14. Seismic Behaviour of a Composite Building

14.1 Structure's Important Dynamic Properties

The principal dynamic properties of importance to structural earthquake response are the structure's modal properties and its damping. The simplest type of structure is the so-called single degree of freedom (SDOF) structure. A classical model of a SDOF structure consists of a single concentrated mass, M, on top of a cantilevered column. **Figure 14.1** represents such a model. If, as shown in the figure, a force, F, is statically applied to the mass, the column will deform laterally, allowing the mass to displace in the direction of the applied force. If the column has stiffness, K, it will deflect to a displacement x, given by Eq. (14.1).

$$x = F/K \qquad \text{... (14.1)}$$

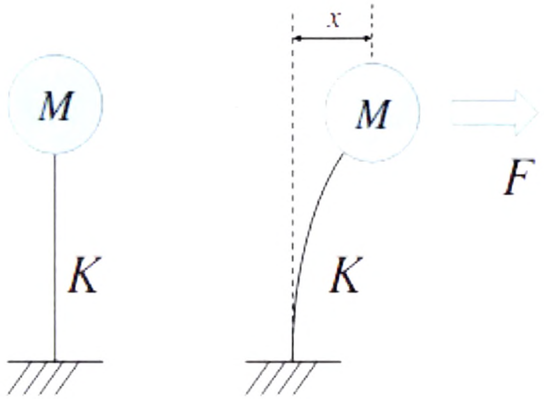


Fig. 14.1 Mathematical Model of SDOF Structure

If the mass is maintained in equilibrium, the column will experience a shear force equal and opposite to the applied external force, F. If this force is suddenly removed, the structure will continue to exert a force F, on the mass, a back-and-forth vibration, with maximum amplitudes +x and –x at a unique natural frequency given by Eq. (14.2).

$$f = \frac{1}{2\pi}\sqrt{\frac{K}{M}} = \frac{1}{2\pi}\sqrt{\frac{K * g}{W}} \qquad \text{... (14.2)}$$

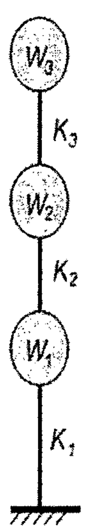
In this equation, W is the weight of mass M, g is the acceleration due to gravity, and the frequency, f, has units of cycles/second.

In earthquake engineering, it is common to use the inverse of the frequency, termed the period, which is the time, in seconds, it would take the structure to undergo one complete cycle of free vibration from +x to –x to +x. This period, which is usually represented by the symbol T, is given by the Eq. (14.3).

$$T = 2\pi\sqrt{\frac{W}{K * g}} \qquad \text{... (14.3)}$$

Multi-storey structures must be treated as multi-degree of freedom (MDOF) structures. The earthquake response of such structures can be calculated using a stick model with the mass in each story lumped at a single point, and the stiffness of the seismic load resisting system in each story can be represented by a single translational spring, as illustrated in **Fig. 14.2** for a three-story structure [118].

MDOF structures will have one natural mode of vibration, i, for each degree of freedom, j. Each mode of vibration will have a unique period, Ti, and a unique deformed shape, Φ_i, at which it will undergo free vibration. These deformed shapes are called mode shapes. **Fig. 14.3** illustrates the three mode shapes for the three-storey structure as shown in **Fig. 14.2**.



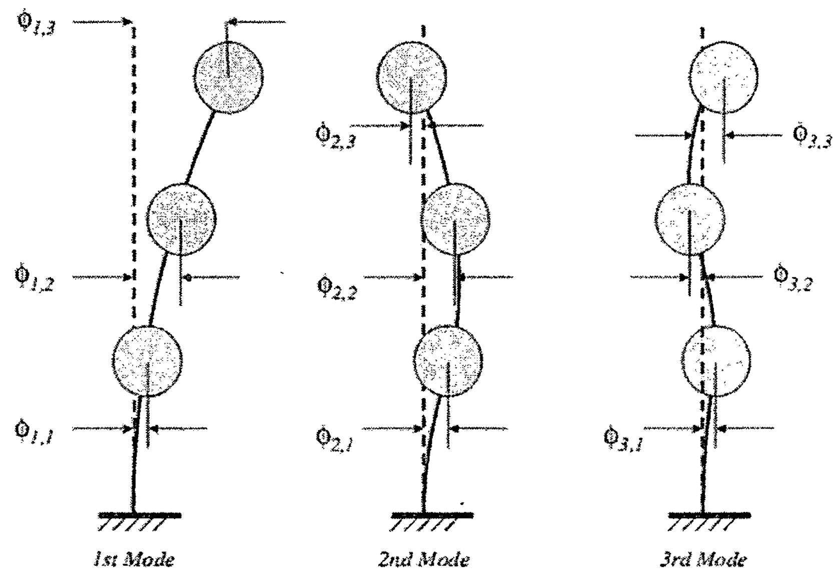


Fig. 14.2 Model Structure **Fig. 14.3 Modal Shapes for a Structure with 3DOF**

In any natural mode shape for MDOF structure some of the masses move more than others. As a result, only a portion of the structure's mass is effectively mobilized during vibration in a particular mode. The effective or modal mass M_i for mode i is given by Eq. 14.4.

$$M_i = \frac{\left(\sum m_j \emptyset_{i,j}\right)^2}{\sum m_j \emptyset_{i,j}^{\ 2}} \qquad \text{... (14.4)}$$

In this equation, M_j is the lumped mass at degree of freedom j and Φ_i, j is the relative deformed shape displacement for mode i at degree of freedom j. The sum of the modal masses for all structure's modes is equal to the structure's total mass.

When performing linear or elastic analysis of a building's response to earthquake shaking, it is common to assume that it inherently has 5% of the critical damping. In actuality, most steel structures have somewhat less damping than this when behaving elastically. The amount of damping that can actually be mobilized depends on many factors, including the amplitude of vibration and the amount of damage, if any, that occurs.

14.2 PROCEDURES FOR SEISMIC ANALYSIS

Structural analysis methods can be divided into the following four categories.

1) Linear Static Analysis (Equivalent Static Analysis)
2) Linear Dynamic Analysis (Response Spectrum Analysis)
3) Non-linear Static Analysis (Push Over Analysis)
4) Non-linear Dynamic Analysis (Time History Analysis)

14.2.1 LINEAR STATIC ANALYSIS

As linear static analysis is already discussed in Chapter 13 in detail, it is not repeated here.

14.2.2 LINEAR DYNAMIC ANALYSIS

The standard method used in design is the modal response using a design spectrum. This is a linear method in which the inelastic behaviour is considered in the definition of the design spectrum, through the use of a behaviour factor. This method is applicable to all types of buildings, be they regular or irregular in plan and/or elevation. This approach permits the multiple modes of response of a building to be taken into account. For each mode, a response is read from the design spectrum, based on the modal frequency and the modal mass, and they are then combined to provide an estimate of the total response of the structure.

There are computational advantages in using the response spectrum method of seismic analysis for prediction of displacements and member forces in structuructure. The method involves the calculation of only the maximum values of the displacements and member forces in each mode using design spectra that are the average of several earthquake motions.

The response spectrum is the "design acceleration spectrum", which refers to the average smoothened plot of maximum acceleration called spectral acceleration coefficient, as a function of the frequency of the structure for a specified damping ratio for earthquake excitations at the base for a single degree freedom system.

For three dimensional seismic motions, the typical modal equation is written as

$$\ddot{y}(t)_n + 2\zeta_n\omega_n\dot{y}(t)_n + \omega_n{}^2y(t)_n = p_{nx}\ddot{u}(t)_{gx} + p_{ny}\ddot{u}(t)_{gy} + p_{nz}\ddot{u}(t)_{gz} \quad \text{... (14.8)}$$

Where the three Mode Participation Factors are defined by $P_{ni} = - \Phi_n{}^T M_i$ in which i is equal to x, y or z. For input in one direction only, Eq. (4.8) is written as

$$\ddot{y}(t)_n + 2\zeta_n\omega_n\dot{y}(t)_n + \omega_n{}^2y(t)_n = p_{ni}\ddot{u}(t)_g \quad \text{... (14.9)}$$

Given a specified ground motion $\ddot{u}(t)_g$, damping value and assuming $p_{ni} = -1.0$ it is possible to solve Eq.(14.9) at various values of ω and plot a curve of the maximum peak response $y(\omega)_{MAX}$. For this acceleration input, the curve is by definition the **displacement response spectrum** for the earthquake motion. A different curve will exist for each different value of damping. A plot of $\omega y(\omega)_{MAX}$ is defined as the **pseudo-velocity spectrum** and a plot of $\omega^2 y(\omega)_{MAX}$ is defined as the **pseudo-acceleration spectrum.**

14.2.2.1 Calculation of Modal Response

The maximum modal displacement, for a structural model, can now be calculated for a typical mode n with period T_n and corresponding spectrum response value $S(\omega_n)$. The maximum modal response associated with period T_n is given by

$$y(T_n)_{max} = \frac{s(\omega_n)}{\omega_n{}^2} \quad \text{... (14.10)}$$

The maximum modal displacement response of the structural model is calculated from

$$u_n = y(T_n)_{MAX}\emptyset_n \quad \text{... (14.11)}$$

The corresponding internal modal forces are calculated from standard matrix analysis using the same equations as required in static analysis.

14.2.2.2 The CQC Method of Modal Combination

The most conservative method that is used to estimate a peak value of displacement or force within a structure is to use the sum of the absolute of the modal response values. This approach assumes that the maximum modal values, for all modes, occur at the same point in time. Another very common approach is to use the Square Root of the Sum of the Squares, SRSS, on the maximum modal values in order to estimate the values of displacement or

forces. The SRSS method assumes that all of the maximum modal values are statistically independent. For 3D structures, in which a large number of frequencies are almost identical, this assumption is not justified.
The relatively new method of modal combination is the Complete Quadratic Combination, CQC, method that was first published in 1981. It is based on random vibration theories and has found wide acceptance by most engineers and has been incorporated as an option in most modern computer programs for seismic analysis.

The peak value of a typical force can now be estimated, from the maximum modal values, by the CQC method with the application of the following double summation equation:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m} \qquad \dots (14.12)$$

Where, f_n is the modal force associated with mode n. The double summation is conducted over all modes. Similar equations can be applied to node displacements, relative displacements and base shears and overturning moments.

The cross-modal coefficients, p_{mn}, for the CQC method with constant damping are

$$\rho_{nm} = \frac{8\zeta^2(1+r)r^{3/2}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2} \qquad \dots (14.13)$$

where $r = \omega_n / \omega_m$ and must be equal to or less than 1.0. It is important to note that the cross-modal coefficient array is symmetric and all terms are positive.

14.2.2.3 Responce Spectra

According to IS1893:2002, high rise and irregular building must be analysed by the response spectrum method. Response spectrum analysis is performed using mode superposition, where free vibration modes are computed using eigen value analysis. The design acceleration spectrum which is recommended by IS 1893:2002 [114] is shown in **Fig. 14.5** are for 5% damping factor. Damping is the effect of internal friction, slipping, sliding, etc. in reducing the amplitude of vibration and is expressed as a function of critical damping.

The damping recommended for different types of buildings are as follows:

a) Steel and timber structures: 2 to 5% critical (2% is usually adopted)
b) Concrete and brick structures: 5 to 10% critical (5% is usually adopted)
c) Earth structures: 10 – 30% critical

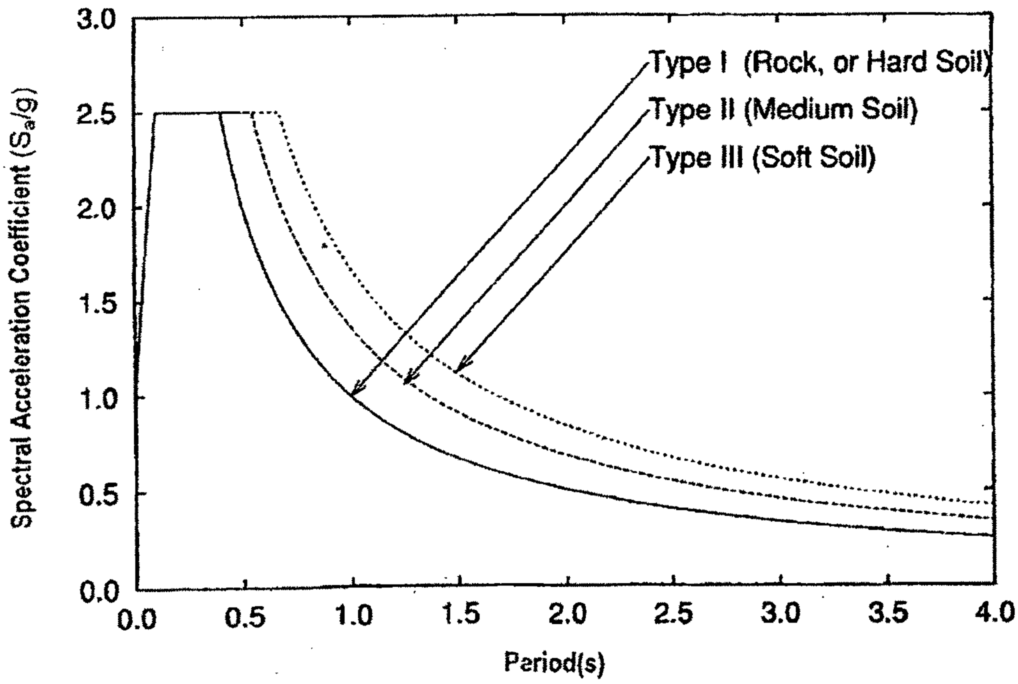


Fig. 14.5 Response Spectra for 5% Damping

14.2.3 NON-LINEAR STATIC ANALYSIS

This approach is also known as "pushover" analysis. A pattern of forces is applied to a structural model that includes non-linear properties (such as steel yield), and the total force is plotted against a reference displacement to define a capacity curve. This can then be combined with a demand curve (typically in the form of an acceleration-displacement response spectrum (ADRS)). This essentially reduces the problem to a SDOF system. The 'Pushover' analysis is a non-linear static analysis carried out under constant gravity loads and monotonically increasing horizontal loads. It is applied essentially:

- To verify or revise the over strength ratio values.
- To estimate the expected plastic mechanisms and the distribution of damage.
- To assess the structural performance of existing or retrofitted buildings.

