CHAPTER 13

SEISMIC EVALUATION USING OPENSEES SOFTWARE

13.1 THE OPEN SOURCE SOFTWARE OPENSEES

The Open System for Earthquake Engineering Simulation (OpenSEES) is a software framework for simulating the seismic response of structural and geotechnical systems. OpenSEES has been developed as the computational platform for research in performance-based earthquake engineering at the Pacific Earthquake Engineering Research Centre. It is an object-oriented software framework for simulation applications using finite element method. As open-source software, it has the potential for a community code for earthquake engineering. Steps involved in Opensees are as follows:

1. Building the model
2. Defining the analysis and
3. Pushover analysis

13.1.1 Building the Model

It involves the outlining of the problem to be solved with appropriate simplification without losing the desired level of accuracy for the analysis. Usually, modeling involves the following steps:

i. **Defining the geometry of the model:** It involves defining the dimensions of the structure to be analyzed, locations of various elements and nodes including control node and displacement and boundary conditions.

ii. **Defining various elements in the model:** Type of elements and geometric transformation required are defined.

iii. **Defining sections of various elements and their properties:** Sectional properties and Constituent Properties of materials used for the section are defined.

iv. **Defining gravity load on the structure:** Gravity loads on various elements and nodes are calculated and assigned accordingly.

v. **Defining lateral loads and distribution for seismic analysis:** Distributions of lateral loads based on mass/weight distributions along building height are calculated by following the codal provisions.

vi. **Defining recorders for getting analysis result:** Recorders for recording displacement of various nodes, support reactions, lateral drifts, element forces, section deformations of axial and curvature are to be defined in this step.

13.1.2 Defining the Analysis

Defining the analysis part involves the following:

i. **CONSTRAINTS handler:** Determines how the constraint equations are enforced in the analysis.

a)*Plain Constraints*: Removes constrained degrees of freedom from the system of equations (only for homogeneous equations).

b)*Lagrange Multipliers*: Uses the method of Lagrange multipliers to enforce constraints.

c)*Penalty Method*: Uses penalty numbers to enforce constraints - good for static analysis with non-homogeneous equations (rigid Diaphragm).

d)*Transformation Method*: Performs a condensation of constrained degrees of freedom.

ii. DOF numberer: Numbers the degrees of freedom in the domain and determines the mapping between equation numbers and degrees-of-freedom.

a)Plain: Uses the numbering provided by the user.

b)RCM: Renumbers the DOF to minimize the matrix band-width using the Reverse Cuthill-McKee algorithm.

iii. SYSTEM used for solving equations: Defines how to store and solve the system of equations in the analysis.

a)Linear Equation Solvers: Provide the solution of the linear system of equations Ku = P. Each solver is tailored to a specific matrix topology.

b)ProfileSPD: Direct profile solver for symmetric positive definite matrices.

c)BandGeneral: Direct solver for banded unsymmetrical matrices.

d)BandSPD: Direct solver for banded symmetric positive definite matrices.

e)SparseGeneral: Direct solver for unsymmetrical sparse matrices.

f)SparseSPD: Direct solver for symmetric sparse matrices.

g)UmfPack: Direct UmfPack solver for unsymmetric matrices.

iv. Convergence TEST: Used to accept the current state of the domain as being on the converged solution path or to determine if convergence has been achieved at the end of an iteration step.

a)NormUnbalance: Specifies a tolerance on the norm of the unbalanced load at the current iteration.

b)NormDispIncr: Specifies a tolerance on the norm of the displacement increments at the current iteration.

c)EnergyIncr: Specifies a tolerance on the inner product of the unbalanced load and displacement increments at the current iteration.

v. Solution ALGORITHM: Used to iterate from the last time step to the current.

a)Linear: Uses the solution at the first iteration and continues.

b)Newton: Uses the tangent at the current iteration to iterate to convergence.

c)ModifiedNewton: Uses the tangent at the first iteration to iterate to convergence.

vi. INTEGRATOR:

Static INTEGRATOR: Determines the next time step for an analysis.

a) LoadControl: Specifies the incremental load factor to be applied to the loads in the domain.

b) DisplacementControl: Specifies the incremental displacement at a specified DOF in the domain.

Transient INTEGRATOR: Determines the next time step for an analysis including inertial effects.

a)Newmark: The two parameter time-stepping method developed by Newmark.

b)HHT: The three-parameter Hilbert-Hughes-Taylor time-stepping method.

c)Central Difference: Approximates velocity and acceleration by centered finite differences of displacement.

vii. ANALYSIS: Defines what type of analysis is to be performed.

a) Static Analysis: Solves the KU=R problem, without the mass or damping matrices.

b) Transient Analysis: Solves the time-dependent analysis. The time step in this type of analysis is constant. The time step in the output is also constant.

c) Variable Transient Analysis: Performs the same analysis type as the Transient Analysis object. The time step, however, is variable. This method is used to construct a variable TimeStepDirectIntegrationAnalysis object.

