Available online: www.ijsrmes.com

International Journal for Scientific Research in Modern Engineering and Science

International Journal for Scientific Research in Modern Engineering and Science, 3(4):46-57 (2022)

SEISMIC ANALYSIS OF MULTISTOREY BUILDING WITH FLOATING COLUMN

Vivek Kumar Mishra [1]*, Vijay Kumar Shukla [2], Aradhna Shrivastava [3]

[1]Mtech Student Structural Engineering, Vishwavidhyalaya Engineering college Ambikapur-CSVTU Bhilai

[2]Phd Scholar, Civil Engineering, Vishwavidhyalaya Engineering college Ambikapur-CSVTU Bhilai

[3]Phd Scholar, Civil Engineering, Vishwavidhyalaya Engineering college Ambikapur-CSVTU Bhilai

Abstract

In present scenario buildings with floating column is a typical feature in the modern multistory construction in urban India. Such features are highly undesirable in building built in seismically active areas. This study highlights the im-portance of explicitly recognizing the presence of the floating column in the analysis of building. Alternate measures, involving stiffness balance of the first storey and the storey above, are proposed to reduce the irregularity introduced by the floating columns. FEM codes are developed for 2D multi storey frames with and without floating column to study the responses of the structure under different earthquake excitation having different frequency content keeping the PGA and time duration factor constant. The time history of floor displacement, inter storey drift, base shear, overturning moment are computed for both the frames with and without floating column.

Keywords: *Finite Element, STADD Pro, Floating Column, Static Analysis*

* *Corresponding author*

1. INTRODUCTION

Many urban multistory buildings in India today have open first storey as an unavoidable feature. This is primarily being adopted to accommodate parking or reception lobbies in the first storey. Whereas the total seismic base shear as experienced by a building during an earthquake is dependent on its natural period, the seismic force distribution is dependent on the distribution of stiffness and mass along the height.

The behavior of a building during earthquakes depends critically on its overall shape, size and geometry, in addition to how the earthquake forces are carried to the ground. The earthquake forces developed at different floor levels in a building need to be brought down along the height to the ground by the shortest path; any deviation or discontinuity in this load transfer path results in poor performance of the building. Buildings with vertical setbacks (like the hotel buildings with a few storey wider than the rest) cause a sudden jump in earthquake forces at the level of discontinuity. Buildings that have fewer columns or walls in a particular storey or with unusually tall storey tend to damage or collapse which is initiated in that storey. Many buildings with an open ground storey intended for parking collapsed or were severely damaged in Gujarat during the 2001 Bhuj earthquake. Buildings with columns that hang or float on beams at an intermediate storey and do not go all the way to the foundation, have discontinuities in the load transfer path..

A column is supposed to be a vertical member starting from foundation level and transferring the load to the ground. The term floating column is also a vertical element which (due to architectural design/ site situation) at its lower level (termination Level) rests on a beam which is a horizontal member. The beams in turn transfer the load to other columns below it.

There are many projects in which floating columns are adopted, especially above the ground floor, where transfer girders are employed, so that more open space is available in the ground floor. These open spaces may be required for assembly hall or parking purpose. The transfer girders have to be designed and detailed properly, especially in earth quake zones. The column is a concentrated load on the beam which supports it. As far as analysis is concerned, the column is often assumed pinned at the base and is therefore taken as a point load on the transfer beam. STAAD Pro, ETABS and SAP2000 can be used to do the analysis of this type of structure. Floating columns are competent enough to carry gravity loading but transfer girder must be of adequate dimensions (Stiffness) with very minimal deflection.

Looking ahead, of course, one will continue to make buildings interesting rather than monotonous. However, this need not be done at the cost of poor behavior and earthquake safety of buildings. Architectural features that are detrimental to earthquake response of buildings should be avoided. If not, they must be minimized. When irregular features are included in buildings, a considerably higher level of engineering effort is required in the structural design and yet the building may not be as good as one with simple architectural features.