14.2.4 NON-LINEAR DYNAMIC ANALYSIS

Nonlinear dynamic analysis utilizes the combination of ground motion records with a detailed structural model, therefore is capable of producing results with relatively low uncertainty. In nonlinear dynamic analyses, the detailed structural model subjected to a ground-motion record produces estimates of component deformations for each DOF in the model and the modal responses are combined using schemes such as the square-root-sum-of-squares. In non-linear dynamic analysis, the non-linear properties of the structure are considered as part of a time domain analysis. This approach is the most rigorous, and is required by some building codes for buildings of unusual configuration or of special importance. However, the calculated response can be very sensitive to the characteristics of the individual ground motion used as seismic input; therefore, several analyses are required using different ground motion records.

14.3 Comparison of Seismic Analysis Methods

During earthquakes, buildings are generally subjected to large inertia forces, which cause members of buildings to behave in a nonlinear manner, i.e. stress does not remain proportional to strain in addition to nonlinearity associated with large deformations. Earthquake shaking of the structure is a nonlinear dynamic problem and structural analysis should be able to incorporate the nonlinear behavior of members for evaluating the actual response of structures. Nonlinear analysis requires a lot of input data related to material and properties and loads, which are generally difficult to obtain accurately. Experimental data are not available in sufficient quantity to develop accurate analytical models for analysis procedures to characterize nonlinear dynamic force-deformation behavior of members. Further, the interpretation of analysis results requires a great deal of expertise and in-depth understanding of the nonlinear behavior of structures.

Therefore, the national codes of a few countries recommend nonlinear analysis only for highly irregular and important structures. In comparison, linear dynamic analysis is simpler and adequately captures dynamic behavior in elastic range and therefore is a better indicator of structural performance than ESMA. However, it fails to capture the capacity-related information of structural members, which is only possible with nonlinear dynamic or static procedures. A simple nonlinear static (push over) analysis is being used now-a-days for certain projects, especially those related to seismic strengthening and rehabilitation.

The main purpose of linear dynamic analysis is to evaluate the time variation of stresses and deformations in structures caused by arbitrary dynamic loads. As in any dynamic system, vibrational properties of building can be estimated by solving Eigen value problem given by:

$$[k - \omega_n^2 m]\Phi_n = 0, \text{ where } \omega_n = 2\pi / T_n \qquad \text{... (14.14)}$$

Where k and m are the stiffness and mass matrices of buildings respectively, and ω_n, Φ_n and T_n are the natural frequency, mode shape and natural period of buildings respectively, for the n^{th} mode.

Buildings can vibrate in different mode shapes, as shown in **Fig. 14.3.** There can be as many mode shapes possible as number of dynamic degrees of freedom in the building. Dynamic degrees of freedom in a structure are the number of independent coordinates in which the structure can undergo motion under dynamic forces. Depending upon the building type, only the first few mode shapes may govern the response of the building. Lateral displacement, u at any point on buildings during earthquakes can be expressed as a linear combination of all the mode shapes of buildings as given below:

$$u = \sum_{n=1}^{N} \Phi_n q_n \qquad \text{... (14.15)}$$

where q_n are the n^{th} modal coordinates and N is the total number of modes.

Shear forces on the buildings can be estimated as stiffness times the lateral displacement. Therefore, mode shapes of buildings play an important role in estimating the design base shear for buildings.

The basic assumption in equivalent static method of analysis is that only the first mode of vibration of building governs the dynamics and the effects of higher modes are not significant; therefore, higher modes are not considered in the analysis. Thus, irrespective of whether the building is regular or irregular, ESMA cannot adequately capture the true behavior of multistory building; the design forces for the members may be grossly underestimated. However, several uncertainties and approximations are involved in dynamic analysis in describing the true dynamic loads, estimating the actual material and sectional properties etc. Therefore, dynamic analysis must be used with great caution.

IS : 1893-2002 [114] has divided India into four seismic zones depending upon the seismic hazard associated with different regions and recommends different analysis methods depending upon height, location (zone) and configuration of buildings. ESMA is permitted for regular buildings of height up to 90 m (~ 30 storeys) in lower seismic zones (zones II and III), and of height up to 40 m (~ 13 storeys) in higher zones (zones IV and V). On the other hand, for irregular buildings, ESMA can be used up to height of 40 m (~ 13 storeys) and 12 m (~ 4 storeys) in lower and higher zones respectively. Linear dynamic analysis is required for buildings not covered under the above restrictions.

14.4 SEISMIC DESIGN PROVISIONS FOR COMPOSITE STRUCTURES

Seismic design provisions for composite steel-concrete building construction have recently been introduced in the United States and other countries. Similar to the seismic provisions for other types of construction, the new seismic provisions for composite construction are forced-based provisions, in which the structure is designed to resist reduced lateral forces. The lateral forces used in design are often significantly smaller than those required to maintain the structure elastic. The reduced lateral forces are computed using an equivalent static procedure or a modal spectral analysis often referred to as dynamic procedure. In both procedures the reduced lateral forces result from the use of linear design spectra and response modification factors. Maximum lateral displacements are typically checked near the end of the design process for serviceability requirements, by comparing the computed displacements to an allowable upper limit on the maximum interstory drift. Lateral displacements are computed as the displacements computed with a linear elastic analysis of the structure when subjected to code-specified (reduced) lateral forces multiplied by a displacement amplification factor that is intended to account for the inelastic deformation expected in the structure during severe earthquake ground motions.

The design of buildings in seismic regions is often controlled by lateral stiffness required to limit the maximum interstorey drifts below the maximum allowed by the code. Hence it is particularly important to examine, how lateral deformation demands are estimated in seismic provisions. The basic procedure consists on conducting a linear elastic structural analysis, using a reduced set of forces. The displacements computed with the reduced set of forces are significantly smaller than those that can actually occur in a structure, hence there is a need to amplify them to take into account the inelastic behavior in the structure. In current seismic design provisions the amplification of displacements is done thru the use of displacement modification factor, which in the seismic provisions are commonly referred to as C_d. Hence an adequate estimation of lateral displacement demands relies on an adequate estimation of the ratio of the force modification factor, that reduces the lateral forces and the displacement modification factor that amplifies the reduced displacements.

The 1997 edition of the NEHRP Recommended Provisions for the Seismic Regulation of New Buildings and Other Structures (BSSC, 1997a) and the recently presented 2000 International Building Code edition (BSSC, 2000) have similar criteria to compute the maximum displacement demand. In the 1997 NEHRP provisions and IBC2000 the equation to estimate ultimate lateral deflections, δ_x, is given by

$$\delta_x = \frac{C_d}{I}\delta_{xe} \qquad \text{... (14.16)}$$

Where δ_{xe} is the lateral deflection determined by an elastic analysis using equivalent lateral forces (reduced forces), C_d is the deflection amplification factor, and I is the occupancy importance factor. The elastic analysis of the seismic-force-resisting system is made using the prescribed seismic base shear distributed along the height of the structure. If an equivalent static lateral force procedure is used the seismic base shear, V, in any direction is computed according to the following equation

$$V = C_s W \qquad \text{... (14.17)}$$

Where C_s is a period-dependent reduced seismic response coefficient that reflects the seismic hazard in a specific region, and W is the weight of the structure. In these documents the seismic response coefficient is defined as

$$C_s = \frac{S_{DS}}{R/I} \leq \frac{S_{D1}}{T(R/I)} \qquad \text{... (14.18)}$$

in which S_{DS} is the design spectral response acceleration in the short period range, S_{D1} is the design response acceleration at a period of 1.0 s., and R is a period-independent response modification factor, that varies from 1.5 to 8 according to the structural material and structural system. Response modification factors, R and deflection amplification factor C_d to be used for different seismic force resisting systems are shown in **Table 14.1**. It can be seen that response modification factors vary from 3.0 for moderately ductile resisting systems such as ordinary composite moment frames to 8.0 for ductile resisting systems such as special composite moment frame buildings or composite eccentrically braced systems. Deflection amplification factors vary from 2.5 for moderately ductile resisting systems such as ordinary composite moment frames to 5.5 for ductile resisting systems such as special composite moment frame buildings.

Table 14.1 Force and Deflection Modification Factors

Basic Seismic Force Resisting Systems	R	Ω_o	Cd	R/Ω_o	C_d/R
Building Frame Systems					
Composite eccentrically braced frame	8	2	4	4	0.5
Composite concentrically braced frame	5	2	4.5	2.5	0.9
Ordinary composite braced frame	3	2	3	1.5	1
Composite steel plate shear walls	6.5	2.5	5.5	2.6	0.85
Special composite RC shear walls with steel elements	6	2.5	5	2.4	0.83
Ordinary composite RC shear walls with steel elements	5	2.5	4.5	2	0.9
Moment Resisting Frame Systems					
Special composite moment frame	8	3	5.5	2.7	0.69
Intermediate composite moment frame	5	3	4.5	1.7	0.9
Composite partially restrained frame	6	3	5.5	2	0.92
Ordinary composite moment frame	3	3	2.5	1	0.83
Dual Systems with Special Moment Frames Capable of Resisting at least 25% of the Prescribed Seismic Forces					
Composite eccentrically braced frame	8	2.5	4	3.2	0.5
Composite concentrically braced frame	6	2.5	5	2.4	0.83
Composite steel plate shear walls	8	2.5	6.5	3.2	0.81
Special composite RC shear walls with steel elements	8	2.5	6.5	3.2	0.81
Ordinary composite RC shear walls with steel elements	7	2.5	6	2.8	0.86
Dual Systems with Intermediate Moment Frames Capable of Resisting at least 25% of the Prescribed Seismic Forces					
Composite concentrically braced frame	5	2.5	4.5	2	0.9
Ordinary composite braced frame	4	2.5	3	1.6	0.75
Ordinary composite reinforced concrete shear walls with steel elements	5.5	2.5	4.5	2.2	0.82

14.5 DESIGN EXAMPLE

14.5.1 THE PROBLEM

An example of 3D commercial composite frame building (G+10-storey) is taken here. The architectural plan and the structural layout of the composite frame building are shown in **Fig. 14.6(a)** and **Fig. 14.6(b)** respectively. A 3D view of the Composite frame building is also shown in **Fig. 14.7**. In this problem only slabs and beams are composite while columns are built up of steel. Brick masonry is used as outer periphery throughout the building; outer wall is 230 mm thick. Here internal walls are considered as moving partition walls and their loads are transferred to the subsequent members. Building is analyzed as bare frame and the weight of brick masonry is added in beam elements. The building is analyzed and designed for medium class soil, for various earthquake zones, using ***Equivalent Static Method of Analysis*** and ***Response Spectrum Method of Analysis.*** The same building is also analyzed and designed with concrete members. Designs are based as per the present Indian standard codal provisions. Limit state method described in IS 800:2007 is adopted for the design. Eurocodes, American codes and British codes are referred wherever the Indian code is silent. The building is modeled, analyzed and designed with the help of STAAD.Pro V8i software. Here, for the comparison of the results, best possible economical and efficient section sizes have been selected. from optimization process and trial-error methods using advantages of post processor mode of STAAD.Pro, for both concrete as well as composite structure.