13.1.3 Pushover Analysis

Pushover analysis is performed after building the model and gravity load analysis. Maximum displacement for pushover and displacement increments are defined.

13.2 PUSH OVER ANALYSIS OF A FRAME WITH OPENSEES

It is proposed to use OpenSEES to evaluate the capacity spectrum for a single bay two storey portal frame and compare the same with the results obtained by ETABS software. The performance point is obtained by using the capacity spectrum obtained from OpenSEES and carrying out manual calculations from the basic equations to obtain the performance point. The same is compared with that calculated by ETABS. A typical RC plane frame of single bay and two storey is considered as a verification problem. The results of the push over analysis by OpenSEES and ETABS are compared in order to verify the reliability of the work done.

13.2.1 Geometry of the Model

The parameters defined for the verification problem are as follows:

Typical bay width = 3m, typical storey height considered = 3m

Material considered is concrete with $E = 2.236 \times 10^{10}$ N/m^2

All columns and beams are considered of size 300mm x 300mm

Moment of inertia for all beams and columns is given as

$$I_{beam} = I_{column} = 1/12 * B * D^3 = 6.75 \times 10^{-4} \text{ m}^4$$

Loads considered on the frame:

Imposed load at each storey = 3 kN/m,

Parameters considered as per IS:1893, Part 1, 2002 **[24]** are :

Zone factor Z = 0.16, for seismic zone III

Type of soil = Medium

Importance factor I = 1

Response reduction factor R = 5 considering ductile detailing.

Earthquake forces calculated as per codal provision are :

At story 2 = 576 N

At story 1 = 144 N.

The problem is defined as shown in **Fig. 13.1**.

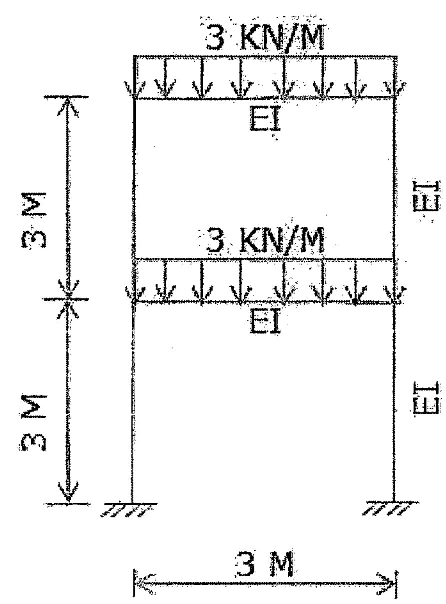


Fig. 13.1 The Verification Problem

13.2.2 Capacity Curve using OpenSEES and Capacity Spectrum

The mathematical model developed in OpenSEES with the joint number, element number and constraints marked is shown in **Fig. 13.2**. The code used as input to the software and the output obtained in the form of roof displacement versus base shear values at various steps of push are presented here.