Hence, the structures already made with these kinds of discontinuous members are endangered in seismic regions. But those structures cannot be demolished, rather study can be done to strengthen the structure or some remedial features can be suggested. The columns of the first storey can be made stronger, the stiffness of these columns can be increased by retrofitting or these may be provided with bracing to decrease the lateral deformation.

2. OBJECTIVE OF STUDY

The objective of the present work is to study the behavior of multistory buildings with floating columns under earth-quake excitations.

Finite element method is used to solve the dynamic governing equation. Linear time history analysis is carried out for the multistory buildings under different earthquake loading of varying frequency content. The base of the building frame is assumed to be fixed. Newmark’s direct integration scheme is used to advance the solution in time.

3. FORMULATION OF FINITE ELEMENT

The finite element method (FEM), which is sometimes also referred as finite element analysis (FEA), is a computational technique which is used to obtain the solutions of various boundary value problems in engineering, approximately. Boundary value problems are sometimes also referred to as field value problems. It can be said to be a mathematical problem wherein one or more dependent variables must satisfy a differential equation everywhere within the domain of independent variables and also satisfy certain specific conditions at the boundary of those domains. The field value prob-lems in FEM generally has field as a domain of interest which often represent a physical structure. The field variables are thus governed by differential equations and the boundary values refer to the specified value of the field variables on the boundaries of the field. The field variables might include heat flux, temperature, physical displacement, and fluid velocity depending upon the type of physical problem which is being analyzed.

3.1 Static analysis

3.1.1 Plane frame element

The plane frame element is a two-dimensional finite element with both local and global coordinates. The plane frame ele-ment has modulus of elasticity E, moment of inertia I, cross-sectional area A, and length L. Each plane frame element has two nodes and is inclined with an angle of θ measured counterclockwise from the positive global X axis as shown in figure. Let C= COS θ and S= SIN θ.

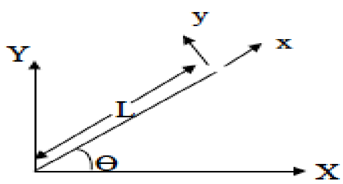

Fig 1.1 The plane frame element

It is clear that the plane frame element has six degree of freedom – three at each node (two displacements and a rotation). The sign convention used is that displacements are positive if they point upwards and rotations are positive if they are counterclockwise. Consequently, for a structure with n nodes, the global stiffness matrix K will be 3n X 3n (since we have three degrees of freedom at each node). The global stiffness matrix K is assembled by making calls to the MATLAB function Plane Frame Assemble which is written specially for this purpose. Once the global stiffness matrix K is obtained, we have the following structure equation: [K]{U} ={F}

Where [K] is stiffness matrix, {U} is the global nodal displacement vector and {F} is the global nodal force vector. At this step boundary conditions are applied manually to the vectors U and F. Then the matrix equation (3.1) is solved by partitioning and Gaussian elimination. Finally, once the unknown displacements and reactions are found, the nodal force vector is obtained for each element as follows: {f} = [k] [R] {u}

Where {f} is the 6 X 1 nodal force vector in the element and {u} is the 6 X 1 element displacement vector. The matrices [k] and [R] are given by the following:

$$[k] = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & \frac{-EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & \frac{-12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & \frac{6EI}{L^2} & \frac{2EI}{L} \\ \frac{-EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & \frac{-12EI}{L^3} & \frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & \frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix}$$

$$[R] = \begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

The first and second element in the vector {u} are the two displacements while the third element is the rotation, respectively, at the first node, while the fourth and fifth element are the two displacements while the sixth element is the rotation, respectively, at the second node.