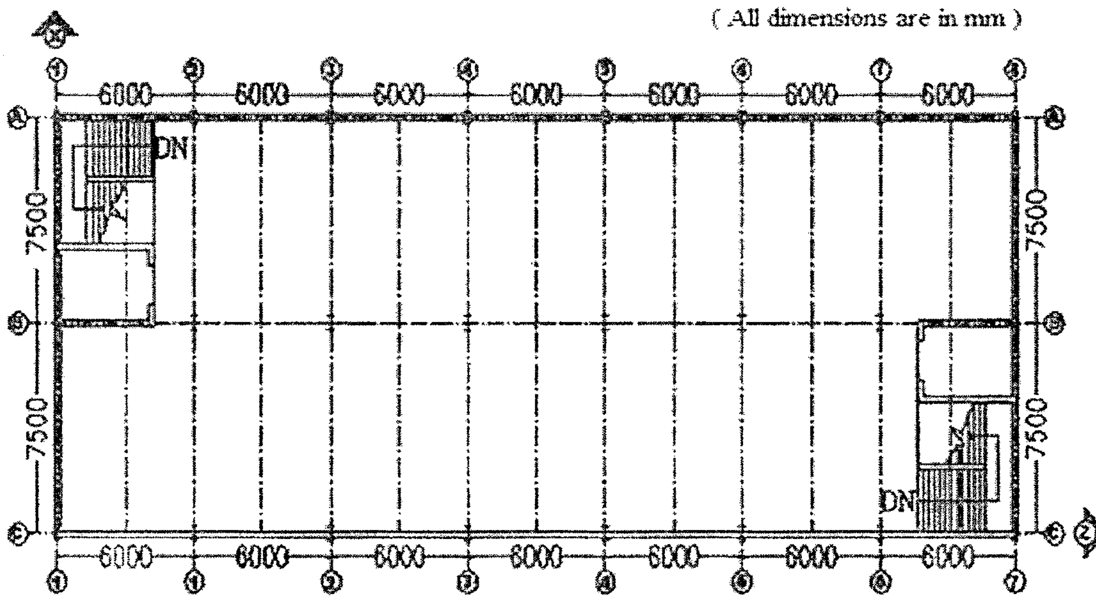


Figure 14.6(a) Typical Floor Plan of Composite Framed Building

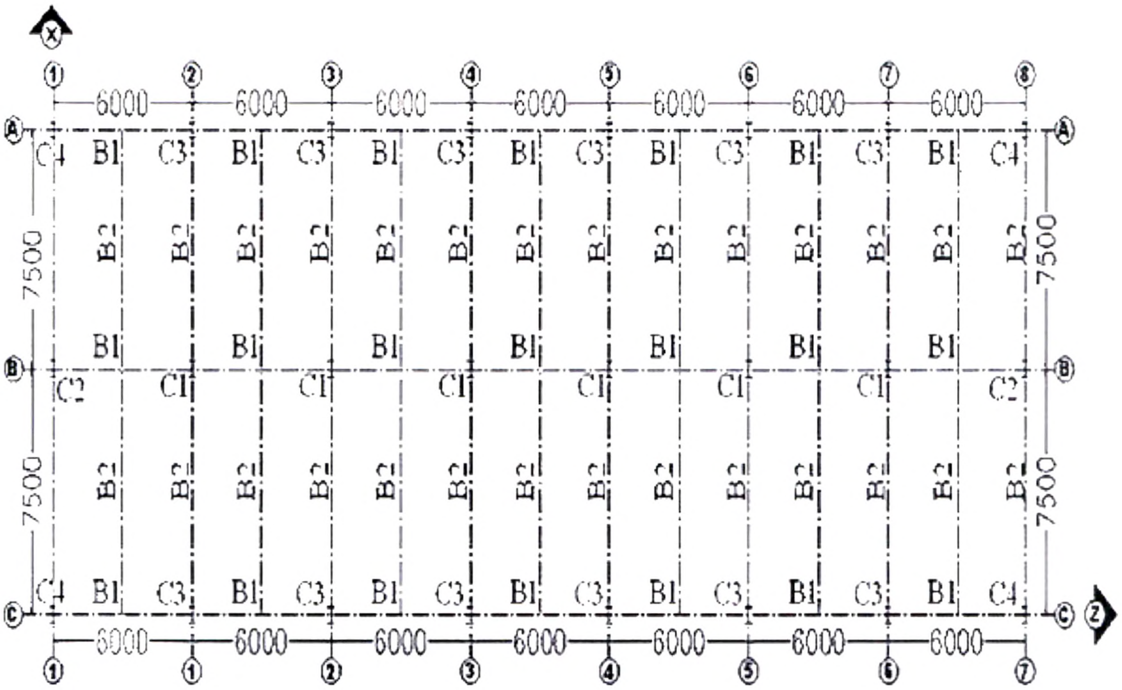


Figure 14.6(b) Structural Layout of Typical Floor for Composite Building

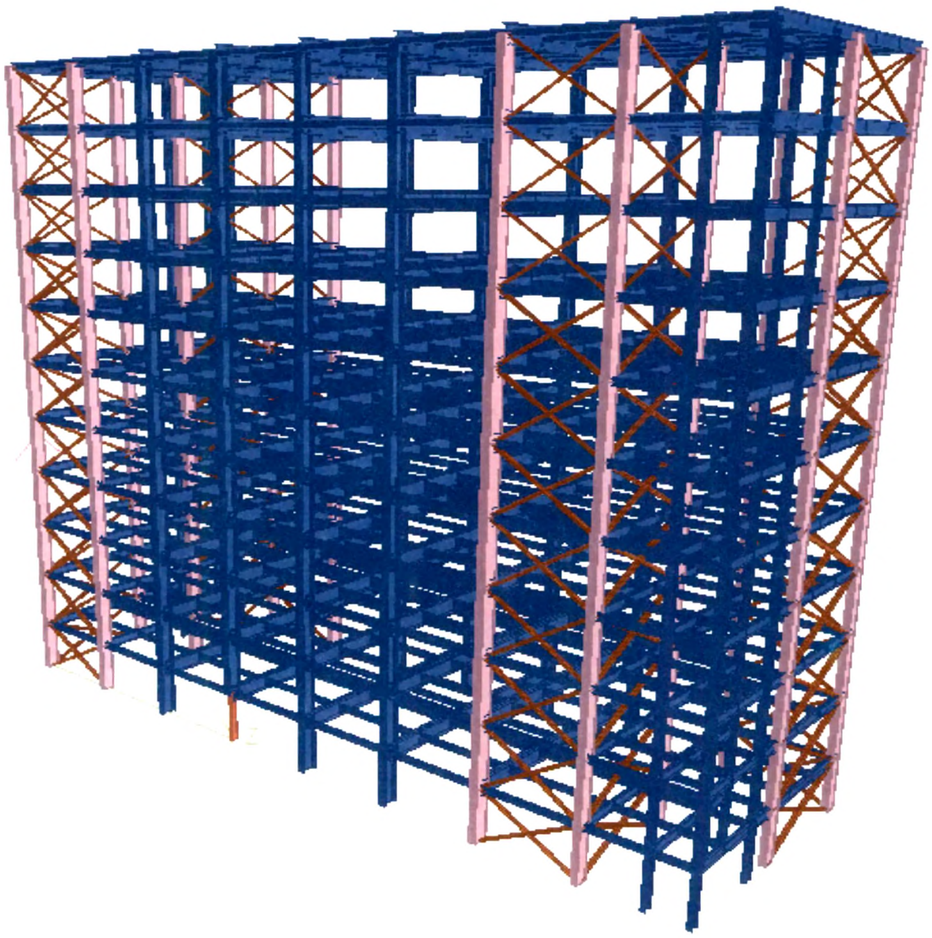


Figure 14.7 3D View of Composite Frame Building

14.5.2 PROBLEM STATISTICS

Geometrical Data

Type of building	:	Commercial
Location of building	:	Vadodara (Gujarat)
Height of building from GL	:	38.5 m
Typical storey height	:	3.5 m
Width (B) of building	:	15.00 m (in X – direction)
Depth (D) of building	:	42.00 m (in Y – direction)

Material Data

Grade of concrete	:	M 25
Yield strength of steel section	:	250 N/mm^2
Yield strength of reinforcement	:	415 N/mm^2
Unit weight of concrete	:	25 kN/m^3
Unit weight of brick masonry	:	20 kN/m^3

Loading Data

Dead Load (DL) at any typical floor level		
Moving Partitions	:	1.50 kN/m2
Floor Finishes	:	1.20 kN/m2
Weight of Metal Deck	:	0.15 kN/m2
Weight of Duct & Plastering	:	0.80 kN/m^2
Dead Load (DL) at roof level		
Weight of Metal Deck	:	0.15 kN/m2
False Ceiling; Ducts etc.	:	1.00 kN/m2
Screened concrete 50 mm thk	:	1.20 kN/m^2
Live Load (LL) at any floor level	:	4.00 kN/m^2
Earthquake Load (EL)		
Zone factor	:	As per Table 14.3
Importance factor	:	1.0
Response reduction factor	:	4.0 (Concentric braced frame) (For composite building)
	:	5.0 (SMRF) (For concrete building)

Table 14.3 Zone Factor

Zone	V	IV	III	II
Zone Factor	0.36	0.24	0.16	0.1

Load factors

For dead load : 1.50

For live load : 1.50

Material Safety factors

For structural steel : 1.15

For reinforcement : 1.15

For concrete : 1.50

Load combinations

1) 1.5(DL + LL)
2) 1.2(DL + LL + EL in X)
3) 1.2(DL + LL – EL in X)
4) 1.2(DL + LL + EL in Z)
5) 1.2(DL + LL – EL in Z)
6) 1.5(DL + EL in X)
7) 1.5(DL – EL in X)
8) 1.5(DL + EL in Z)
9) 1.5(DL – EL in Z)
10) 0.9DL + 1.5EL in X
11) 0.9DL - 1.5EL in X
12) 0.9DL - 1.5EL in Z
13) 0.9DL - 1.5EL in Z
14) 1.2DL + 0.5LL + 2.5EL in X
15) 1.2DL + 0.5LL - 2.5EL in X
16) 1.2DL + 0.5LL + 2.5EL in Z
17) 1.2DL + 0.5LL - 2.5EL in Z
18) 0.9DL + 2.5EL in X
19) 0.9DL - 2.5EL in X
20) 0.9DL + 2.5EL in Z
21) 0.9DL - 2.5EL in Z

Note: *It should be noted that, as per clause 12.2.3 of IS 800-2007 [100], load combinations from Sr. No. 14 to 21 are used for the design of bracings in case of composite structure.*

The earthquake load as per IS 1893:2002 (Part-1) is defined for earthquake in horizontal direction X (EQ-X) and Z (EQ-Z) in software and the required parameters for that are defined.

14.5.3 ASSUMPTIONS MADE

Following are the assumptions made for arrangement of building, analysis and design:

- Floor is made of reinforced cement concrete with steel deck acting as form work and bottom reinforcement, with topping for floor finish.
- All beams, columns and bracings are made of steel.
- Stair stringers are made of channel sections (ISMC) with steps and landings of chequered plates. Width of flight shall be 1000 mm. Stair well shall be open on sides with 1.0 m high steel handrails with R.C.C. roof.
- Beams of topmost decks will be sloped (say 1:100) towards external sides for drainage. It can be followed for intermediate decks also.
- Propped method of construction has been considered in this design.
- Steel concrete composite structure is designed by the limit state method using partial safety factors for loads and material strengths as specified in IS 456:2000
- All beams are simply supported in longitudinal as well as lateral directions and the concrete floor has been considered as diaphragm to transfer the horizontal loads to the braced frames on rows and axes.
- Composite beam design is made as per AISC LRFD.
- The effective width of beam is taken as span/4 for T-beams and span/8 for L-beams as per codal provisions.
- Stud shear connector is used and capacity is taken from IS 11384:1985.
- In absence of Indian code for composite slab with profiled deck, British code BS: 5950 – Part 4 is used for the design of composite slab.

14.5.4 MANUAL DESIGN OF COMPOSITE SLAB

Since there is no Indian code at present for composite slab design, British standard BS-5950-Part 4 : Structural use of Steel Work in Building – code of practice for design of composite slabs with profiled steel sheeting [93] and Eurocode 4 [7] are followed.

Effective span	:	3.00 m
Total depth of slab	:	125 mm
Live load on the floor	:	4.0 kN/m^2
Grade of concrete	:	M25
Density of concrete (Dry)	:	24 kN/m^3
Density of concrete (Wet)	:	25 kN/m

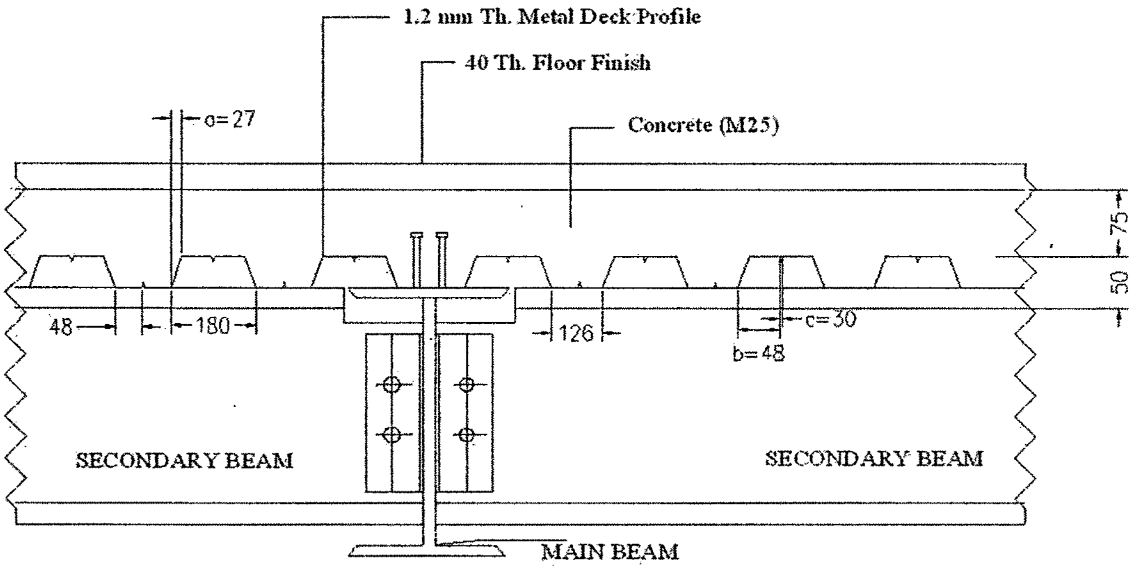


Fig. 14.20 Composite Deck Slab Details

Slab is designed as a composite slab with 125 mm deep, spanning 3.00 m using metal deck profile of thickness 1.20 mm considering Yield strength of steel, $f_{yy} = 345\ N/mm^2$ with other dimensions as:

a = 27 mm; b = 48 mm; c = 30 mm; d = 50 mm;

e = 56.82 mm; f = 48 mm; h = 75mm

Trough Spacing = 126 + 180 = 306 mm

Thickness of sheet t_p = 1.2 mm

Load Data

Dead Load

<u>At any floor level:</u>

Load from slab = 0.125 x 25 = 2.50 kN/m^2

Total dead load = 6.15 kN/m^2 (including S/W of slab, partitions, floor finish, duct, plastering &self-weight of profiled deck)

At roof level:

Total dead load = 5.00 x 1.50 = 7.50 kN/m^2 (including S/W of slab, screed concrete, false ceiling ducts etc. & self-weight of profiled deck)

Imposed load = 4.00 kN/m^2 i.e. 4.00 x 1.50 = 6.00 kN/m^2

Construction load = 1.50 kN/m^2 i.e. 1.50 x 1.50 = 2.25 kN/m^2

At construction stage

Total self weight = 2.50 + 0.15 = 2.65 kN/m^2

Ultimate design moment at construction stage = 1.4*(1.5+2.65)*32/8 = 6.54 kN-m

Assuming simply supported deck slab, the moment is due to the positive (sagging) elastic moment resistance of the section.

Check if $\frac{b}{t} \leq \frac{560}{\sqrt{f_y}}$.

$$\text{Here,} \frac{560}{\sqrt{f_y}} = \frac{560}{\sqrt{345}} = 30.15$$

$$\frac{b}{t} = \frac{48}{1.2} = 40$$

$$\frac{b}{t} > \frac{560}{\sqrt{f_y}}$$

$$b_{es} = \frac{828t}{\sqrt{f_y}} \; 1 - \frac{181t}{b\sqrt{f_y}} = 40.46 \; < b = 48.0$$

b = 48 mm in adopted

According to BS 5950: Part 4 – The stiffener area is not considered in the calculation of compressive resistance of the plate

Neutral axis position

$$x = \frac{(48 * 2 + 65.8) * 60}{(126 + 2 * 65.80 + 2 * 48)} = 27.46mm$$

Moment of inertia

$$I = 1.20*[48*2*(58.8-27.46)^2 + 126*27.46^2 + 2*65.8*\frac{60^2}{12} + 2*65.8*(29.4- 27.46)^2]$$

I = 275.13 *10^3 mm^4 / Trough

= 899.12 *10^3 mm^4 / m width

Depth of web in compression = (58.8 – 27.46) mm = 31.35 mm

$$\frac{d}{t} = \frac{31.35}{1.20} = 26.12$$. It is not necessary to check for local Bucking of Web

Elastic section modulus $Z_c = 899.12 * 10^3/(59.4 - 27.46) = 28.15 * 103\ mm3/m$

Elastic moment resistance of section (Sagging moment)

$$M_e = (Z_e * 0.93 * f_y) = \{(28.15*10^3)*0.93*(345*10^{-6})\} = 9.03 \text{ kN-m} > M_u$$

After Construction

Bending moment on deck after construction $= \left(2.65 * \frac{3^2}{8}\right) = 2.98$ kNm.

Equivalent stress in compression plate, $\sigma = 2.98/9.711 * 345 = 100.93$ N/mm^2

Using this stress,

$$\frac{b}{t} = \frac{48}{1.2} = 40$$

$$\frac{560}{\sqrt{f_y}} = 56.74 \ (Here, f_y = \delta\ )$$

$$\frac{b}{t} > \frac{560}{\sqrt{f_y}}$$

b = 48 mm is adopted

$$b_{es} = \frac{828t}{\sqrt{f_y}}\left|1 - \frac{181t}{b\sqrt{f_y}}\right| = 53.46 > b$$

b = 48 mm is adopted

X = 27.46 mm

I = 899.12*10^3 mm^4/m width

Deflection of soffit of simply supported slab

$$\delta = \frac{5}{384} * \frac{2.65 * (3000)^4}{205 * 10^3 * 899.12 * 10^3} = 14.5\ mm < \left(\frac{span}{180}\right) = 16.67\ mm$$

Since, the deflection is less than (span/180), the effects of ponding of concrete at greater deflections are not being considered.

