MIDDLE EAST TECHNICAL UNIVERSITY

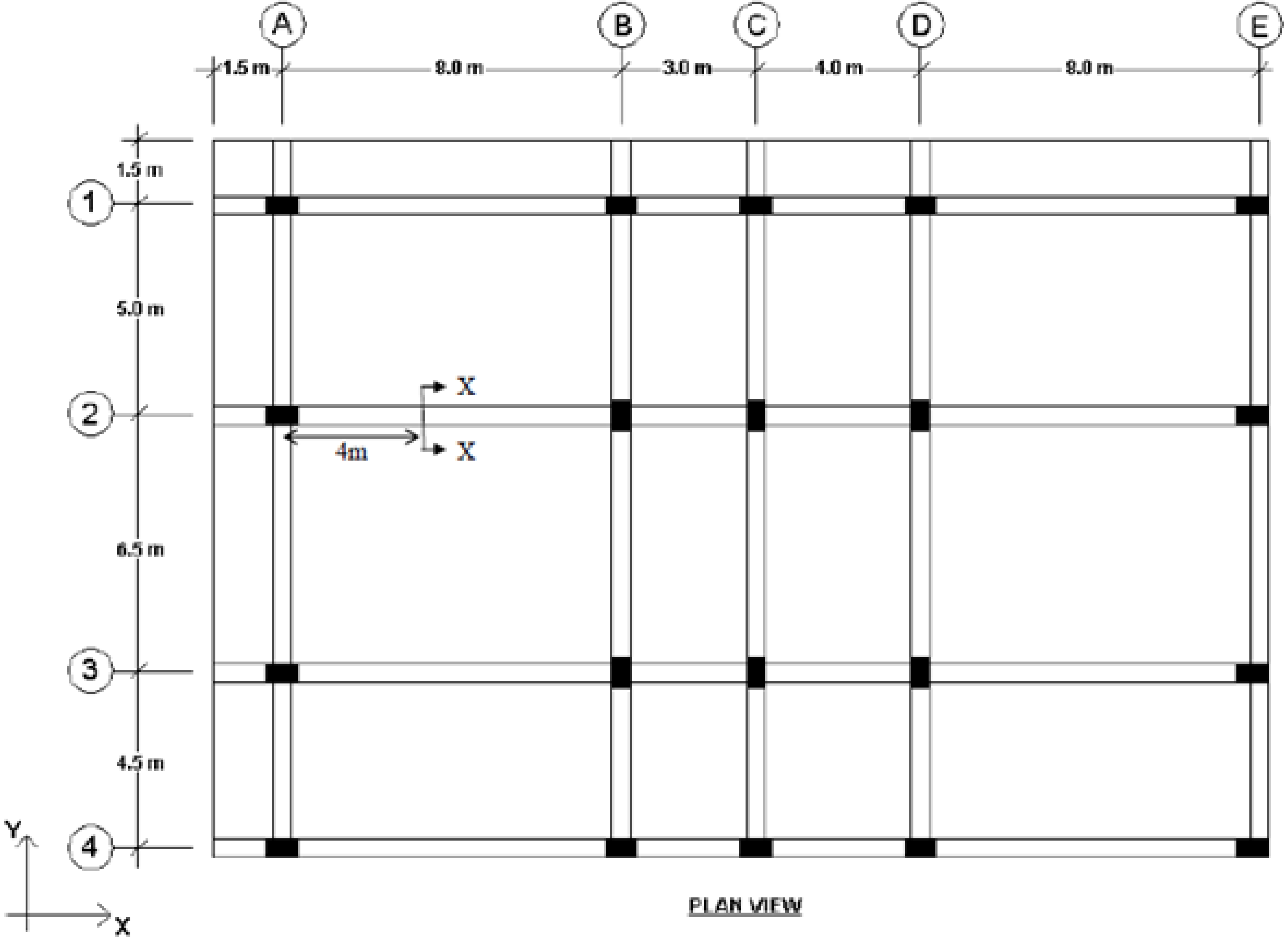
CE486 STRUCTURAL DESIGN CONCRETE STRUCTURES

SAP2000 TUTORIAL

OUTLINE

Create a 3D model of the given building and do the gravity and dynamic analyses.

- Set the unit system
- Form a grid.
- Define the material properties
- Define the section properties (cracked section)
- Draw the geometry (for one floor)
- Define Rigid End Zones
- Define the loads (load patterns → load cases → load combinations)
- Replicate
- Define the boundary conditions
- Define rigid diaphragms at floor levels
- Run the analysis
- Define the mass source (tie it to the diaphragm)
- Define the response spectrum
- Define the dynamic load case
- Run the dynamic analysis



PLAN VIEW

The plan of a 7-story reinforced concrete residential building is shown below.