3.1.2 Steps followed for the analysis of frame

1. Discretising the domain: Dividing the element into number of nodes and numbering them globally i;e breaking down the domain into smaller parts.
2. Writing of the Element stiffness matrices: The element stiffness matrix or the local stiffness matrix is found for all elements and the global stiffness matrix of size 3n x 3n is assembled using these local stiffness matrices.
3. Assembling the global stiffness matrices: The element stiffness matrices are combined globally based on their degrees of freedom values.
4. Applying the boundary condition: The boundary element condition is applied by suitably deleting the rows and columns which are not of our interest.
5. Solving the equation: The equation is solved in MATLAB to give the value of U.
6. Post- processing: The reaction at the support and internal forces are calculated.

3.2 Dynamic analysis

Dynamic analysis of structure is a part of structural analysis in which behavior of flexible structure subjected to dynamic loading is studied. Dynamic load always changes with time. Dynamic load comprises of wind, live load, earthquake load etc. Thus in general we can say almost all the real life problems can be studied dynamically.

If dynamic loads changes gradually the structure's response may be approximately by a static analysis in which inertia forces can be neglected. But if the dynamic load changes quickly, the response must be determined with the help of dy-namic analysis in which we cannot neglect inertial force which is equal to mass time of acceleration (Newton's 2nd law).

Mathematically, F = M x a

Where F is inertial force, M is inertial mass and 'a' is acceleration. Furthermore, dynamic response (displacement and stresses) are generally much higher than the corresponding static displacements for same loading amplitudes, especially at resonant conditions.

The real physical structures have many numbers of displacement. Therefore the most critical part of structural analysis is to create a computer model, with the finite number of mass less member and finite number of displacement of nodes which simulates the real behavior of structures. Another difficult part of dynamic analysis is to calculate energy dissipation and to boundary condition. So it is very difficult to analyze structure for wind and seismic load. This difficulty can be re-duced using various programming techniques. In our project we have used finite element analysis and programmed in MATLAB.

3.3 Time history analysis

A linear time history analysis overcomes all the disadvantages of modal response spectrum analysis, provided non-linear behavior is not involved. This method requires greater computational efforts for calculating the response at discrete time. One interesting advantage of such procedure is that the relative signs of response qualities are preserved in the response histories. This is important when interaction effects are considered in design among stress resultants.

Here dynamic response of the plane frame model to specified time history compatible to IS code spectrum and Elcentro (EW) has been evaluated.

The equation of motion for a multi degree of freedom system in matrix form can be expressed as

$$[m]\{\ddot{x}\}+ [c]\{\dot{x}\}+[k]\{x\}=-x\ddot{g}(t)[m]\{I\}$$

Where,

$[m]$= mass matrix $\quad$ $[k]$= stiffness matrix

$[c]$= damping matrix $\quad$ $\{I\}$= unit vector $\quad$ $x\ddot{g}(t)$= ground acceleration

The mass matrix of each element in global direction can be found out using following expression:

$$m = [T^T]\,[m_e]\,[T]$$

$$[m_e]= \frac{\rho A L}{420}\begin{bmatrix} 140 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22L & 0 & 54 & -13L \\ 0 & 22L & 4L^2 & 0 & 13L & -3L^2 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & -22L \\ 0 & -13L & -3L^2 & 0 & -22L & 4L^2 \end{bmatrix}$$

$$[T] = \begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \end{bmatrix}$$

The solution of equation of motion for any specified forces is difficult to obtain, mainly due to due to coupling variables {x} in the physical coordinate. In mode superposition analysis or a modal analysis a set of normal coordinates i.e principal coordinate is defined, such that, when expressed in those coordinates, the equations of motion becomes uncoupled. The physical coordinate {x} may be related with normal or principal coordinates {q} from the transformation expression as,

$\{ x \} = [\Phi] \{ q \}$ $\quad$ $[\Phi]$ is the modal matrix

Time derivative of $\{ x \}$ are,

$\{\dot{x}\} = [\Phi] \{\dot{q}\}$ $\quad$ $\{\ddot{x}\} = [\Phi] \{\ddot{q}\}$

Substituting the time derivatives in the equation of motion, and pre-multiplying by [Φ]T results in,