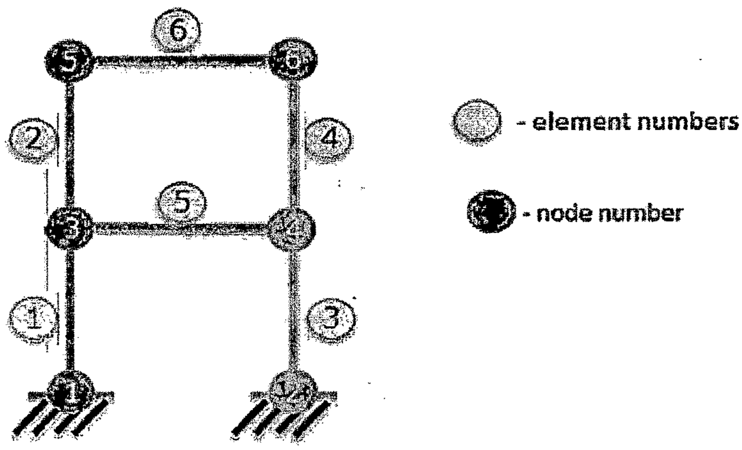


Fig. 13.2 Portal Frame Model Developed in OpenSEES

The following code is used as the input file to solve the problem using OpenSEES.

```
wipe
# units: N/M^2,M,SEC
# create model
model basic -ndm 2 -ndf 3
node 1 0.0 0.0
node 2 3 0.0
node 3 0.0 3
node 4 3 3
node 5 0.0 6
node 6 3 6
fix 1 1 1 1
fix 2 1 1 1
fix 3 0 0 0
fix 4 0 0 0
fix 5 0 0 0
fix 6 0 0 0
```

```
geomTransf Linear 1
element elasticBeamColumn 1 1 3 0.09 22360.0e+6 0.000675 1
element elasticBeamColumn 2 3 5 0.09 22360.0e+6 0.000675 1
element elasticBeamColumn 3 2 4 0.09 22360.0e+6 0.000675 1
element elasticBeamColumn 4 4 6 0.09 22360.0e+6 0.000675 1
element elasticBeamColumn 5 3 4 0.09 22360.0e+6 0.000675 1
element elasticBeamColumn 6 5 6 0.09 22360.0e+6 0.000675 1
recorder Node -file node12.out -time -node 1 2 -dof 1 reaction
recorder Node -file node34.out -time -node 3 4 -dof 1 disp
recorder Node -file node56.out -time -node 5 6 -dof 1 disp
# gravity load analysis
pattern Plain 1 Linear {
load 3 0.0 -4500 -2250
load 4 0.0 -4500  2250
load 5 0.0 -4500 -2250
load 6 0.0 -4500  2250
}
constraints Transformation
numberer RCM
system BandGeneral
test NormDispIncr 1.0e-6 10
algorithm Newton
integrator LoadControl 0.1
analysis Static
analyze 10
loadConst -time 0.0
# pushover analysis
pattern Plain 2 "Linear" {
load 3 72.0 0.0 0.0
```

```
load 4 72.0 0.0 0.0
load 5 288.0 0.0 0.0
load 6 288.0 0.0 0.0
}
integrator DisplacementControl 5 1 0.024
analyze 10
```

The code generates an output in the form of base shear and roof displacement values which are used to construct the capacity curve. The output obtained is presented in **Table 13.1**.

Table 13.1 Output from OpenSEES Analysis

Base shear in kN	Roof Displacement in mm
85.0	24
170.2	48
255.3	72
340.5	96
425.6	112
510.7	144
595.8	168
680.9	192
766.0	216
851.2	240

The capacity curve drawn from the output given in **Table 13.1** is plotted in **Fig. 13.3**. Next, considering the horizontal displacements at the 1^{st} and 2^{nd} storey as the two degrees of freedom, the dynamic matrix $M^{-1}K_s$ is developed and the eigen values representing the natural frequencies and eigen vectors corresponding to mode shapes are calculated. The mass considered at both storey level is 917 kg.

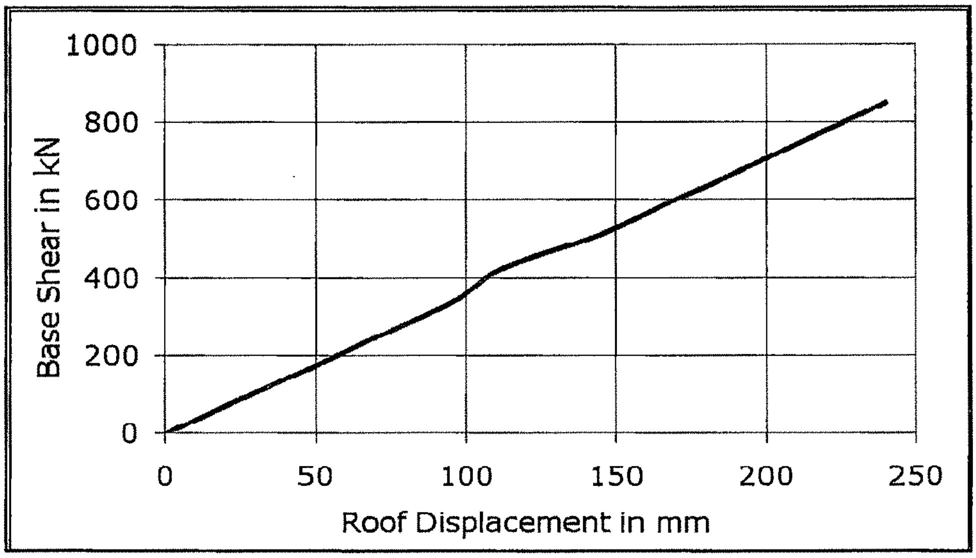


Fig. 13.3 Capacity Curve from the Output of OpenSEES

The normalized eigenvalues are ${}_1\Phi^1$ =0.0211 & ${}_1\Phi^2$ =0.00996. **Table 13.2** presents the modal calculations in standard notations.

Table 13.2 Modal Analysis Calculations for the Portal Frame

Level	W in N	W/g	Φ1	W/g x Φ1	W/g x Φ1 x Φ1
roof	18000	1834.86	0.02110	38.72	0.8169
1	18000	1834.86	0.00996	18.28	0.1820
sum		3669.72		56.99	1.0000

Referring to ATC 40 [1] the modal participation factor for the first natural mode is calculated using PF1 and α1 as follows:

$$PF_1 = \left[\frac{\sum_{i=1}^{N} (w_i \phi_{i1}) / g}{\sum_{i=1}^{N} (w_i \phi_{i1}^2) / g} \right] \qquad \alpha_1 = \frac{\left[\sum_{i=1}^{N} (w_i \phi_{i1}) / g \right]^2}{\left[\sum_{i=1}^{N} w_i / g \right] \left[\sum_{i=1}^{N} (w_i \phi_{i1}^2) / g \right]}$$

PF1 = 56.99 * 0.0211/1.0 = 1.202, $\alpha_1 = 56.99^2/(3669.72*1.0) = 0.885$

Using these values, Spectral acceleration Sa and Spectral displacement Sd are evaluated as given in **Table 13.3**.