The floor-to-floor height is 3.5 m.

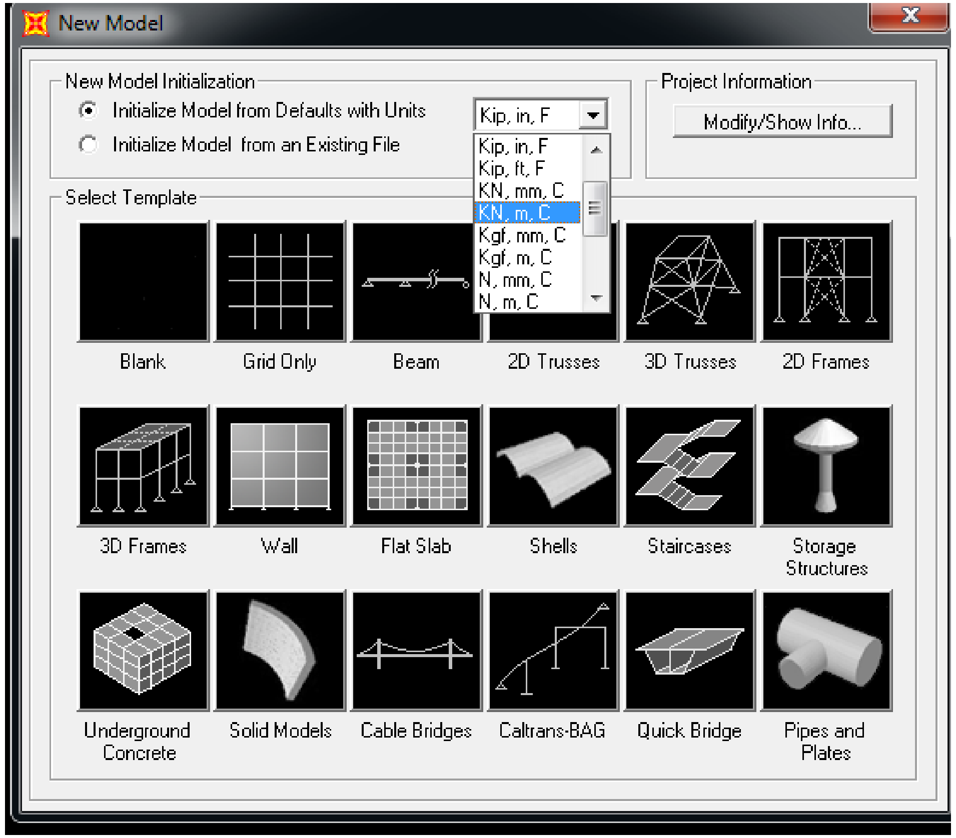
C30 class concrete will be used in the structural elements.

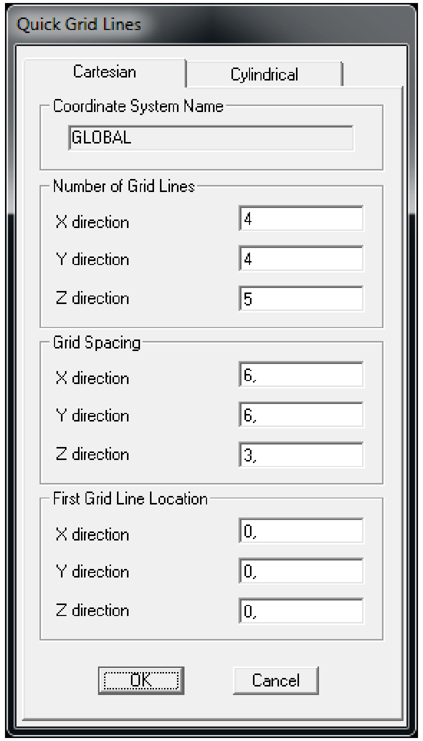
The dead load including the self-weight on beams on grid lines A, C and E is 15 kN/m and on grid lines B and D is 30 kN/m. Likewise, the dead load on beams on grid lines 1 and 4 is 15 kN/m and on grid line 2 and 3 is 30 kN/m.

