2000), "State-of –the-art review of steel moment resisting frames-Part II: Strength and Drift criteria", Journal of structural engineering, Vol 27, No 2, pp 117-124.**

- **S. Krishnan (2008), "Modeling steel moment frame and braced frame buildings in three dimensions using FRAME3D", 14th World conference on earthquake engineering.**

- **P.fajfar "Structural analysis in earthquake engineering-A breakthrough of simplified non-linear methods" 12th European conference on earthquake engineering.**