$[\Phi]T[m][\Phi]\{\ddot{q}\}+[\Phi]T[c][\Phi]\{\dot{q}\}+[\Phi]T[k][\Phi]\{q\}=(-x\dot{g}(t)[\Phi]T[m]\{I\})$

More clearly it can be represented as follows:

$[M]\{\ddot{q}\}+[C]\{\dot{q}\}+[K]\{q\}=\{Peff(t)\}$

Where,

$[M]=[\Phi]T[m][\Phi]$ $\quad$ $[C]=[\Phi]T[c][\Phi]=2\,\zeta\,[M]\,[\omega]$

$[K]=[\Phi]T[k][\Phi]$ $\quad$ $\{Peff(t)\}=(-x\dot{g}(t)[\Phi]T[m]\{I\}$

[M], [C] and [K] are the diagonalised modal mass matrix, modal damping matrix and modal stiffness matrix, respectively, and {Peff(t)} is the effective modal force vector.

3.4 Newmark's method

Newmark's numerical method has been adopted to solve the equation 3.9. Newmark's equations are given by

$di+1=\dot{d}i+(\Delta t)(1-\gamma)\ddot{d}i+\gamma\ddot{d}i+1$ $\quad$ $\dot{d}i+1=\dot{d}i+(\Delta t)\dot{d}i+(\Delta t)2\,(0.5-\beta)\ddot{d}i+\beta\ddot{d}i+1$

Where β and γ are parameters chosen by the user. The parameter β is generally chosen between 0 and ¼, and γ is often taken to be ½. For instance, choosing γ = ½ and β = 1/6, are chosen, eq. correspond to those for which a linear acceleration assumption is valid within each time interval. For γ = ½ and β = ¼, it has been shown that the numerical analysis is stable; that is, computed quantities such as displacement and velocities do not become un-bounded regardless of the time step chosen.

To find $ddii+1$, we first multiply the mass matrix MM and then substitute the value of $dd\ddot{i}i+1$ into this eq. to obtain

$M\,\ddot{d}i+1=M\,di+(\Delta t)M\,\dot{d}i+(\Delta t)2M(1/2-\beta)\ddot{d}i+\beta(\Delta t)2(Fii+1-Kddi+1\,)$

Combining the like terms we obtain

$(M+\beta(\Delta t)2K)di+1=\beta(\Delta t)2Fi+1+M\,di+(\Delta t)M\,\dot{d}i+(\Delta t)2M(1/2-\beta)\ddot{d}i$

Finally, dividing above eq. by $\beta\beta(\Delta tt)2$, we obtain

$K'di+1=F'i+1$ $\quad$ $K'=K+1/\beta(\Delta t)2M$

$F'i+1=Fi+1+M/\beta(\Delta t)2(di+(\Delta t)di+(1/2-\beta)(\Delta t)2\ddot{d}i$

The solution procedure using Newmark's equations is as follows:

1. Starting at time t=0, $dd0$ is known from the given boundary conditions on displacement, and $d\dot{d}0$ is known from the initial velocity conditions.
2. Solve eq. at t=0 for $d0$ (unless $d0$ is known from an initial acceleration condition); that is, $d0=M-1(F0-Kd0)$
3. Solve eq. for $d1$, because $F'i+1$ is known for all time steps and , $d0$,$\dot{d}0$,$d0$ are known from steps 1 and 2.
4. Use eq. to solve for $\dot{d}1$ as $\quad$ $\dot{d}1=1/\beta(\Delta t)2(d1-d0-(\Delta t)\dot{d}0-(\Delta t)2(1/2-\beta)d0)$
5. Solve eq. 4.12 directly for $d\dot{d}1$
6. Using the results of steps 4 and 5, go back to step 3 to solve for $d2$ and then to steps 4 and 5 to solve for $d2$ and $\dot{d}2$. Use steps 3-5 repeatedly to solve for $di+1$, $\dot{d}i+1$ and $\ddot{d}i+1$.

4. RESULT & DISCUSSION

The behavior of building frame with and without floating column is studied under static load, free vibration and forced vibration condition. The finite element code has been developed in MATLAB platform.