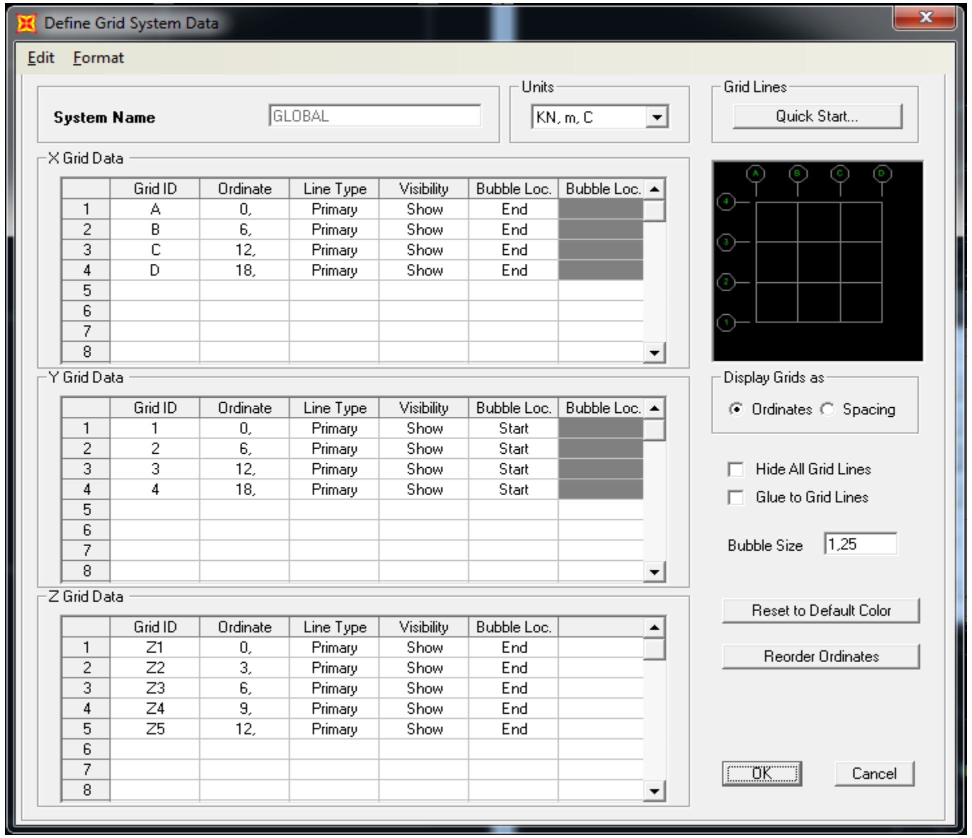
Assume that ***the live load*** on all beams is equal to 12 kN/m.

The beam and columns are rectangular and the dimensions are 50 cm x 60 cm and 50 cm x 70 cm, respectively.