Table 13.3 Capacity Spectrum from Capacity Curve in ADRS format

Base Shear V (kN)	Roof Displacement Δroof (mm)	Modal Participation Factor PF1	Modal Mass Coefficient α1	Weight Assigned to Level 1 W (kN)	Spectral Acceleration Sa/g = V/(W*α) (g)	Spectral Displacement Sd = Δroof/PF1 (mm)
0	0	1	1	18	0	0
85	24	1.202	0.885	18	5.34	19.97
170	48	1.202	0.885	18	10.68	39.93
255	72	1.202	0.885	18	16.03	59.90
341	96	1.202	0.885	18	21.37	79.87
426	112	1.202	0.885	18	26.72	93.18
511	144	1.202	0.885	18	32.06	119.80
596	168	1.202	0.885	18	37.40	139.77
681	192	1.202	0.885	18	42.74	159.73
766	216	1.202	0.885	18	48.09	179.70
851	240	1.202	0.885	18	53.43	199.67

The values calculated in **Table 13.3** are plotted to construct the capacity spectrum in ADRS format for the portal frame as shown in **Fig. 13.4**.

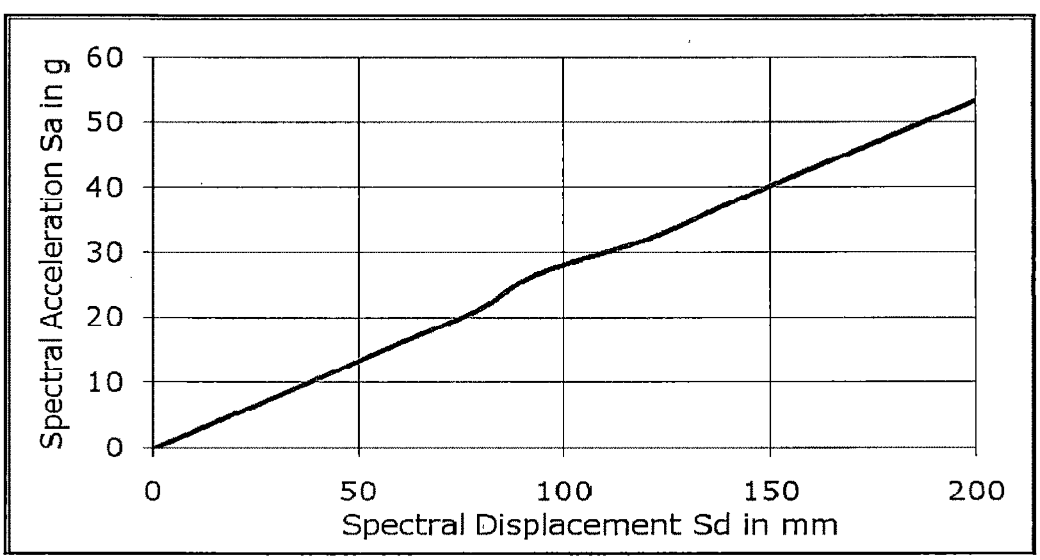


Fig. 13.4 Capacity Spectrum in ADRS Format for the Portal Frame

13.2.3 Development of the Demand Spectrum

Value of seismic coefficient C_A should be taken to be equal to 0.4 times the spectral response acceleration (units of g) at a period of 0.3 seconds i.e. effective peak acceleration (EPA). A factor of about 2.5 times C_A represents the average value of a 5 % damped short period system in the acceleration domain. The seismic coefficient C_V represents 5 % damped response of a 1-second system and when divided by period, it defines response in the velocity domain. **Figure 13.5** illustrates the construction of an elastic response spectrum for 5% damping (ATC-40 [1]).

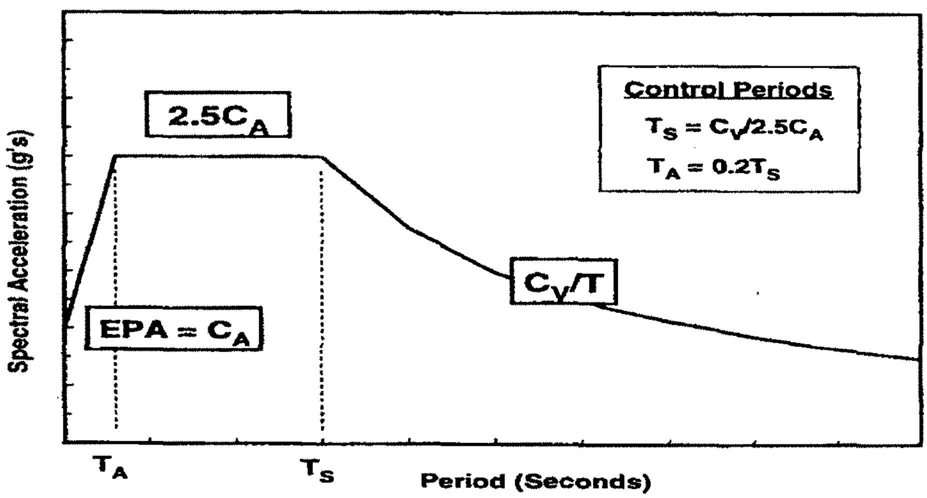


Fig. 13.5 Construction of a 5% Damped Elastic Response Spectrum

For medium soil with seismic zone factor z=0.16, as per proposed draft provisions and commentary on Indian seismic code IS:1893, Part 1, 2002 **[24]**, equivalent seismic coefficient C_A as per the code is given by,