Composite Condition

Moment resistance of the composite section as a reinforced concrete slab, (using full shear connection),

Area of deck, A_p = (126+65.8)*2*1 = 460.32 mm^2 / Trough

= 1504.31 mm^2 / m

Tensile resistance = (0.93*348*1504.31*10^{-3}) = 482.66 kN

Neutral axis depth into concrete,

$$x_c = \frac{482.66 * 10^3}{0.4 * f_{ck} * 10^3} = \frac{482.66 * 10^3}{0.4 * 25 * 10^3} = 48.27\ mm$$

Plastic moment resistance of composite section,

M_p = 482.66 * (125-48-48.27/2)*10^{-3} = 25.52 kN m

Applied moment in composite condition,

M_u = [1.6(4.0) +1.4(6.15)*$(3.0)^2$/8] = 16.89 kN m

Check for shear-bond capacity

$$V_u = \frac{B_s d_s}{1.25 L_v}\left[\frac{m_r A_P}{B_s L_v} + k_r\sqrt{f_{ck}}\right]$$

Where, = $B_s d_d$ = A_c= (65+60/2)*1000 = 95000 mm^2, d_e= 95 mm, A_p = 1504.31 mm^2 and L_v = 3000/4 = 750 mm.

Typical values of empirical constants are

M_r = 130, k_r = 0.004 and V_u = 28.45 kN

Applied ultimate shear force,

$$V = \frac{4M_u}{L} = \frac{4 * 16.89}{3.0} = 22.52 < v_u$$

Shear stress on concrete $V_c = 22.52 * 10^3/95 * 10^3$= 0.237 N/mm^2.

This value is nominal as per BS 8110. Hence, the slab section is safe under shear stress on concrete.

Check for deflection of composite slab

Properties of cracked section:

Neutral axis depth below upper surface on slab

$$x_e = d_e\left[\sqrt{(\alpha_e p)^2 + 2\alpha_e p} - \alpha_e p\right]$$

α_e =Modular Ration = 13

$$P = \frac{A_p}{A_c} = \frac{1504.31}{95000} = 0.01584$$

$\alpha_e p$ =(0.01584*13) = 0.206

X_e= 95[$\sqrt{(0.206)^2 + 2 * (0.206) - 0.206}$] = 23.60 mm

Moment of inertia of cracked section

$$Ic = \frac{(23.6)^3}{3 * 13} + 0.01584 * 95 * (95 - 23.6)^2 + 899.12$$

= 8907.55 mm^4/mm width (in steel units)

Service load on composite slab

W_s = 4.0 kN/m^2

Deflection under service load

$$\alpha = \frac{5}{384} * \frac{4.0 * (3000)^2 * 10^{-3}}{205 * 10^3 * 8907.5} = 2.311\, mm < \left(\frac{span}{615}\right) = 4.88\, mm$$

Hence, OK.

Mesh reinforcement provided = 0.1% of cross-sectional area of concrete

$$= 0.1\,\%\text{ of Ac } = \left(\frac{0.1}{100} * 95000\right) mm^2 = 95 mm^2.$$

This reinforcement is provided for crack control and fire resistance requirements. All primary and secondary beams are designed as composite beam sections, as per BS 5950: Part 3 [93]; modified to suit IS code provisions.

As per the above provisions,

$$\alpha_e = \text{ Effective modular ratio and it is given by} = \alpha_s + \frac{p_1}{\alpha_1 - \alpha_s}$$

Where,

α_s= Short term modular ratio = 6 for normal weight concrete, α_s = Long term modular ratio = 18 for normal weight concrete and p_1= Proportion of total loading which is ling term.

In this case,

$$p_1 = \frac{Long\ tem\ load}{Total\ load} = 1$$

$$\alpha_e = 6 + 1.0\,(18 - 6) = 18\,.$$

14.6 STRUCTURAL MODELLING IN STAAD.PRO ENVIRONMENT

STAAD.Pro V8i is the most popular structural engineering software product for 3D model generation, analysis and multi-material design. It has an intuitive, user-friendly GUI, visualization tools, powerful analysis and design facilities and seamless integration to several other modeling and design software products.

For static or dynamic analysis of bridges, containment structures, embedded structures (tunnels and culverts), pipe racks, steel, concrete, aluminum or timber buildings, transmission towers, stadiums or any other simple or complex structure, STAAD.Pro has been the choice of design professionals around the world for their specific analysis needs. In the present work, a Basement + Ground + 9 storeys, commercial building has been modeled, analyzed and designed with the graphical environment of STAAD.Pro V8i, version 20.07.07.23.

14.6.1 Steps for Modelling a Composite Structure in STAAD.Pro

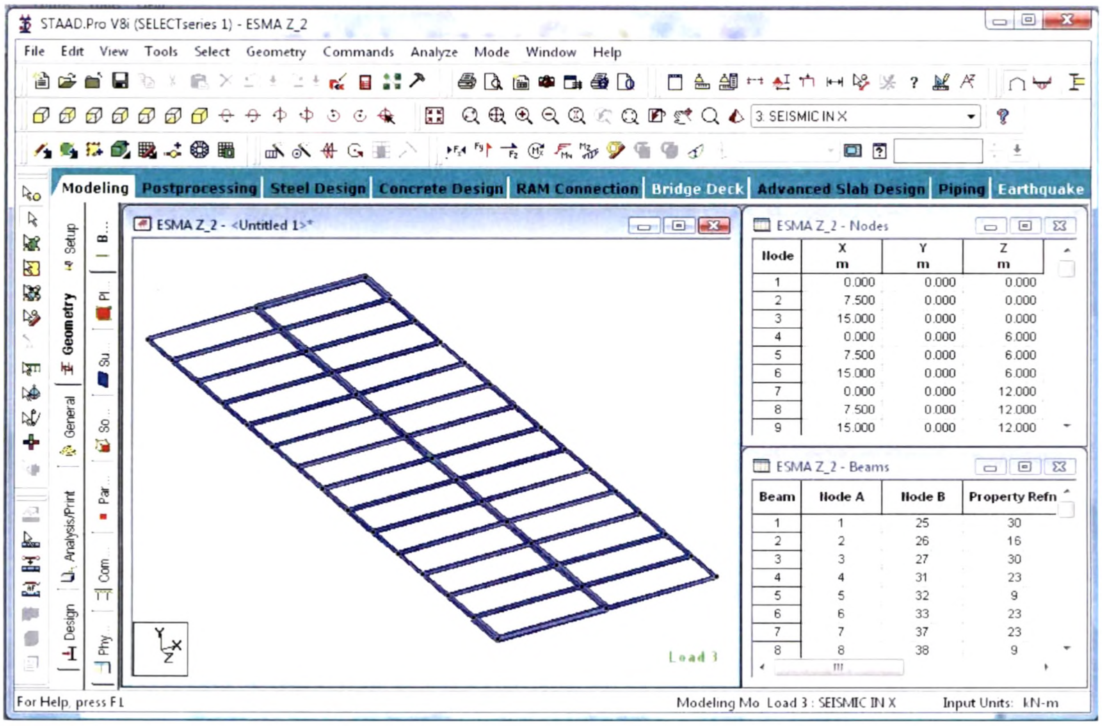


Fig. 14.21 Model Generation in STAAD.Pro Software

- Create beams for one floor with the help of Geometry option as shown in **Fig. 14.21**.
- Translate created floor in Y-direction as shown in **Fig. 14.22**.

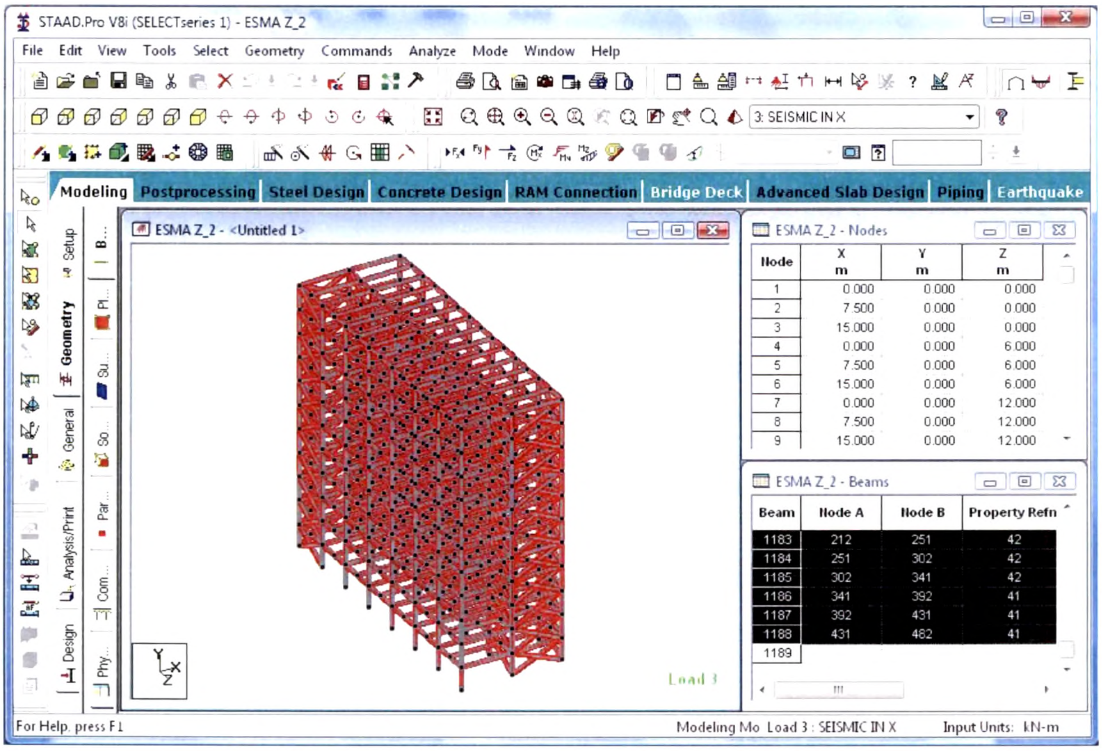


Fig. 14.22 Translation of Created Floor

➢ Define deck with the help of composite deck command as depicted in **Fig. 14.23**.

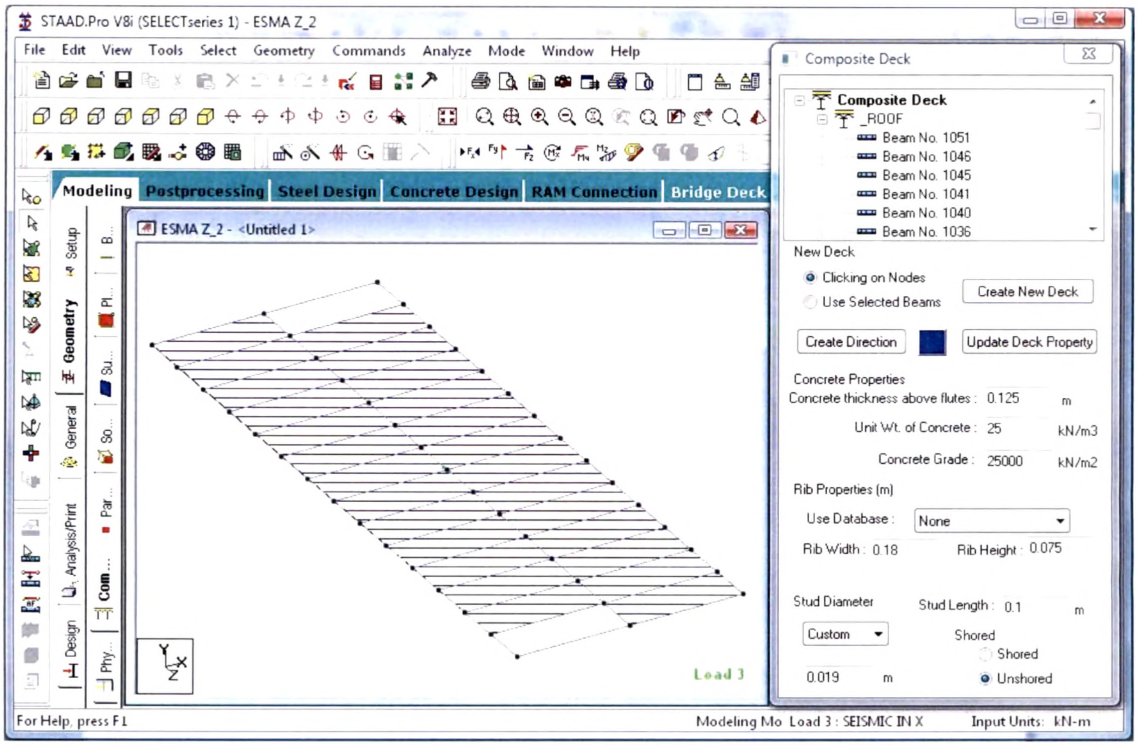


Fig. 14.23 Composite Deck Definition

➢ Select section from the section data base and attach to the structural elements with the help of *Property* tab as shown in the screen shot given in **Fig. 14.24**.