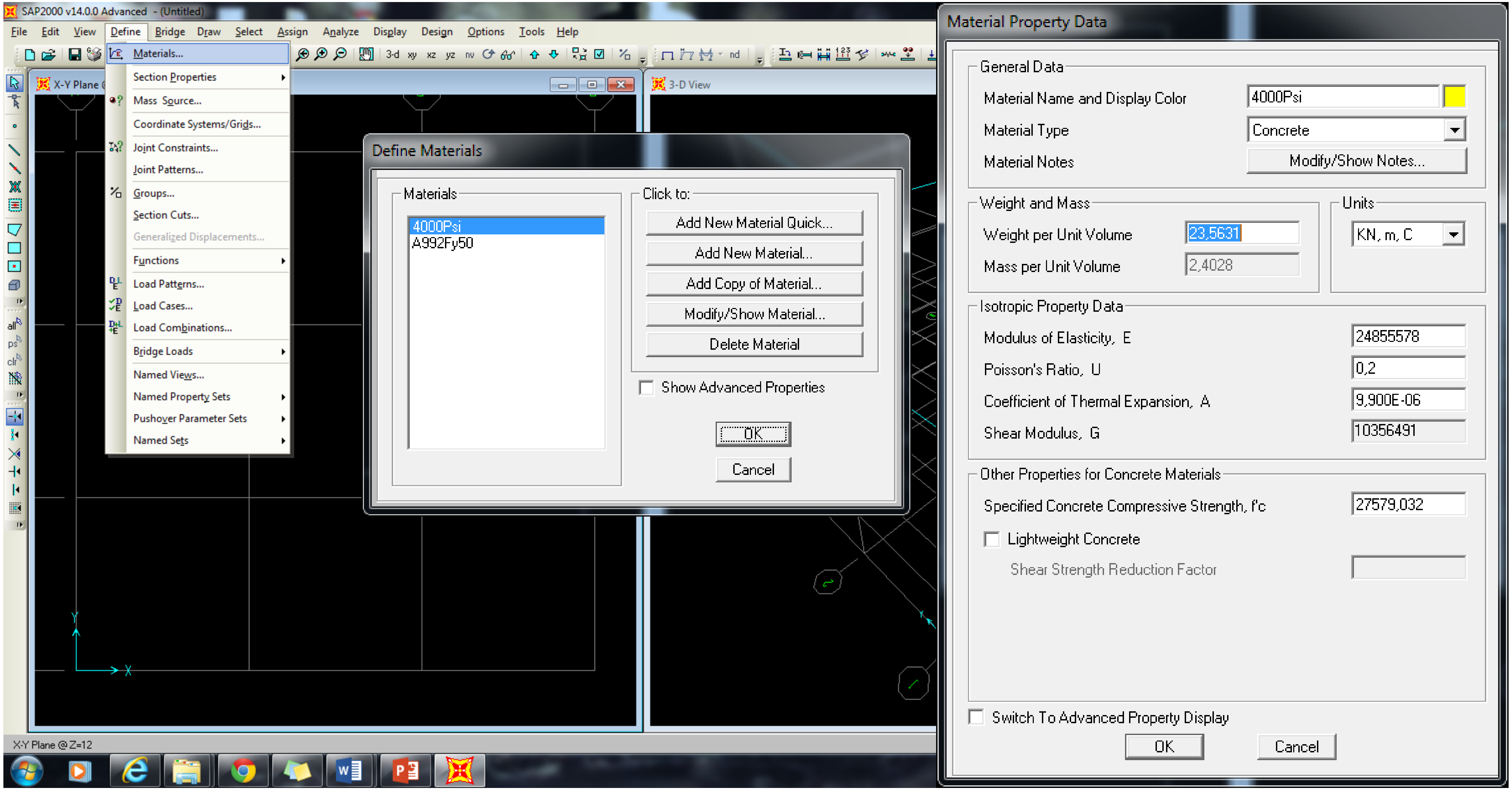
Set the unit system and Form the grids







Define the material properties



TS500

ÇİZELGE 3.2 - Beton Sınıfları ve Dayanımları

Beton Sınıfı	Karakteristik Basınç Dayanımı, f_{ck} MPa	Eşdeğer Küp (200 mm) Basınç Dayanımı MPa	Karakteristik Eksenel Çekme Dayanımı, f_{ctk} MPa	28 Günlük Elastisite Modülü, E_c MPa
C16	16	20	1,4	27 000
C18	18	22	1,5	27 500
C20	20	25	1,6	28 000
C25	25	30	1,8	30 000
C30	30	37	1,9	32 000
C35	35	45	2,1	33 000
C40	40	50	2,2	34 000
C45	45	55	2,3	36 000
C50	50	60	2,5	37 000

$$E_{cj} = 3250\sqrt{f_{ckj}} + 14000 \quad \text{(MPa)}$$

TEC 2007 Cracked Section Definition

7.4.13 – Eğilme etkisindeki betonarme elemanlarda çatlamış kesite ait *etkin eğilme rijitlikleri* $(EI)_e$ kullanılacaktır. Daha kesin bir hesap yapılmadıkça, etkin eğilme rijitlikleri için aşağıda verilen değerler kullanılacaktır:

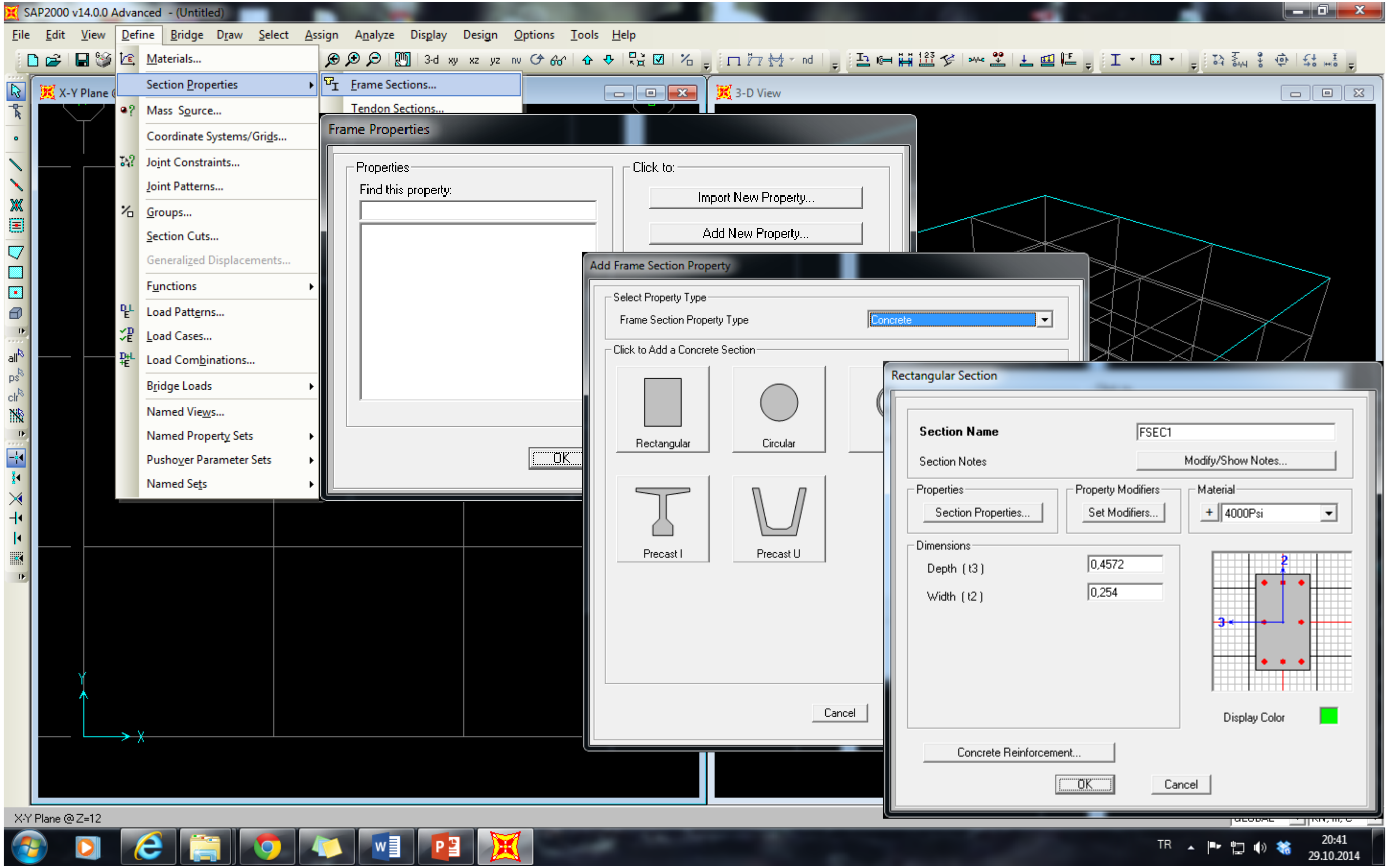
(a) Kirişlerde: $(EI)_e = 0.40\ (EI)_o$

(b) Kolon ve perdelerde, $N_D\ /\ (A_c f_{cm}) \leq 0.10$ olması durumunda: $(EI)_e = 0.40\ (EI)_o$