$$C_A = Z * g * Sa/g \text{ (at EPA)} \qquad .....\text{Eq. 13.1}$$

Therefore, $C_A = 0.16 * 2.5 = 0.4$

$$C_V = 2.5 * C_A * T_S \qquad .....\text{Eq. 13.2}$$

Therefore, $C_V = 2.5 * 0.4 * 0.55 = 0.55$

To convert the traditional spectrum (Sa versus T format) into ADRS spectrum (Sa versus Sd format), **Eq. 4.9** is to be used. Thus, the demand spectrum can be developed from the values tabulated in **Table 13.4** as shown in **Fig. 13.6**.

Table 13.4 5% Damped Demand Spectrum Values

Time Period T in sec	Spectral Acceleration Sa in g's	Spectral Displacement Sd in mm
0.00	0.40	0.00
0.08	1.00	1.59
0.40	1.00	39.80
0.50	1.00	68.40
1.00	0.55	136.81
1.50	0.37	205.21
2.00	0.28	273.62
2.50	0.22	342.02
3.00	0.18	410.42
3.50	0.16	478.83
4.00	0.14	547.23

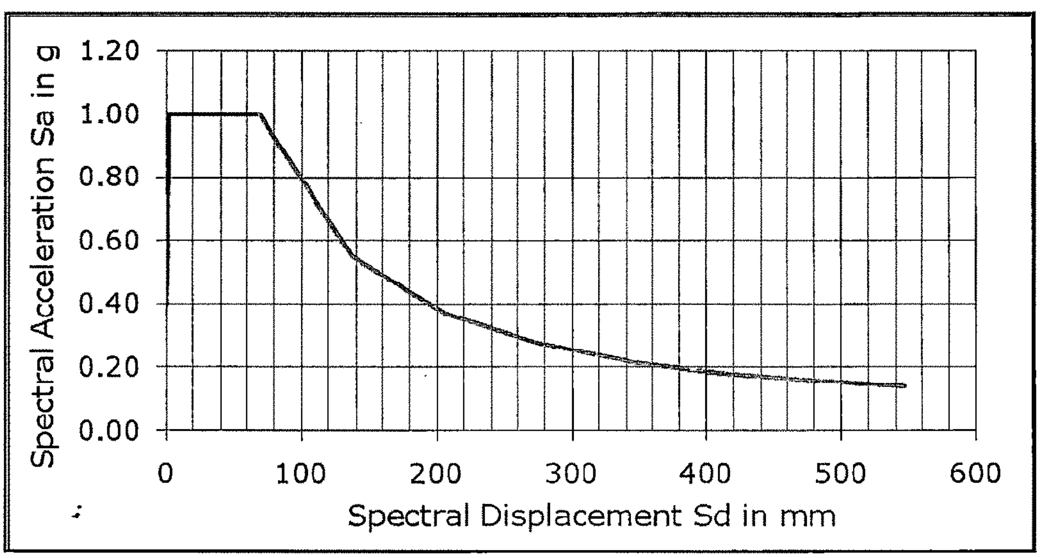


Fig. 13.6 5% Damped Elastic Response Spectrum for the Frame

13.2.4 Locating the Performance Point

Performance point of one bay two storey portal frame is the intersection point of the capacity spectrum with appropriate demand spectrum. The superimposed plot of Capacity Spectrum and Demand Spectrum in ADRS format is shown in **Fig. 13.7**.