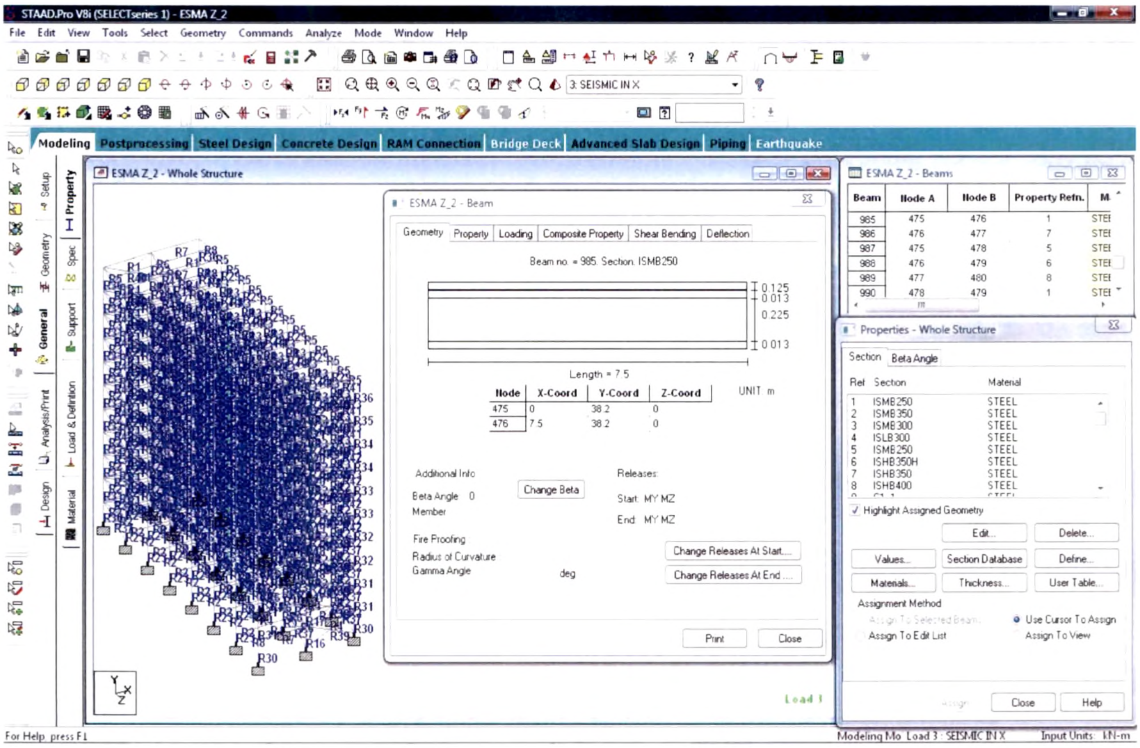


Fig. 14.24 Attach Support Condition, Material and Sectional Properties

In STAAD.Pro, one can define the sectional properties from available database or one can define his own desired sectional properties with the help of *Create User Table* command **(Fig. 14.25)**. These created user table with simple GUI can be loaded and attached to the elements from *Property tab.*

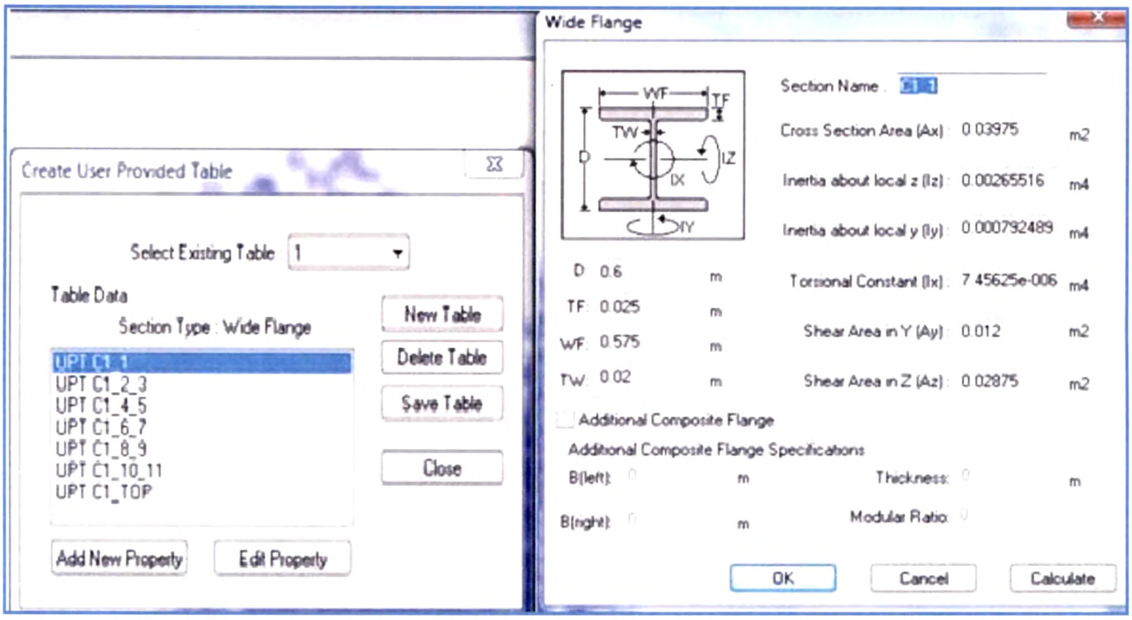


Fig. 14.25 User Table Definition

Following 9 types of sections can be defined and attached to the geometry using Create User Table command.

1. Wide Flange sections	4. Double Angle sections	7. Tube sections
2. Channel Sections	5. Tee Sections	8. I sections
3. Angle Sections	6. Pipe Sections	9. Prismatic sections

- ➢ Define end conditions for beams using *Release* command from specification menu.
- ➢ Define bracings as Truss/Tension members using specification menu.
- ➢ Define *Master Slave* command from specification menu as composite deck will act as rigid diaphragm.
- ➢ Create basic loads as DL, LL, and EQ loads and make different load combinations **(Fig. 14.26)**.
- ➢ Define seismic load in load definition with the help of Load & Definition menu.
- ➢ Under seismic load, define command for seismic weight calculation of the structure.
- ➢ Create basic load cases for dead load, live load and earthquake under Load cases details.
- ➢ For response spectrum analysis define Response spectrum load in basic Load cases.
- ➢ Also define load combinations under Load cases details.

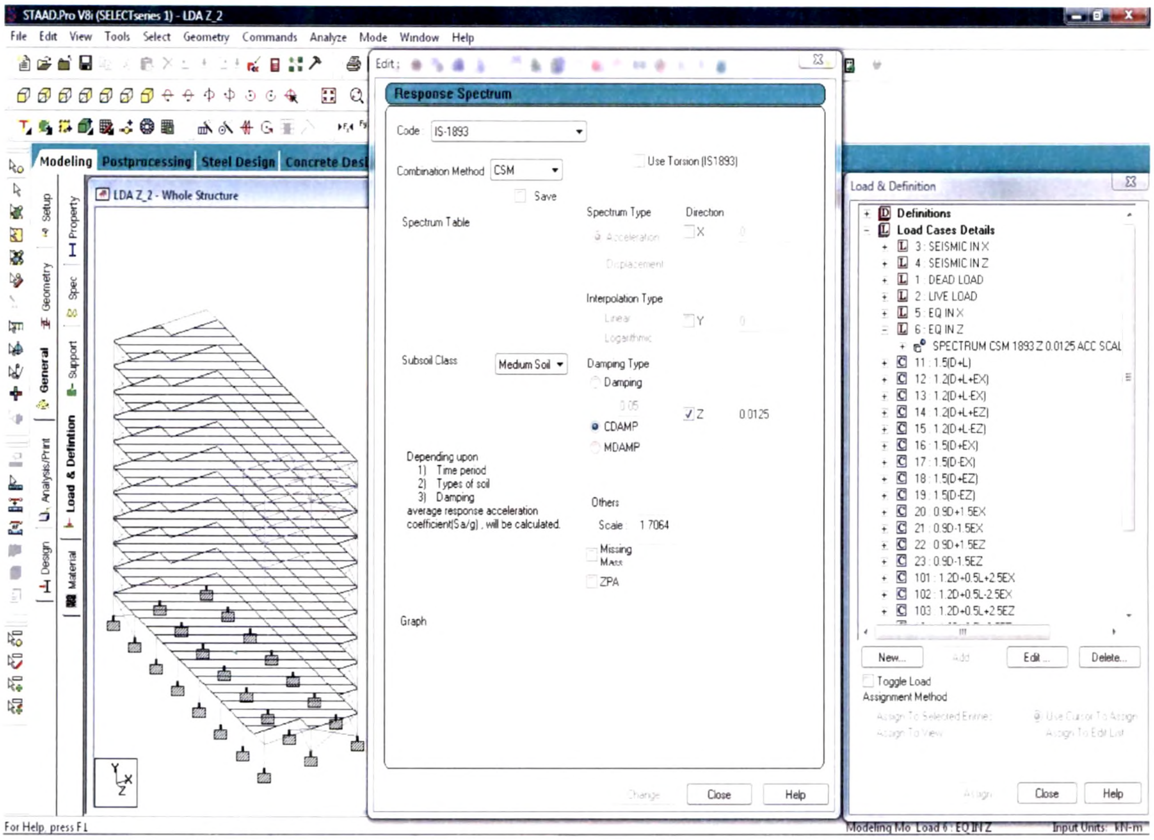


Fig. 14.26 Response Spectrum Load Definition

14.6.2 Analysis and Design Using STAAD.Pro

After following all the above steps, select *Analysis/Print* command as shown in **Fig. 14.27.**

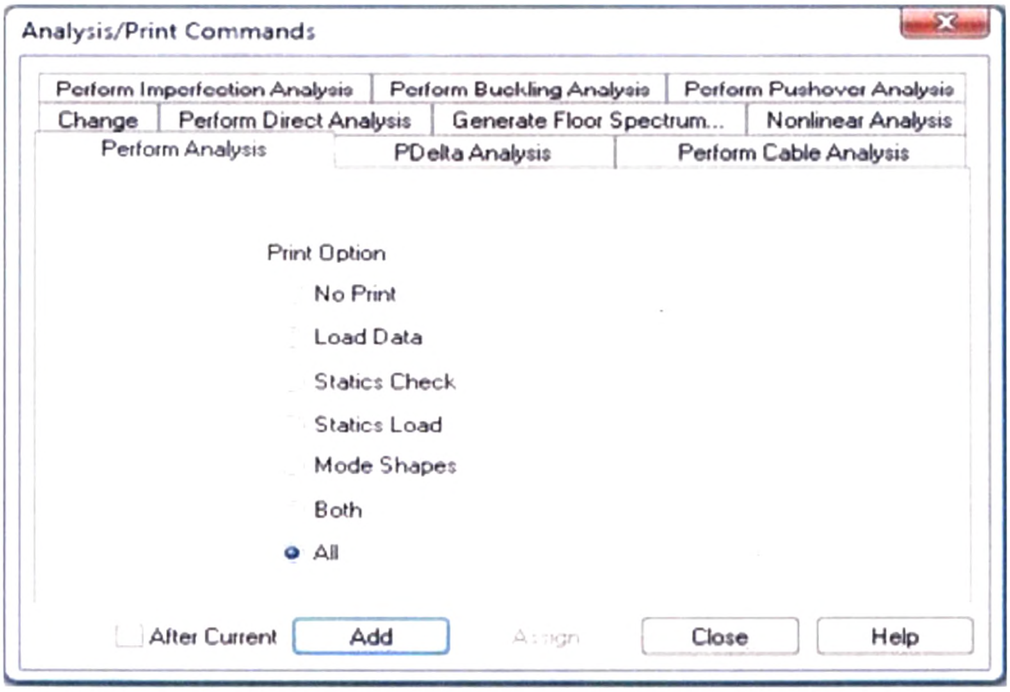


Fig. 14.27 Analysis/Print Command

With defining Analysis/print command, software will analyze the structure according to defined type of analysis **(Fig. 14.28)**. Print option will give results in output file in accordance with selected option.

After defining Analysis/print command, run analysis and check output file for warnings and errors if any. If any discrepancy is found, one has to apply corrections and run the analysis again until all the errors are removed. After removing all the errors, define design parameters and design commands.

For designing this building here I used design parameter given in IS 1893:2002 code are used **(Fig. 14.29)** and various load combination should be given in **Fig. 14.30.**