4.1 Static analysis

A four storey two bay 2d frame with and without floating column are analyzed for static loading using the present FEM code and the commercial software STAAD Pro.

Modeling in FEM as well as STADD. Pro

The following are the input data of the test specimen:

Size of beam – 0.1 X 0.15 m Size of column – 0.1 X 0.125 m

Span of each bay – 3.0 m Storey height – 3.0 m

Modulus of Elasticity, E = 206.84 X 106kN/m2 Support condition – Fixed

Loading type – Live (3.0 kN at 3rd floor and 2 kN at 4th floor)

Fig. 4.1 and Fig.4.2 show the sketchmatic view of the two frame without and with floating column respectively. From Table 4.1 and 4.2, we can observe that the nodal displacement values obtained from present FEM in case of frame with floating column are more than the corresponding nodal displacement values of the frame without floating column. Table 4.3 and 4.4 show the nodal displacement value obtained from STAAD Pro of the frame without and with floating column respectively and the result are very comparable with the result obtained in present FEM.

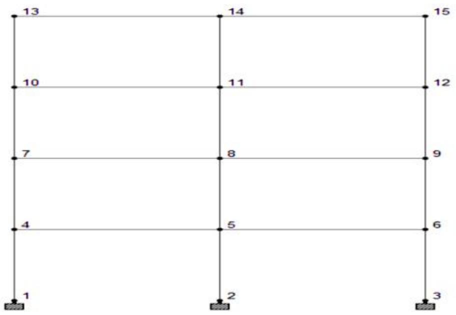

Fig. 4.1 2D Frame with usual columns

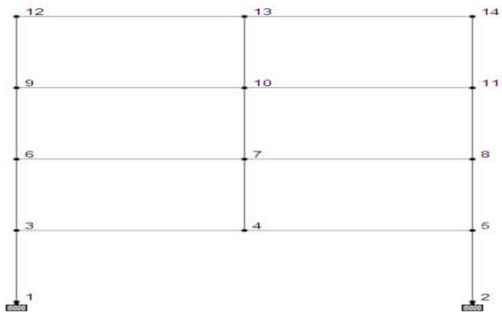

Fig.4.2 2D Frame with Floating column

Table 1 Global deflection at each node for general frame obtained in present FEM as well ass STADD Pro.

Node	Horizontal X mm	Vertical Y mm	Rotational rZ rad
1	0	0	0
2	0	0	0
3	0	0	0
4	1.6	0	0
5	1.6	0	0
6	1.6	0	0
7	3.8	0	0
8	3.8	0	0
9	3.8	0	0
10	5.8	0	0

11	5.8	0	0
12	5.8	0	0
13	6.7	0	0
14	6.7	0	0
15	6.7	0	0

Table 2 Global deflection at each node for frame with floating column obtained in present FEM and STADD Pro.

Node	Horizontal X mm	Vertical Y mm	Rotational rZ rad
1	0	0	0
2	0	0	0
3	0	0	0
4	2.6	0	0
5	2.6	0	0
6	2.6	0	0
7	4.8	0	0
8	4.8	0	0
9	4.8	0	0
10	6.8	0	0
11	6.8	0	0
12	6.8	0	0
13	7.7	0	0
14	7.7	0	0
15	7.7	0	0

4.2 Free Vibrational Analysis

A two storey one bay 2D frame is taken. Fig.4.3 shows the sketchmatic view of the 2D frame. The results obtained are compared with Maurice Petyt[21]. The input data are as follows:

Span of bay = 0.4572 m Storey height = 0.2286 m

Size of beam = (0.0127 x 0.003175) m Size of column = (0.0127 x 0.003175) m

Modulus of elasticity, E = 206.84 x106 kN/m2 Density, ρ = 7.83 x 103 Kg/m3

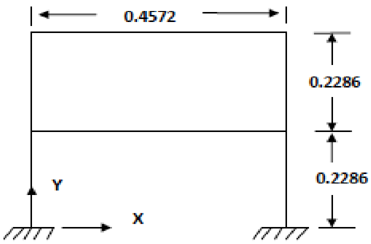

Fig-4.3Geometry of the 2-dimensional framework. Dimensions are in meter

The value of free vibration frequency of the 2D frame calculated in present FEM. It is observed from Table 4.5 that the present results are in good agreement with the result given by Maurice Petyt].