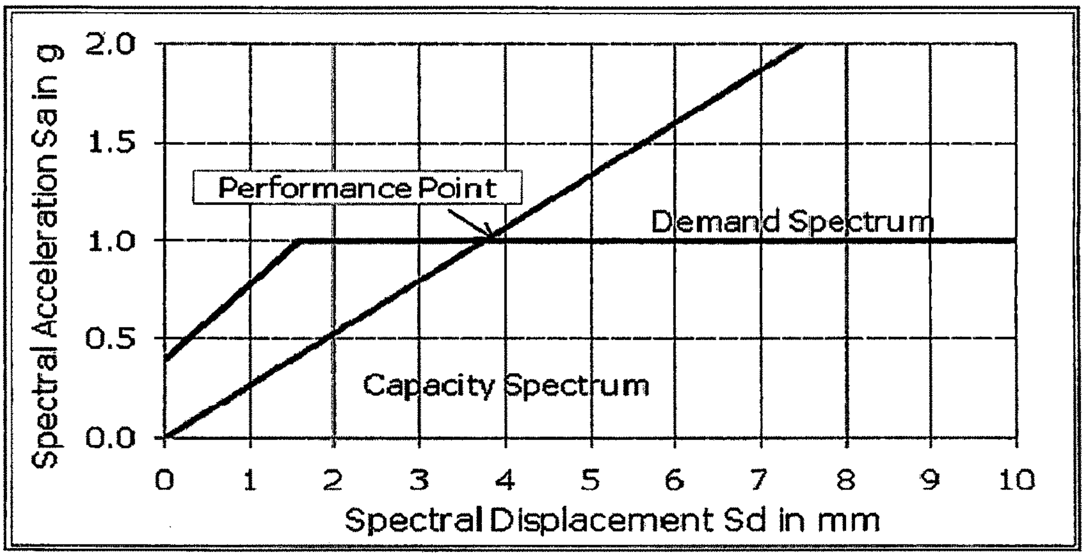


Fig. 13.7 Superimposed Demand and Capacity Spectrum

From the **Fig. 13.7** at performance point, spectral acceleration Sa = 1 in g's and spectral displacement Sd = 3.75 mm. So, using **Eqs. 4.14** and **4.15**, base shear V and roof displacement Δ_{roof} at performance point can be calculated as follows :

Base Shear V = Sa*W*α = 1*18*0.885 = 15.93 kN

Roof Displacement Δ_{roof} = Sd*PF1 =3.75*1.202 = 4.51 mm

13.3 COMPARISON OF RESULTS – OPENSEES AND ETABS

13.3.1 For 1 Bay 2 Storey Plane Frame

The results obtained from OpenSEES and ETABS for the capacity curve are compared for the two storey one bay plane frame solved as a verification problem. The result comparison is presented in **Table 13.5** and plotted in

Fig. 13.8 to view the same graphically. The close agreement between both the results can be clearly seen in the plot. Moreover, the results at performance point worked out by manual calculations is compared with those from ETABS software and presented in **Table 13.6**.

Table 13.5 Comparison of Push over Results for Plane Frame

Roof displacement Δ_{roof} in mm	OpenSEES	ETABS
	Base shear V in kN	Base shear V in kN
0	0.00	0.00
24	85.00	82.93
48	170.20	165.85
72	255.30	248.78
96	340.50	331.71
120	425.60	414.63
144	510.70	497.56
168	595.80	580.48
192	680.90	663.41
216	766.00	746.33
240	851.20	829.26

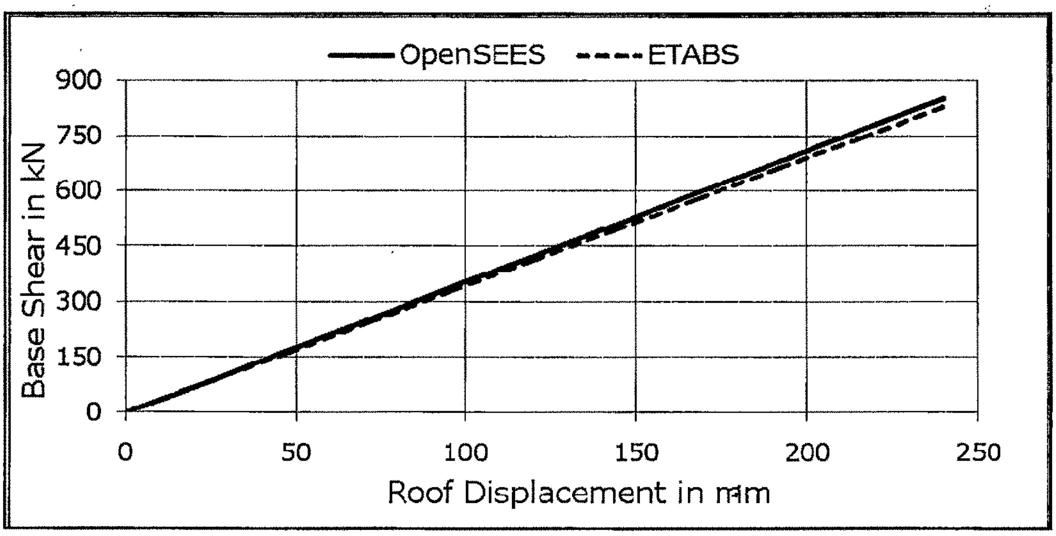


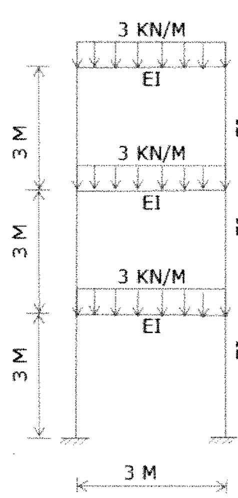
Fig. 13.8 Capacity Curve for Plane Frame by OpenSEES and ETABS