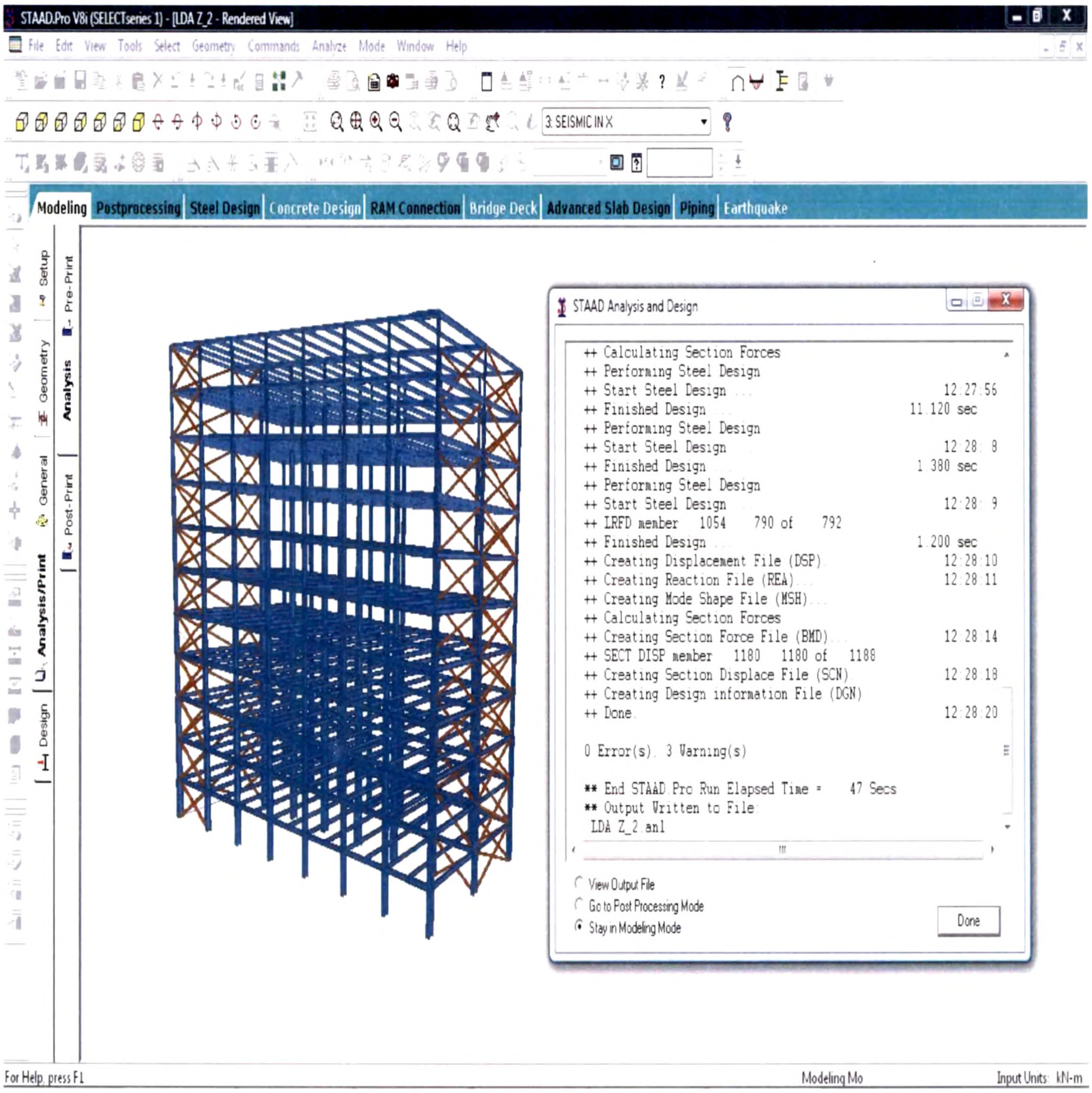


Fig. 14.28 Analysis of Structure in STAAD.Pro

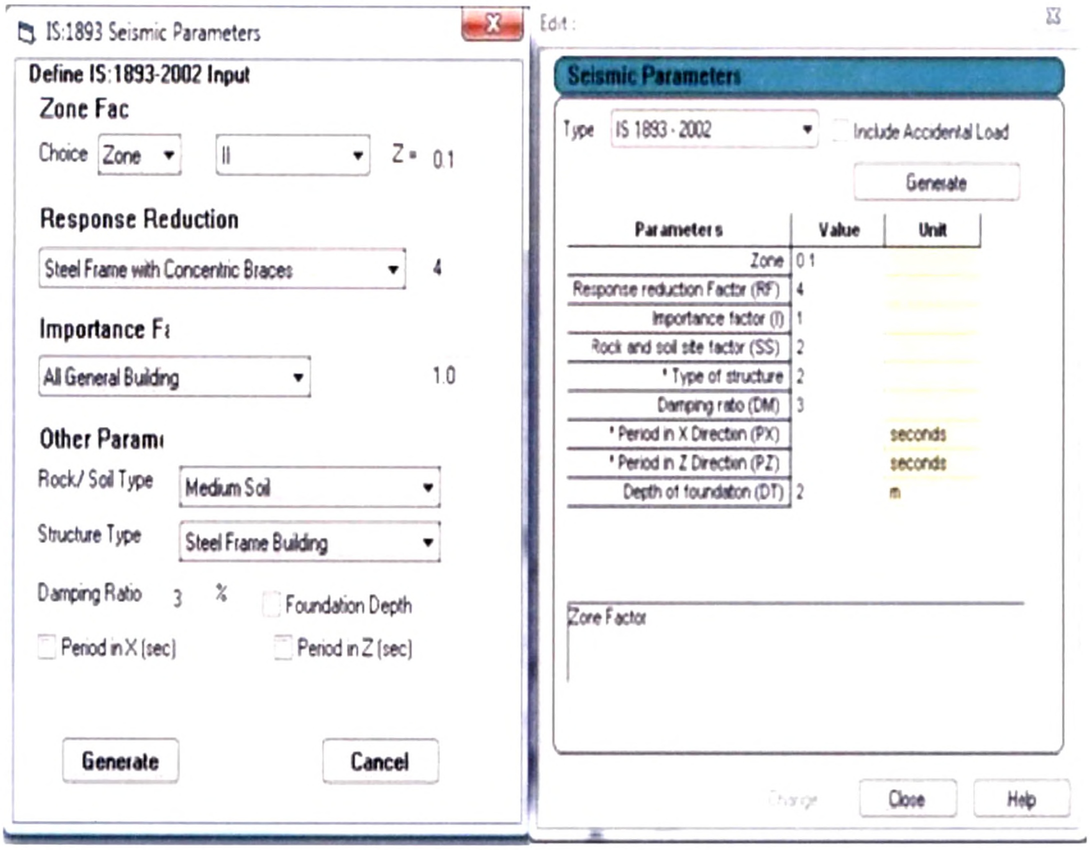


Fig. 14.29 Seismic Parameters Definition

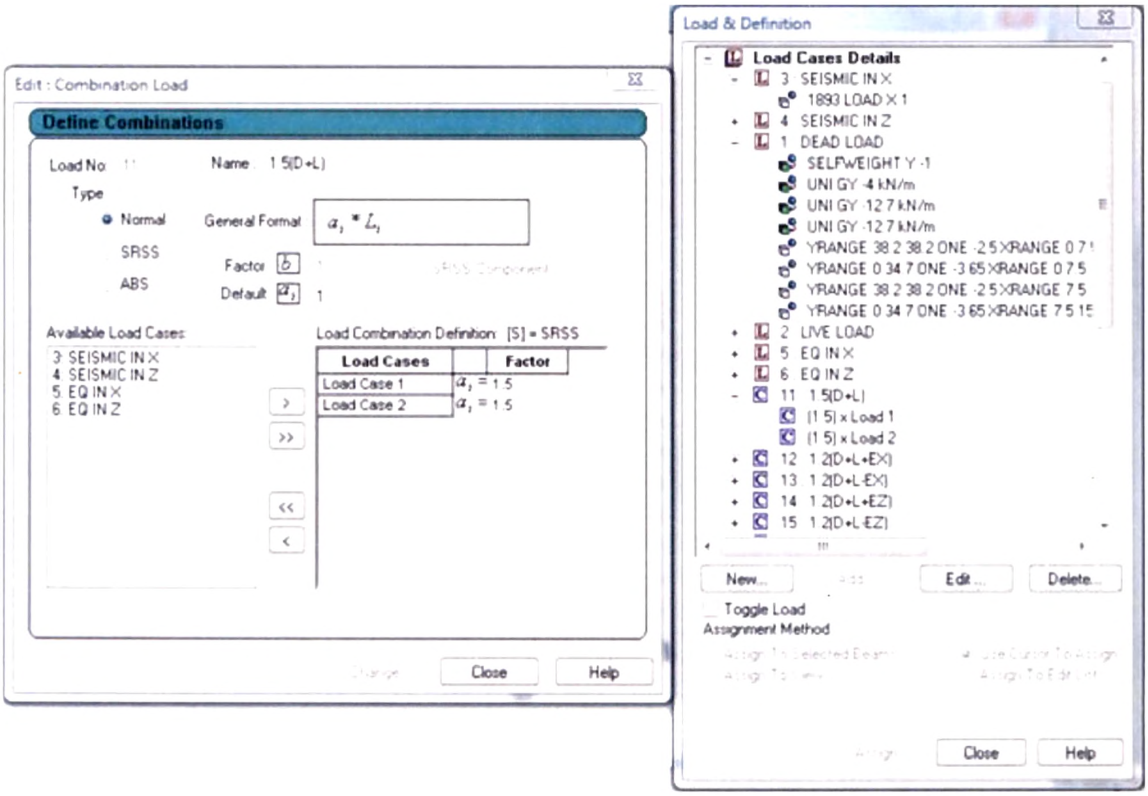


Fig. 14.30 Definition of Load Cases and Load Combinations

Here, a G+10 storied commercial building is designed as steel-concrete composite structure as well as concrete structure to facilitate a comparison. In case of composite structure, beams are designed as per AISC LRFD 3rd edition and columns are designed as per the specifications of IS 800: 2007 [100]. While in case of concrete structure, all beams and columns have been designed as per IS 456: 2000 and IS 13920: 1993 [119]. The parameters defined for the above standards are as follows:

14.6.3 COMPOSITE BEAM DESIGN AS PER AISC-LRFD

The design of composite beams as per the 3rd edition of the American LRFD code has been implemented.

Table 14.4 Composite Beam Design Parameters for AISC-LRFD

Name	Default value	Description
RBH	0.0 inches	Rib Height
EFFW	Value used in analysis	Effective width of slab
FPC	Value used in analysis	Ultimate compressive strength of concrete

14.6.4 STEEL DESIGN AS PER IS 800:2007

The design process follows the following design checks.

Slenderness

The slenderness ratio (KL/r) of compression members shall not exceed 180, and the slenderness ratio (L/r) of tension members shall not exceed 400. The user can edit the default values through MAIN and TMAIN parameters, as defined in the Design Parameter list.

Section Classification

The IS 800: 2007 specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as Plastic, Compact, Semi-Compact or Slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section as well as nature of the load the member is withstanding. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and design the section for the critical load case accordingly.

Tension

The criteria governing the capacity of Tension members are based on:

- Design Strength due to Yielding in Gross Section.
- Design Strength due to Rupture of Critical Section.
- Design Strength due to Block Shear.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per Section 6.2 of the code. The Design strength, involving rupture at the section with the net effective area, is evaluated as per section 6.3 of the code. Here, number of bolts in the connection may be specified by the user through the use of the design parameter ALPHA. The Design strength, involving block shear at an end connection, is evaluated as per section 6.4 of the code. This criteria is made optional by the parameter DBS. If the value of DBS is specified as 1, additional design parameters AVG, AVN, ATG and ATN must be supplied to the program for that member. STAAD calculates the tension capacity of a given member based on these three limit states.The Net Section Area may be specified by the user through the use of the parameter NSF.

Compression

The design capacity of the section against Compressive Force, the guiding phenomenon is the flexural buckling. The buckling strength of the member is affected by residual stress, initial bow and accidental eccentricities of load.

To account for all these factors, the strength of the members subjected to axial compression is defined by buckling class a, b, c or d as per Section 7.1.2.2 and Table 7 of the code. Imperfection factor, obtained from buckling class, and Euler's Buckling Stress ultimately govern compressive force capacity of the section as per Section 7.1.2.

Shear

The design capacities of the section against Shear Force in major- and minor-axis directions are evaluated as per Section 8.4 of the code, taking care of the following phenomena:

- Nominal Plastic Shear Resistance
- Resistance to Shear Buckling

Shear area of the sections are calculated as per Sec. 8.4.1.1. Nominal plastic shear resistance is calculated as per Sec. 8.4.1.Among shear buckling design methods, Simple post-critical method is adopted as per sec. 8.4.2.2(a).

Bending

The design bending moment capacity of a section is primarily dependent on whether the member is laterally supported or unsupported.

If the member is laterally supported, then the design strength is calculated as per the provisions of the Section 8.2.1 of the code, based on the following factors:

- Whether section with webs susceptible to shear buckling before yielding.
- Shear Force to Design Shear Strength Ratio.
- Section Classification.

If the member is laterally unsupported, then the design strength is calculated as per the provisions of the Section 8.2.2 of the code, based on the following factors:

- Lateral Torsional Buckling.
- Section Classification.

Combined Interaction Check

Members subjected to various forces – axial, shear, moment, torsion - are checked against combined interaction check. This interaction check is done taking care of two aspects -

- Section Strength and
- Overall Member Strength

Section Strength interaction ratio is calculated as per Sec. 9.3.1 of the code. Overall Member Strength interaction ratio is calculated as per Sec. 9.3.2, taking care of the design parameters PSI, CMX, CMY and CMZ.

14.6.5 STAAD.PRO DESIGN PARAMETERS AS PER IS 800: 2007

The program contains a large number of parameter names which are required to perform design and code checks. These parameter names, with their default values, are already listed in the **Table 13.7** of the previous chapter.

14.6.6 STAAD.PRO CONCRETE DESIGN PARAMETERS AS PER IS 13920: 1993

Earthquake motion often induces force large enough to cause inelastic deformations in the structure. If the structure is brittle, sudden failure could occur. But if the structure is made to behave ductile, it will be able to sustain the earthquake effects better with some deflection larger than the yield deflection by absorption of energy. Therefore ductility is also required as an essential element for safety from sudden collapse during severe shocks. STAAD has the capabilities of performing concrete design as per IS: 13920. While designing it satisfies all provisions of IS: 456 - 2000 and IS: 13920 for beams and columns.

The program contains a number of parameters that are needed to perform design as per IS 13920. It accepts all parameters that are needed to perform design as per IS: 456. Over and above it has some other parameters that are required only when designed is performed as per IS: 13920. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. **Table 14.5** contains a complete list of the available parameters and their default values. These design parameter presented in **Fig. 14.30.** It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

Table 14.5 IS 13920:1993 Concrete Design Parameters

Parameter Name	Default Value	Description
FYMAIN	415 N/mm^2	Yield stress for main reinforcing steel.
FYSEC	415 N/mm^2	Yield stress for secondary reinforcing steel.
FC	30 N/mm^2	Concrete yield stress.
CLEAR	25 mm 40 mm	For beam members. For column members.
MINMAIN	10 mm	Minimum main reinforcement bar size.
MAXMAIN	60 mm	Maximum main reinforcement bar size.
MINSEC	8 mm	Minimum secondary reinforcement bar size.
BRACING	0.0	BEAM DESIGN A value of 1.0 means the effect of axial force will be taken into account for beam design. COLUMN DESIGN 1.0 means the column is unbraced about major axis. 2.0 means the column is unbraced about minor axis. 3.0 means the column is unbraced about both axis.
RATIO	4.0	Maximum % of longitudinal reinforcement in columns.
RFACE	4.0	A value of 4.0 means longitudinal reinforcement in column is arranged equally along 4 faces. A value of 2.0 for 2 faced distributions about major axis.

		A value of 3.0 for 2 faced distributions about minor axis.
ELZ	1.0	Ratio of effective length to actual length of column about major axis.
ELY	1.0	Ratio of effective length/actual length of column about minor axis.
REINF	0.0	Tied column. A value of 1.0 will mean spiral reinforcement.
TRACK	0.0	BEAM DESIGN: For TRACK = 0.0, output consists of reinforcement details at START, MIDDLE and END. For TRACK = 1.0, critical moments are printed in addition to TRACK 0.0 output. For TRACK = 2.0, required steel for intermediate sections defined by NSECTION are printed in addition to TRACK 1.0 output. COLUMN DESIGN: With TRACK = 0.0, reinforcement details are printed. With TRACK = 1.0, column interaction analysis results are printed in addition to TRACK 0.0 output. With TRACK = 2.0, a schematic interaction diagram and intermediate interaction values are printed in addition to TRACK 1.0 output.
SPSMAIN	25 mm	Minimum clear distance between main reinforcing bars in beam and column. For column centre to centre distance between main bars cannot exceed 300 mm.
SFACE	0.0	Face of support location at start of beam. It is used to check against shear at the face of the support in beam design. The parameter can also be used to check against shear at any point from the start of the member.
EFACE	0.0	Face of support location at end of beam. The parameter can also be used to check against shear at any point from the end of the member.
ENSH	0.0	Perform shear check against enhanced shear strength as per Cl. 40.5 of IS: 456-2000. ENSH = 1.0 means ordinary shear check to be performed (no enhancement of shear strength at sections close to support) For ENSH = a positive value (say x), shear strength will be enhanced up to a distance x from the start of the member. This is used only when a span of a beam is subdivided into two or more parts. For ENSH = a negative value (say −y), shear strength will

		be enhanced up to a distance y from the end of the member. If default value (0.0) is used the program will calculate Length to Overall Depth ratio. If this ratio is greater than 2.5, shear strength will be enhanced at sections (< 2d) close to support otherwise ordinary shear check will be performed.
RENSH	0.0	Distance of the start or end point of the member from its nearest support. This parameter is used only when a span of a beam is subdivided into two or more parts.
EUDL	None	Equivalent U.D.L on span of the beam. This load value must be the unfactored load on span. During design the load value is multiplied by a factor 1.2. If no UDL is defined, factored shear force due to gravity load on span will be taken as zero. No elastic or plastic moment will be calculated. Shear design will be performed based on analysis result.
GLD	None	Gravity load number to be considered for calculating equivalent UDL on span of the beam, in case no EUDL is mentioned in the input.
IPLM	0.0	Default value calculates elastic/plastic hogging and sagging moments of resistance of beam at its ends. A value of 1.0 means calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at start node of beam. This implies no support exists at start node. A value of -1.0 means calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at start node of beam. . This implies support exists at start node. A value of 2.0 means calculation of elastic/plastic hogging and sagging moments of resistance of beam to be ignored at end node of beam. This implies no support exists at end node. A value of -2.0 means calculation of elastic/plastic hogging and sagging moments of resistance of beam to be considered at end node of beam.This implies support exists at end node.
COMBINE	0.0	Default value means there will be no member combination. A value of 1.0 means there will be no printout of sectional force and critical load for combined member in the output. A value of 2.0 means there will be printout of sectional force for combined member in the output. A value of 3.0 means there will be printout of both sectional force and critical load for combined member in the output.