Table-3 Free vibration frequency (Hz) of the 2D frame without floating column

Mode	Maurice Petyt	Present FEM	% Variation
1	15.14	15.14	0.00
2	53.32	53.31	0.02

3	155.48	155.52	0.03
4	186.51	186.59	0.04
5	270.85	270.64	0.05

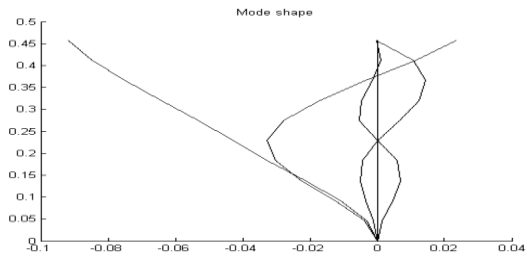

Fig-4.4 Mode shape of the 2D framework

4.3 Forced Vibrational Analysis

For the forced vibration analysis, a two bay four storey 2D steel frame is considered. The frame is subjected to ground motion, the compatible time history of acceleration as per spectra of IS 1893 (part 1): 2002.

The dimension and material properties of the frame is as follows:

Young's modulus. E= 206.84 x 106 kN/m2 Density, ρ = 7.83 x103 Kg/m3

Size of beam = (0.1 x 0.15) m Size of column = (0.1 x 0.125) m

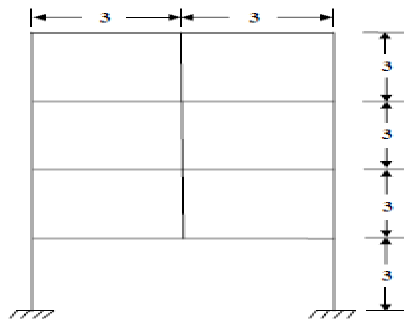

Fig. 4.5 Geometry of the 2-dimensional frame with floating column. Dimensions are in meter

The compatible time history as per spectra of IS 1893 (part 1): 2002. Fig.4.7 and 4.8 show the maximum top floor displacement of the 2D frame obtained in present FEM and STAAD Pro respectively.

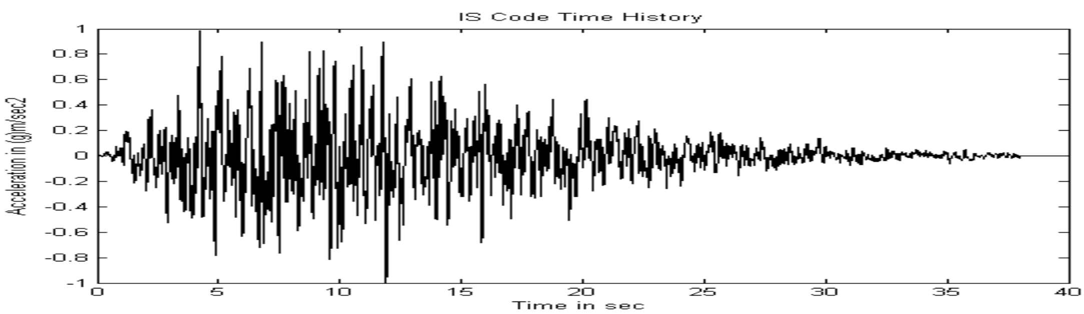

Fig-4.6 Compatible time history as per spectra of IS 1893 (part 1): 2002

Free vibration frequencies of the 2D steel frame with floating column are presented in Table 4.6. In this table the values obtained in present FEM and STAAD Pro are compared. Table 4.7 shows the comparison of maximum top floor displacement of the frame obtained in present FEM and STAAD Pro which are in very close agreement.