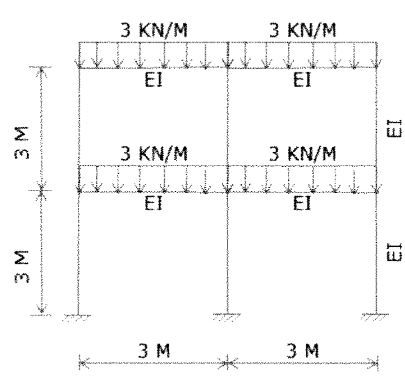
Table 13.6 Performance Point Parameters Comparison

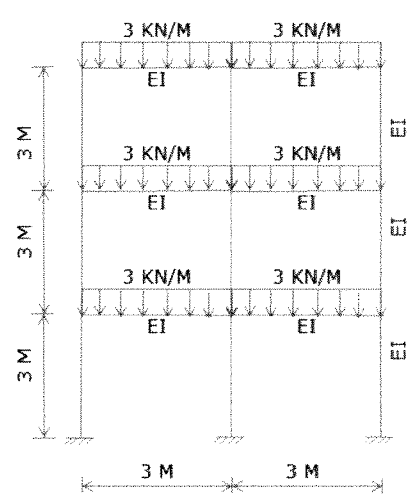
Parameter	OpenSEES	ETABS	% Difference
Spectral acceleration Sa in g's	1.00	0.998	-0.20
Spectral displacement Sd in mm	3.75	3.801	1.34
Base shear V in kN	15.93	15.830	-0.63
Roof displacement Δ_{roof} in mm	4.51	4.580	1.53

13.3.2 Results for Three More Plane Frames

Comparisons of results obtained for 3 more models of plane frames with similar geometric and loading conditions are made. The models considered are shown in **Fig. 13.9**. They consist of one bay three storey, two bay two storey and two bay three storey plane frames. The results of roof displacement and base shear obtained by push over analysis using OpenSEES and ETABS are presented in **Tables 13.7** thru **13.9** and the corresponding plots in **Figs. 13.10** to **13.12**.







a)1 bay 3 storey **b) 2 bay 2 storey** **c) 2 bay 3 storey**

Fig. 13.9 Plane Frame Models Considered for Push Over Analysis

Table 13.7 Comparison of Results for 1 Bay 3 Storey Frame

Roof displacement Δ_{roof} in mm	Base shear V in kN	
	OPENSEES	ETABS
0	0.0	0.0
36	76.5	74.5
72	152.9	149.0
108	229.3	223.5
144	305.8	298.0
180	382.2	372.5
216	458.7	447.0
252	535.1	521.4
288	611.6	596.0
324	688.0	670.4
360	764.5	745.0

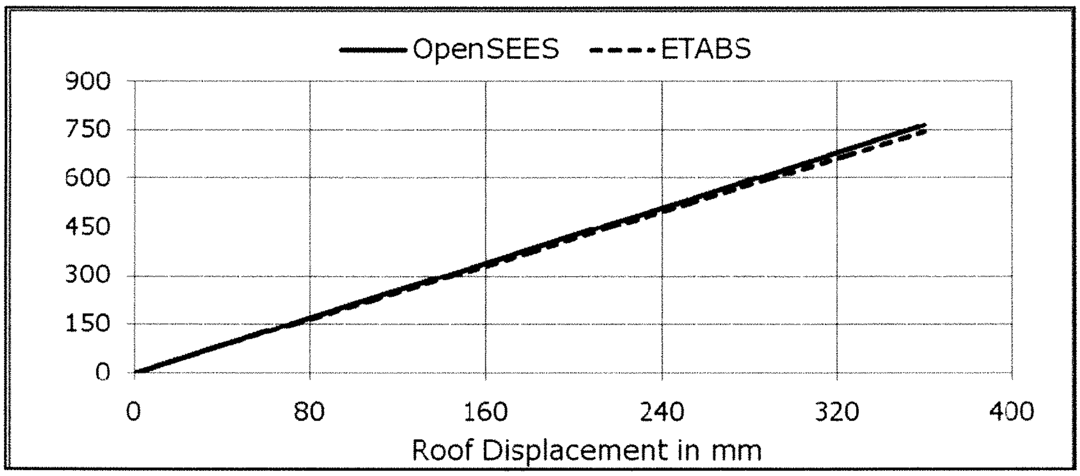


Fig. 13.10 Capacity Curve for One Bay Three Storey Frame

Table 13.8 Comparison of Results for 2 Bay 2 Storey Frame

Roof displacement Δ_{roof} in mm	Base shear V in kN	
	OPENSEES	ETABS
0	0.0	0.0
24	141.1	137.4
36	282.2	274.8
72	423.3	412.2
96	564.4	549.7
120	705.5	687.1
144	846.6	824.5
168	987.7	961.9
192	1128.8	1099.3
216	1269.9	1236.7
240	1411.0	1374.1