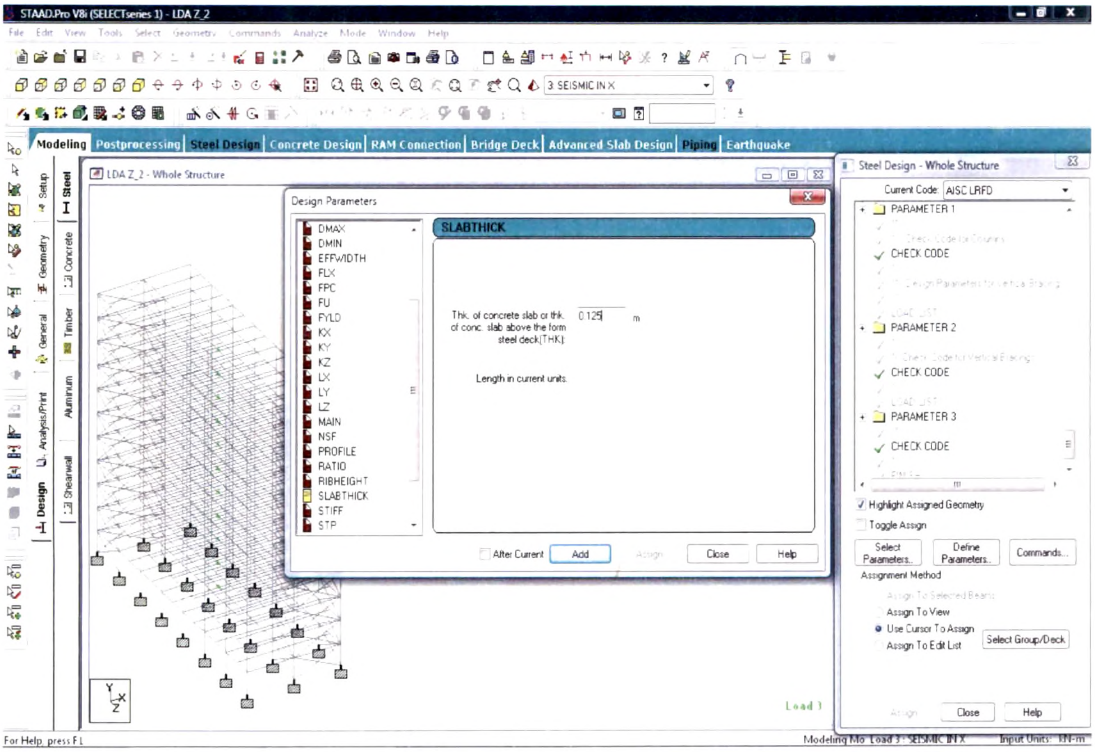


Fig. 14.30 Definition of Design Parameters

14.7 COMPARISON OF RESULTS

Here, the building has been designed for various earthquake zones using Equivalent Static Method of Analysis (ESMA) and Linear Dynamic Analysis (LDA) i.e. Response Spectrum Method of Analysis. Results obtained using STAAD.PRO software are presented in **Tables 14.6 to 14.36**. While designing the structure, following parameters are compared:

1. Nodal displacements, support reactions, support moments, beam end forces and beam end moments for two methods of earthquake analysis in case of composite structure. Also above values have been compared for various earthquake zones.
2. Values obtained for composite structure are also compared with those obtained for concrete structure in case of earthquake zone III.
3. Base shear, time period and frequency are compared in case of Response Spectrum Method of Analysis for various earthquake zones.
4. Three basic mode shapes for various earthquake zones are compared in case of composite structure.
5. In case of composite structure, in different earthquake zones, weight of the structure is compared for Response Spectrum Method of Analysis and ESMA.

6. Base shear, time period and frequency are compared in case of Response Spectrum Method of Analysis for composite and concrete structures in earthquake zone III.

Table 14.6 Nodal Displacement in EQ Zone II for Composite Structure

	ESMA			LDA		
	X mm	Y mm	Z mm	X mm	Y mm	Z mm
Max X	**58.78**	-10.59	3.53	**50.75**	-10.29	10.80
Min X	**-60.26**	-23.89	-0.11	**-50.76**	-24.66	-8.31
Max Y	0.40	**2.02**	23.83	28.49	**4.16**	5.63
Min Y	-1.51	**-45.74**	1.88	0.70	**-46.24**	0.87
Max Z	-7.80	-16.03	**53.79**	4.13	-13.37	**45.86**
Min Z	-11.75	-11.48	**-53.60**	-11.15	-11.49	**-45.87**

Table 14.7 Nodal Displacement in EQ Zone III for Composite Structure

	ESMA			LDA		
	X mm	Y mm	Z mm	X mm	Y mm	Z mm
Max X	**81.81**	-9.74	2.92	**57.08**	-8.04	5.00
Min X	**-83.95**	-25.39	-0.47	**-57.09**	-22.34	-3.22
Max Y	0.11	**3.05**	37.30	36.39	**4.38**	2.74
Min Y	-1.06	**-45.47**	1.31	0.00	**-44.20**	0.00
Max Z	-6.92	-13.96	**75.98**	1.25	-9.41	**54.91**
Min Z	-11.75	-11.03	**-75.83**	-6.26	-9.22	**-54.92**

Table 14.8 Nodal Displacement in EQ Zone IV for Composite Structure

	ESMA			LDA		
	X mm	Y mm	Z mm	X mm	Y mm	Z mm
Max X	**97.14**	-8.27	2.55	**74.14**	-7.07	4.54
Min X	**-99.70**	-26.39	-0.21	**-74.14**	-24.52	-3.06
Max Y	-0.06	**4.16**	53.23	2.30	**6.52**	52.37
Min Y	0.00	**-45.17**	-1.33	0.00	**-44.70**	0.00
Max Z	-5.94	-11.20	**102.51**	1.37	-7.25	**79.30**
Min Z	-11.56	-10.20	**-102.30**	-5.55	-8.89	**-79.31**

It is clear, from **Tables 14.6 - 14.8**, that the values obtained for nodal displacements by Linear Dynamic Analysis are less compared to those obtained by Equivalent Static Method of Analysis (ESMA), for all the three earthquake zones.

Table 14.9 Support Reactions in EQ Zone II for Composite Structure

	ESMA			LDA		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**124.32**	1277.99	0.01	**449.64**	5192.98	208.77
Min Fx	**-117.37**	1304.15	0.01	**-449.63**	1517.77	-208.75
Max Fy	0.22	**8420.58**	0.00	0.08	**8423.48**	0.00
Min Fy	14.04	**-589.64**	-89.69	-0.02	**-608.85**	-52.25
Max Fz	0.65	2184.30	**123.58**	199.78	4622.09	**337.32**
Min Fz	-1.04	2183.45	**-123.64**	-199.76	2088.73	**-337.29**

Table 14.10 Support Reactions in EQ Zone III for Composite Structure

	ESMA			LDA		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**209.69**	808.76	-0.002	**517.14**	5843.48	202.19
Min Fx	**-198.8**	850.53	0.055	**-517.13**	1285.22	-202.18
Max Fy	0.08	**8421.10**	0.00	0.00	**8423.32**	0.00
Min Fy	20.59	**-921.47**	-134.30	-0.09	**-964.63**	-84.33
Max Fz	0.08	1686.46	**191.79**	8.71	6832.42	**455.79**
Min Fz	-0.69	1684.19	**-191.95**	-8.71	2459.55	**-455.79**

Table 14.11 Support Reactions in EQ Zone IV for Composite Structure

	ESMA			LDA		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**298.81**	253.62	0.05	**808.00**	7413.04	304.34
Min Fx	**-286.14**	-89.69	0.10	**-808.00**	366.34	-304.32
Max Fy	-0.24	**8425.10**	0.00	0.14	**8420.28**	0.00
Min Fy	-0.23	**-1364.58**	-201.90	-0.23	**-1457.03**	-135.33
Max Fz	-0.25	986.63	**289.00**	1.52	7318.92	**684.94**
Min Fz	-0.74	983.22	**-289.19**	-1.53	885.019	**-684.95**

From **Tables 14.9 - 14.11**, it can be seen that the support forces reactions in two orthogonal directions are higher for LDA compared to ESMA. This is due to the reason that in LDA effect of infill walls are not considered in analysis. It can also be seen from the tables that the force in Y direction which is the force governing for the foundation design are very close to each other in both the cases, i.e. ESMA and LDA.

Table 14.12 Support Moments in EQ Zone II for Composite Structure

	ESMA			LDA		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**98.15**	0.00	-13.62	**83.82**	0.00	40.15
Min Mx	**-98.15**	0.00	-9.24	**-83.82**	0.00	-43.86
Max My	-0.53	**0.00**	180.12	0.00	**0.00**	2.60
Min My	-90.32	**0.00**	-11.64	-16.68	**0.00**	-272.49
Max Mz	-0.57	0.00	**208.00**	16.68	0.00	**272.49**
Min Mz	0.461	0.00	**-210.89**	-16.68	0.00	**-272.49**

Table 14.13 Support Moments in EQ Zone III for Composite Structure

	ESMA			LDA		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**140.82**	0.00	-14.89	**110.90**	0.00	36.66
Min Mx	**-140.82**	0.00	-8.90	**-110.90**	0.00	-35.99
Max My	-1.68	**0.00**	258.12	0.00	**0.00**	4.19
Min My	-140.82	**0.00**	-8.90	-34.94	**0.00**	-0.32
Max Mz	-1.28	0.00	**303.92**	16.70	0.00	**277.36**
Min Mz	1.00	0.00	**-300.58**	-16.70	0.00	**-277.37**

Table 14.14 Support Moments in EQ Zone IV for Composite Structure

	ESMA			LDA		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**219.74**	-0.001	-16.655	**185.63**	0.00	1.814
Min Mx	**-219.75**	-0.002	-6.776	**-185.63**	0.00	-1.82
Max My	-1.721	**0.001**	377.151	-0.012	**0.00**	0.193
Min My	-219.75	**-0.002**	-6.776	-7.785	**0.00**	-524.335
Max Mz	-1.31	0.00	**448.75**	7.789	0.00	**524.34**
Min Mz	0.909	-0.001	**-456.59**	-7.785	0.00	**-524.33**

It is clear from **Tables 14.12 - 14.14**, that the support moments in global X direction is comparatively low by LDA compared to ESMA. While for Z direction values of moments are nearly same for both the methods of earthquake analysis. Thus, from **Tables 14.9** to **14.14,** one can say that, the LDA will lead to lower size of foundations compared to ESMA.

Table 14.15 Beam End Forces in EQ Zone II for Composite Structure

	ESMA			LDA		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**8420.58**	-0.222	0.00	**8423.48**	-0.088	0.00
Min Fx	**-539.50**	0.112	-11.906	**-584.66**	0.025	-6.901
Max Fy	0.00	**198.37**	0.00	0.00	**198.93**	0.00
Min Fy	0.00	**-199.97**	0	0.00	**-199.99**	0.00
Max Fz	4270.20	-1.252	**28.64**	6344.96	9.439	**26.25**
Min Fz	5984.87	-1.688	**-28.64**	1621.44	-10.48	**-26.25**

Table 14.16 Beam End Forces in EQ Zone III for Composite Structure

	ESMA			LDA		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**8421.10**	-0.08	0.00	**8423.32**	-0.006	0.00
Min Fx	**-848.56**	0.169	-19.618	**-924.86**	0.093	-9.76
Max Fy	0.00	**198.23**	0.00	0.00	**202.02**	0.00
Min Fy	0.00	**-199.97**	0.00	0.00	**-202.04**	0.00
Max Fz	3797.92	-0.351	**39.527**	6634.19	8.717	**34.27**
Min Fz	6477.75	-1.123	**-39.527**	655.46	-8.864	**-34.27**

Table 14.17 Beam End Forces in EQ zone IV for Composite Structure

	ESMA			LDA		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**8425.1**	0.249	0.00	**8420.28**	-0.142	0.00
Min Fx	**-1274.66**	0.239	-33.301	**-1390.22**	0.23	-10.06
Max Fy	0.00	**198.02**	0.00	0.00	**201.87**	0.00
Min Fy	0.00	**-199.96**	0.00	0.00	**-201.89**	0.00
Max Fz	3126.83	0.04	**59.46**	6904.19	-15.591	**57.21**
Min Fz	7129.59	-0.858	**-59.46**	913.67	15.591	**-57.21**

It can be seen from **Tables 14.15 - 14.17**, that maximum values of Beam End Forces for all i.e. Fx, Fy and Fz are almost same by both the methods of analysis for all the three earthquake zones.