Table 4 Comparison of predicted frequency (Hz) of the 2D steel frame with floating column obtained in present FEM and STAAD Pro.

Mode	Stadd Pro	Present FEM	% Variation
1	2.16	2.17	0.28
2	6.78	7.00	3.13
3	11.57	12.62	8.32
4	12.37	13.04	5.14

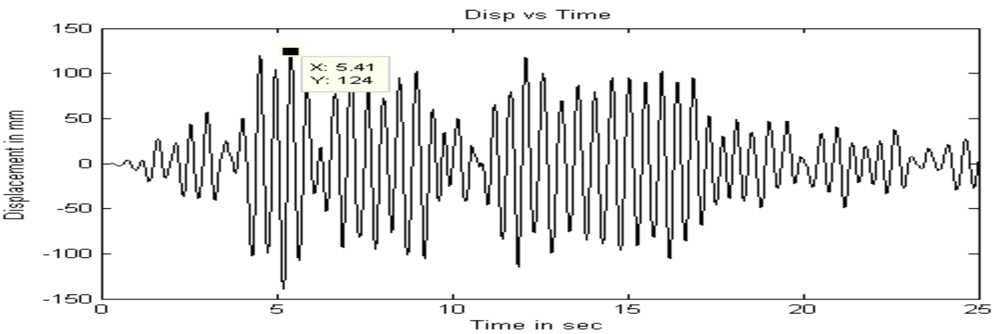

Fig-4.7 Displacement vs time response of the 2D steel frame with floating column obtained in present FEM

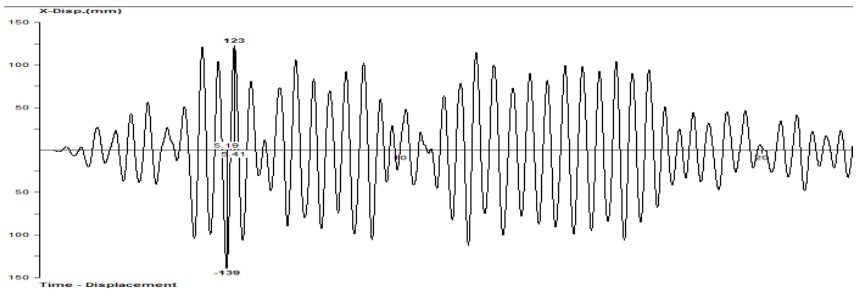

Fig-4.8 Displacement vs time response of the 2D steel frame with floating column obtained in STAAD Pro

Table 5 Comparison of predicted maximum top floor displacement (mm) of the 2D steel frame with floating column in present FEM and STAAD Pro.

Maximum top floor displacement (mm)		% Variation
STADD Pro	Present FEM	
123	124	0.81

The frame used in Example 4.3 is taken only by changing the material property and size of structural members. Size and material property of the structural members are as follows:

Size of beam = (0.25 x 0.3) m Size of column = (0.25 x 0.25) m

Young’s modulus, E= 22.36 x 109 N/m2 Density, ρ = 2500 Kg/m3

Fig.4.9 and 4.10 show the maximum top floor displacement of the 2D frame obtained in STAAD Pro and present FEM and respectively. Free vibration frequencies of the 2D concrete frame with floating column are presented in Table 4.8. In this table the values obtained in present FEM and STAAD Pro are compared. Table 4.9

shows the comparison of maximum top floor displacement of the frame obtained in present FEM and STAAD Pro which are in very close agreement

Table 6 Comparison of predicted frequency(Hz) of the 2D concrete frame with floating column obtained in present FEM and STAAD Pro.

Mode	Stadd Pro	Present FEM	% Variation
1	2.486	2.52	1.37
2	7.78	8.09	3.98
3	13.349	14.67	9.89
4	13.938	14.67	5.25

Table 7 Comparison of predicted maximum top floor displacement (mm) of the 2D concrete frame with floating column obtained in present FEM and STAAD Pro.