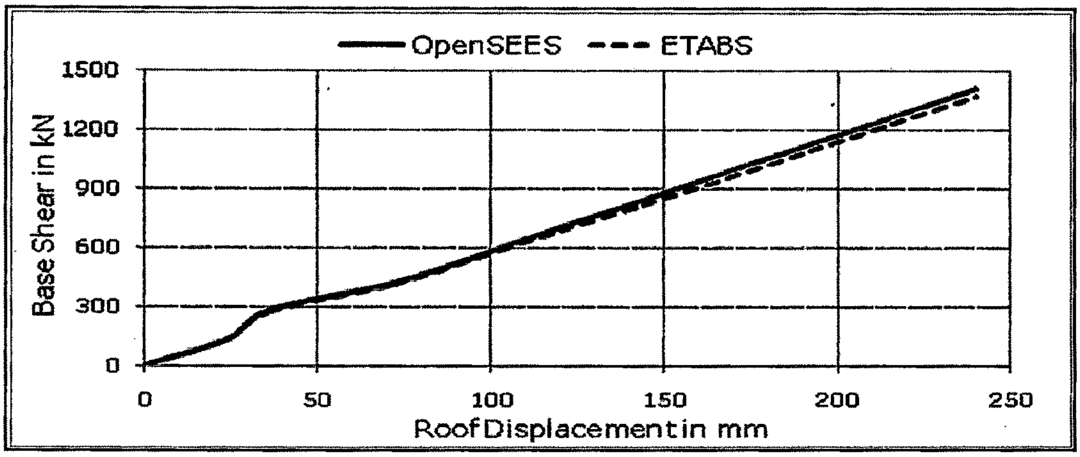


Fig. 13.11 Capacity Curve for Two Bay Two Storey Frame

Table 13.9 Comparison of Results for 2 bay 3 storey frame

Roof displacement Δ_{roof} in mm	**Base shear V in kN**	
	OPENSEES	**ETABS**
0	0.0	0.0
36	143.0	139.3
72	286.1	278.6
108	429.1	417.9
144	572.2	557.2
180	715.2	696.5
216	858.2	835.7
252	1001.3	975.0
288	1144.3	1114.3
324	1287.4	1253.6
360	1430.4	1392.9

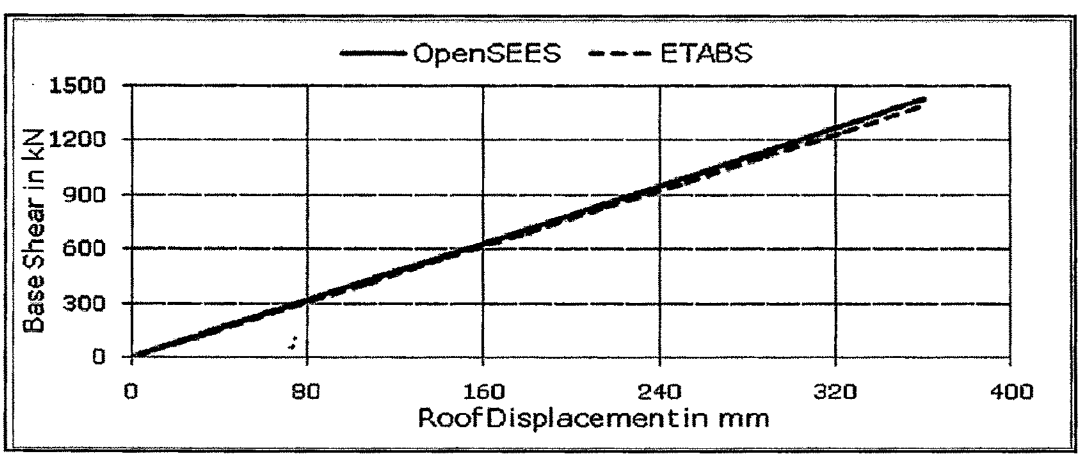


Fig. 13.12 Capacity Curve for Two Bay Three Storey Frame

13.4 COMMENTS ON RESULTS

In order to establish the confidence in any commercial software like ETABS, it has to be verified with some verification problems. In the present chapter, this verification was done for the results of static push over analysis with the help of an open source earthquake engineering simulation software – OpenSEES. It was done in two steps. First, the push over analysis was carried out for a one bay two storey plane frame by OpenSEES and ETABS. The result in the form of the capacity curve obtained from both the softwares was presented in **Fig. 13.8**. The close agreement of the results is seen in the plot. Further, for the same problem, manual calculations were carried out to obtain the performance point from the OpenSEES results. The performance point values evaluated by both OpenSEES and ETABS were presented in **Table 13.6**. The results for performance point show a maximum difference of 1.5% which can be considered acceptable and in good agreement.

The further comparison of the push over analysis results for 1 bay 3 storey, 2 bay 2 storey and 2 bay 3 storey frames were presented in **Figs. 13.10**, **13.11** and **13.12** respectively. The plots once more show a good agreement between the results of the push over analysis by OpenSEES and ETABS. This fact is helpful in reinforcing the confidence in the work carried out using the commercial software ETABS. The maximum difference in the base shear values for all the mathematical models considered is found as 2.63%.