Table 14.18 Beam End Moments in EQ Zone II for Composite Structure

	ESMA			LDA		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**23.81**	0.00	0.00	**28.98**	0.00	0.00
Min Mx	**-23.87**	0.00	0.00	**-31.62**	0.00	0.00
Max My	0.00	**98.15**	-9.24	0.00	**83.82**	41.08
Min My	0.00	**-98.15**	-13.62	0.00	**-83.82**	-44.79
Max Mz	0.00	0.573	**208.00**	0.00	16.685	**272.49**
Min Mz	0.475	0.00	**-579.35**	-0.335	0.00	**-579.40**

Table 14.19 Beam End Moments in EQ Zone III for Composite Structure

	ESMA			LDA		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**30.37**	0.00	0.00	**34.68**	0.00	0.00
Min Mx	**-30.88**	0.00	0.00	**-40.52**	0.00	0.00
Max My	0.00	**140.82**	-8.908	0.00	**110.90**	36.50
Min My	0.00	**-140.82**	-14.892	0.00	**-110.90**	-35.825
Max Mz	0.001	1.283	**303.92**	0.00	16.70	**277.36**
Min Mz	0.32	0	**-579.32**	-2.98	0	**-585.54**

Table 14.20 Beam End Moments in EQ Zone IV for Composite Structure

	ESMA			LDA		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**37.34**	0.00	0.00	**51.18**	0.00	0.00
Min Mx	**-39.28**	0.00	0.00	**-53.34**	0.00	0.00
Max My	0.00	**219.75**	-6.77	0.00	**185.63**	33.83
Min My	0.00	**-219.74**	-16.65	0.00	**-185.63**	-33.84
Max Mz	0.00	1.31	**448.75**	0.00	7.827	**524.34**
Min Mz	0.296	0.00	**-579.29**	-3.151	0.00	**-585.08**

For all the three earthquake zones, **Tables 14.18 - 14.20** show that the values of beam end moments are nearly same with difference of about 5% in all the three global directions by both the methods of analysis.

Table 14.21 Nodal Displacements in EQ Zone III by ESMA

	Composite Structure			Concrete Structure		
	X mm	Y mm	Z mm	X mm	Y mm	Z mm
Max X	**82.38**	-15.73	3.56	**40.46**	-12.29	-0.70
Min X	**-82.38**	-27.81	-2.04	**-40.46**	-8.44	0.70
Max Y	0.73	**3.48**	47.83	-0.25	**0.53**	19.74
Min Y	0.40	**-45.85**	-0.67	0.00	**-24.9**	0.00
Max Z	-3.53	-15.34	**93.66**	-0.37	-6.975	**29.97**
Min Z	-7.81	-16.21	**-93.66**	0.38	-13.75	**-29.98**

Table 14.22 Nodal Displacements in EQ Zone III by LDA

	Composite Structure			Concrete Structure		
	X mm	Y mm	Z mm	X mm	Y mm	Z mm
Max X	**59.66**	-13.63	3.43	**53.054**	-9.58	0.034
Min X	**-59.66**	-25.69	-2.38	**-53.05**	-14.77	0.027
Max Y	1.83	**5.44**	42.56	0.00	**0.89**	26.37
Min Y	-0.63	**-45.51**	-1.05	0.02	**-25.06**	-0.003
Max Z	1.64	-11.04	**64.36**	-0.06	-12.31	**39.59**
Min Z	-3.85	-14.16	**-64.36**	-0.07	-12.03	**-39.6**

In **Tables 14.21** and **14.22**, nodal displacements are compared in earthquake zone III for composite and concrete structure. Here, displacement values for composite structure are higher compared to concrete structure in both the methods. This is because of the end conditions of the beams. In case of composite structure end conditions of the beams are considered as flexible, while for concrete structure rigid end conditions are considered.

Table 14.23 Support Reactions in EQ Zone III by ESMA

	Composite Structure			Concrete Structure		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**237.28**	1554.64	0.00	**173.40**	6240	5.44
Min Fx	**-237.28**	1554.64	0.00	**-173.69**	10400	9.07
Max Fy	0.00	**11205.21**	0.00	-0.07	**14689**	0.00
Min Fy	16.12	**-1263.33**	-167.12	0.62	**-285**	-32.49
Max Fz	-0.28	2415.17	**254.77**	-1.67	10439	**181.68**
Min Fz	0.28	2415.17	**-254.77**	-1.31	10438	**-181.73**

Table 14.24 Support Reactions in EQ Zone III by LDA

	Composite Structure			Concrete Structure		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**637.29**	7860.64	-167.55	**346.91**	11213.52	-7.19
Min Fx	**-637.29**	2968.92	167.55	**-346.91**	11214.51	7.12
Max Fy	0.12	**11208.80**	0.00	-0.07	**15467.52**	0.00
Min Fy	27.34	**-1349.68**	-120.41	0.62	**-308.66**	-35.34
Max Fz	6.94	9829.58	**627.41**	-0.48	11305.73	**355.17**
Min Fz	-6.94	3876.44	**-627.41**	0.48	11122.31	**-355.24**

Table 14.25 Support Moments in EQ Zone III by ESMA

	Composite Structure			Concrete Structure		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**262.96**	0.00	-8.83	**775.93**	0.35	5.58
Min Mx	**-262.96**	0.00	-3.47	**-775.89**	-0.49	6.78
Max My	-142.66	**0.00**	2.65	5.35	**1.61**	773.88
Min My	226.55	**0.00**	-14.40	8.91	**-1.7**	-774.83
Max Mz	-1.90	0.00	**549.68**	5.35	1.61	**773.88**
Min Mz	1.90	0.00	**-549.68**	8.91	-1.73	**-774.83**

Table 14.26 Support Moments in EQ Zone III by LDA

	Composite Structure			Concrete Structure		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**241.24**	0.00	26.58	**1366.88**	-0.07	-0.19
Min Mx	**-241.24**	0.00	-39.20	**-1366.85**	-0.07	0.19
Max My	0.00	**0.00**	1.89	0.00	**0.90**	462.46
Min My	11.92	**0.00**	869.44	-458.57	**-0.23**	-3.51
Max Mz	11.92	0.00	**882.07**	-6.87	-0.07	**1332.36**
Min Mz	-11.92	0.00	**-882.07**	6.914	-0.07	**-1332.36**

In **Table 14.23** to **14.26**, Support reactions and moments obtained for composite and concrete structure in EQ zone III are compared the reaction in downward direction (Fy) are much less for composite structure. But, forces in the two orthogonal directions are higher compared to those for concrete structure. Also the moments in two orthogonal horizontal directions for composite structure are least. In short, one can say that the area required for the foundations in case of composite structure will be less compared to that for concrete structure as values of reactions in Y direction and moments in X and Z directions are less for composite structure.

Table 14.27 Beam End Forces in EQ Zone III by ESMA

	Composite Structure			Concrete Structure		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**11205.21**	0.00	0.00	**14689.2**	0.07	0.00
Min Fx	**-1180.96**	0.30	-22.53	**-284.87**	-0.62	-32.49
Max Fy	0.00	**260.94**	0.00	0.00	**334.70**	0.00
Min Fy	0.00	**-260.94**	0.00	0.00	**-335**	0.00
Max Fz	5339.34	0.22	**67.16**	10438.67	1.67	**181.68**
Min Fz	8907.70	0.35	**-67.16**	10437.82	1.31	**-181.73**

Table 14.28 Beam End Forces in EQ Zone III by LDA

	Composite Structure			Concrete Structure		
	Fx kN	Fy kN	Fz kN	Fx kN	Fy kN	Fz kN
Max Fx	**11208.80**	0.126	0	**15467.5**	0.07	0.008
Min Fx	**-1286.88**	0.205	-20.42	**-308.66**	-0.62	-35.34
Max Fy	0.00	**262.16**	0.00	0.00	**369.19**	0.00
Min Fy	0.00	**-262.16**	0.00	0.00	**-369.18**	0.00
Max Fz	9569.92	6.94	**72.82**	11305.73	0.551	**355.17**
Min Fz	3897.62	-8.67	**-72.82**	11122.31	-0.551	**-355.24**

Table 14.29 Beam End Moments in EQ Zone III by ESMA

	Composite Structure			Concrete Structure		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**33.33**	0.00	0.00	**6.66**	0.00	216.47
Min Mx	**-62.27**	0.00	0.00	**-6.14**	0.00	263.61
Max My	0.00	**262.96**	-3.47	-0.49	**775.90**	6.78
Min My	0.00	**-262.96**	-8.83	0.35	**-776**	5.58
Max Mz	0.00	1.90	**549.68**	1.61	-5.35	**773.88**
Min Mz	0.295	0.00	**-762.06**	-1.73	-8.91	**-774.83**

Table 14.30 Beam End Moments in EQ Zone III by LDA

	Composite Structure			Concrete Structure		
	Mx kNm	My kNm	Mz kNm	Mx kNm	My kNm	Mz kNm
Max Mx	**47.87**	0	0	**17.29**	0	217.24
Min Mx	**-89.67**	0	0	**-8.94**	0	156.41
Max My	0.00	**241.24**	26.58	-0.07	**1366.84**	0.43
Min My	0.00	**-241.24**	-39.20	-0.07	**-1366.8**	-0.43
Max Mz	0.00	11.92	**882.07**	-0.07	7.16	**1332.36**
Min Mz	0.00	-11.92	**-882.07**	-0.07	-7.20	**-1332.36**

Beam end forces and moments for composite and concrete structures have been compared for earthquake zone III in **Table 14.27** to **14.30**. Here, values of reactions as well as moments are very less in all the directions for composite structure compared to concrete structure. Because of lower values of forces and moments in case of composite structure, it is possible to have lighter overall weight of the structure and thus reduction in seismic forces and base shear.

Table 14.31 Frequency and Time Period for Composite Structure

MODE	ZONE II		ZONE III		ZONE IV	
	Frequency Cycles/ Second	Time Period Second	Frequency Cycles/ Second	Time Period Second	Frequency Cycles/ Second	Time Period Second
1	0.443	2.259	0.484	2.064	0.493	2.028
2	0.473	2.112	0.516	1.939	0.544	1.837
3	0.804	1.244	0.875	1.143	0.92	1.087
4	1.538	0.65	1.661	0.601	1.787	0.559
5	1.635	0.611	1.817	0.55	1.894	0.528
6	2.48	0.403	2.677	0.373	2.867	0.348

In **Table 14.31**, for various earthquake zones, frequencies and time period for first six mode shapes have been compared. In each EQ zone, frequency increases with decrease in time period.

Maximum shear force at each storey level for different earthquake zones is shown in **Table 14.32**. Here, shear in both principle directions are same but increases with increase of earthquake zone.

Table 14.32 Storey Shear for Different EQ Zones for Composite Structure

STOREY NO.	Peak Storey Shear In kN					
	ZONE II		ZONE III		ZONE IV	
	X	Z	X	Z	X	Z
11	191.53	195.98	384.12	394.43	573.59	596.40
10	431.31	419.54	741.00	717.79	1098.89	1078.27
9	573.28	533.54	940.43	858.67	1393.44	1283.09
8	634.96	563.27	1020.89	873.79	1514.02	1301.82
7	666.48	565.11	1069.56	875.58	1577.40	1307.82
6	714.94	603.39	1157.37	964.49	1699.41	1451.52
5	803.7	707.65	1309.45	1158.42	1927.73	1750.93
4	918.84	851.14	1495.58	1395.41	2216.32	2108.21
3	1027.84	988.92	1664.78	1608.40	2483.20	2424.19
2	1102.84	1087.89	1776.52	1754.92	2662.78	2638.99
1	1130.71	1130.71	1816.75	1816.75	2729.56	2729.56

Table 14.33 shows self weight of the composite frame for different methods of dynamic analysis under different seismic zones. Here, increase in weight with increase of zone is only about 8%, while corresponding increase in base shear (Table 14.32) is about 35%. Also maximum difference in weight by different methods is 11.60%.

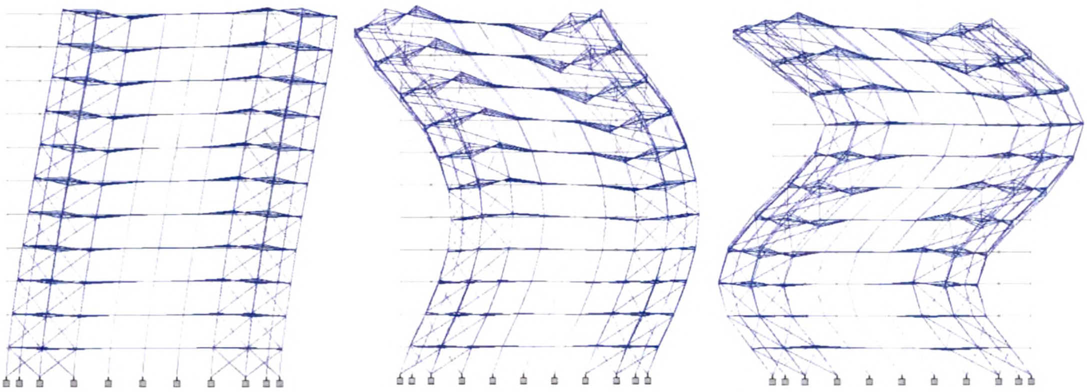
Table 14.33 Self Weight of Composite Frame for Different EQ Zones

ZONE	Weight in kN		Weight in Kg		Difference In %
	ESMA	LDA	ESMA	LDA	
II	3377.41	3524.65	344376	359389	4.18
III	3400.19	3846.21	346699	392177	11.60
IV	3512.35	3966.13	358135	404405	11.44

Frequencies and time period between composite and concrete structure for different mode shapes have been compared in **Table 14.34**. for earthquake zone III. Basic mode shapes of eleven-storey composite building for first three basic modes of vibration are depicted in **Fig. 14.31** followed by results for frequency and time period for different zones and different modes in **Table 14.35** and **14.36**.

Table 14.34 Frequency and Time Period for EQ Zone III

MODE	Composite Structure		Concrete Structure	
	Frequency Cycles/Second	Time Period Second	Frequency Cycles/Second	Time Period Second
1	0.484	2.064	0.516	1.940
2	0.516	1.939	0.576	1.736
3	0.875	1.143	0.582	1.718
4	1.661	0.601	1.331	0.752
5	1.817	0.55	1.494	0.670
6	2.677	0.373	1.594	0.627



(a) 1st mode of vibration (b) 2nd mode of vibration (c) 3rd mode of vibration

Fig. 14.31 Basic Mode Shapes of an Eleven-Storey Composite Building

Table 14.35 Frequency and Time Period for Three Basic Mode Shapes

ZONE	1st MODE		2nd MODE		3rd MODE	
	Frequency Cycles/Second	Time Period Second	Frequency Cycles/Second	Time Period Second	Frequency Cycles/Second	Time Period Second
II	0.443	2.259	1.635	0.611	2.48	0.403
III	0.484	2.064	1.817	0.55	2.677	0.373
IV	0.493	2.028	1.894	0.528	2.867	0.348

Table 14.36 Frequency and Time Period for Basic Mode Shapes

MODE	Composite Structure		Concrete Structure	
	Frequency Cycles/Second	Time Period Second	Frequency Cycles/Second	Time Period Second
1st	0.484	2.064	0.582	1.718
2nd	1.817	0.55	1.494	0.670
3rd	2.677	0.373	1.594	0.627