Maximum top floor displacement (mm)		% Variation
STADD Pro	Present FEM	
118	121.2	2.71

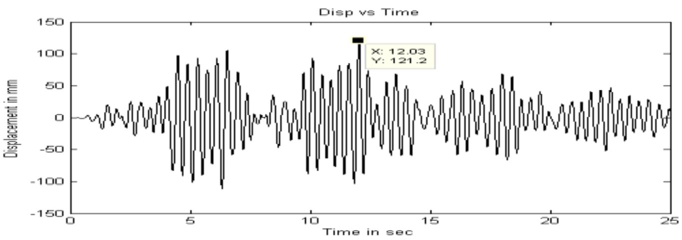

Fig. 4.10 Displacement vs time response of the 2D

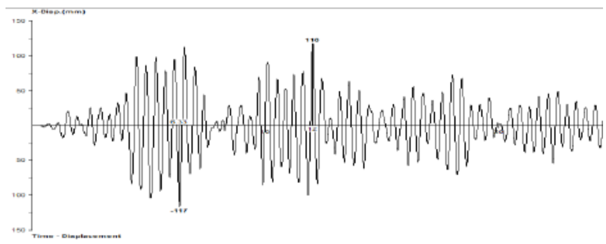

Fig 4.11 Displacement vs time response of the 2D

Concrete frame with floating column plotted in present FEM concrete frame with floating column given by STAAD Pro. The behavior of multistory building with and without floating column is studied under different earthquake excitation. The compatible time history and Elcentro earthquake data has been considered. The PGA of both the earthquake has been scaled to 0.2g and duration of excitation are kept same. A finite element model has been developed to study the dynamic behavior of multi-story frame. The static and free vibration results obtained using present finite element code are validated. The dynamic analysis of frame is studied by varying the column dimension. It is concluded that with increase in ground floor column the maximum displacement, inter storey drift values are reducing. The base shear and overturning moment vary with the change in column dimension.

5. CONCLUSION

The behavior of multistory building with and without floating column is studied under different earthquake excitation. The compatible time history and Elcentro earthquake data has been considered. The PGA of both the earthquake has been scaled to 0.2g and duration of excitation are kept same. A finite element model has been developed to study the dy-namic behavior of multi story frame. The static and free vibration results obtained using present finite element code are validated. The dynamic analysis of frame is studied by varying the column dimension. It is concluded that with increase in ground floor column the maximum displacement, inter storey drift values are reducing. The base shear and overturning moment vary with the change in column dimension.

References

[1] Agarwal Pankaj, Shrikhande Manish (2009), “Earthquake resistant design of structures”, PHI learning private limited, New Delhi.

[2] Arlekar Jaswant N, Jain Sudhir K. and Murty C.V.R, (1997), “Seismic Response of RC Frame Buildings with Soft First Storeys”. Proceedings of the CBRI Golden Jubilee Conference on Natural Hazards in Urban Habitat, 1997, New Delhi.

[3] Awkar J. C. and Lui E.M, “Seismic analysis and response of multistory semirigid frames”, Journal of Engineering Structures, Volume 21, Issue 5, Page no: 425-442, 1997.

[4] Balsamoa A, Colombo A, Manfredi G, Negro P & Prota P (2005), ”Seismic behavior of a full-scale RC frame repaired using CFRP laminates”. Engineering Structures 27 (2005) 769–780.

[5] Bardakis V.G., Dritsos S.E. (2007), “Evaluating assumptions for seismic assessment of existing buildings “.Soil Dynamics and Earthquake Engineering 27 (2007) 223–233.

[6] Brodericka B.M., Elghazouli A.Y. and Goggins J, “Earthquake testing and response analysis of concentrically-braced sub-frames”, Journal of Constructional Steel Research, Volume 64, Issue 9, Page no: 997-1007,2008.