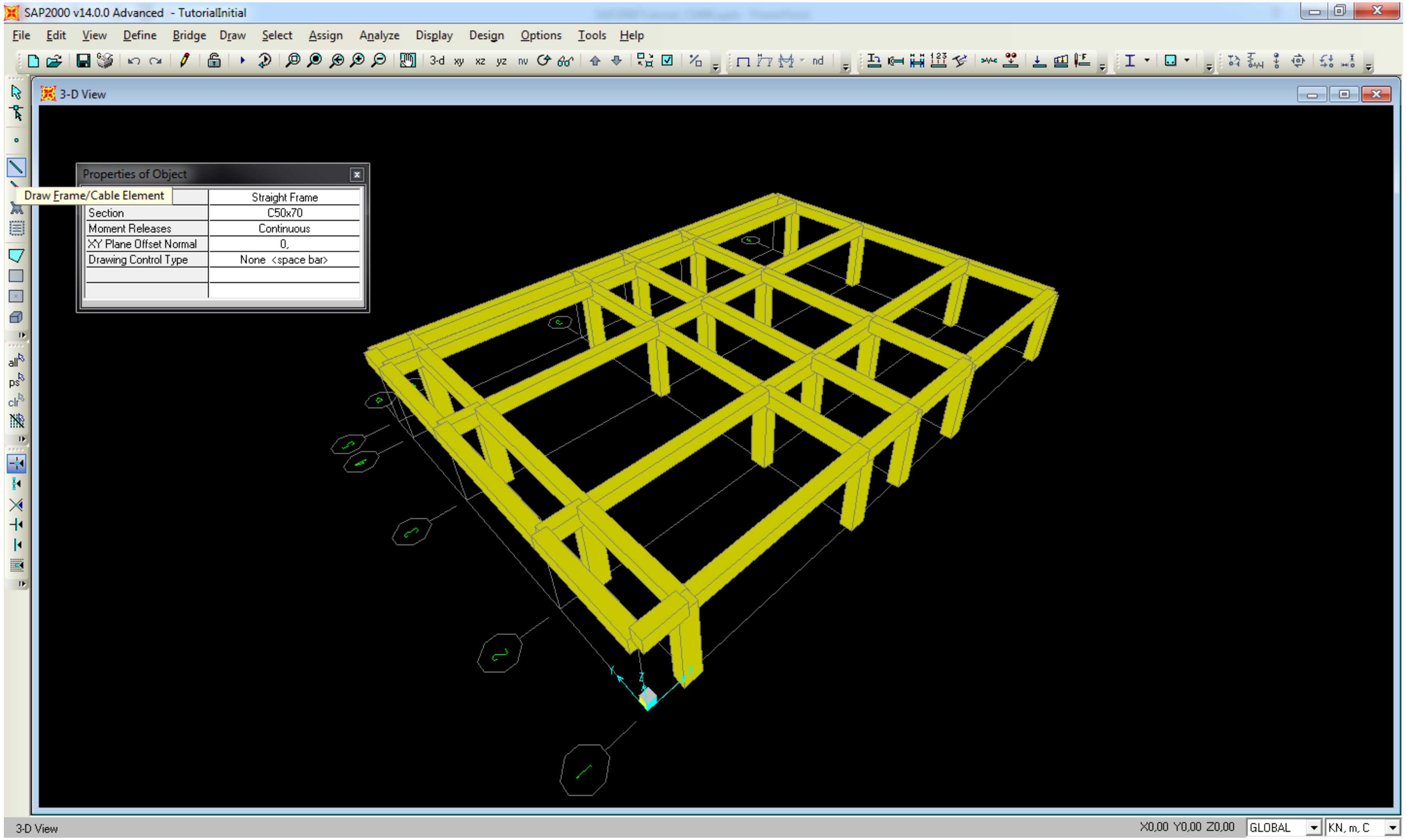
$N_D\ /\ (A_c f_{cm}) \geq 0.40$ olması durumunda:. $(EI)_e = 0.80\ (EI)_o$

Eksenel basınç kuvveti N_D'nin ara değerleri için doğrusal enterpolasyon yapılabilir. N_D, deprem hesabında esas alınan toplam kütlelerle uyumlu yüklerin gözönüne alındığı ve çatlamamış kesitlere ait $(EI)_o$ eğilme rijitliklerinin kullanıldığı bir ön düşey yük hesabı ile belirlenecektir. Deprem hesabı için başlangıç durumunu oluşturan düşey yük hesabı ise, yukarıda belirtildiği şekilde elde edilen etkin eğilme rijitliği $(EI)_e$ kullanılarak, deprem hesabında esas alınan kütlelerle uyumlu yüklere göre yeniden yapılacaktır. Deprem hesabında da aynı rijitlikler kullanılacaktır.

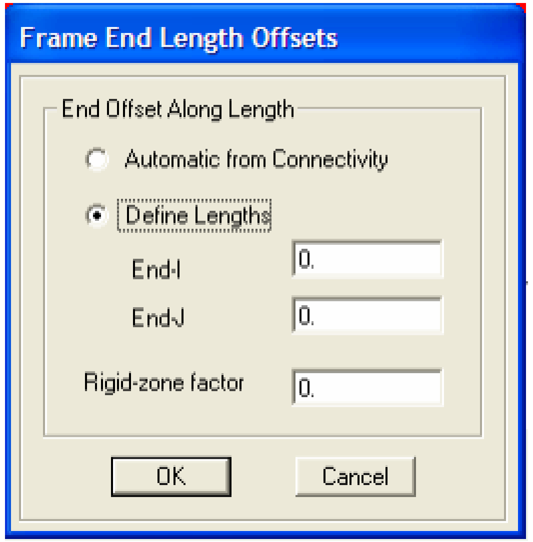
Define the section properties (cracked section)



Draw the geometry (for one floor)



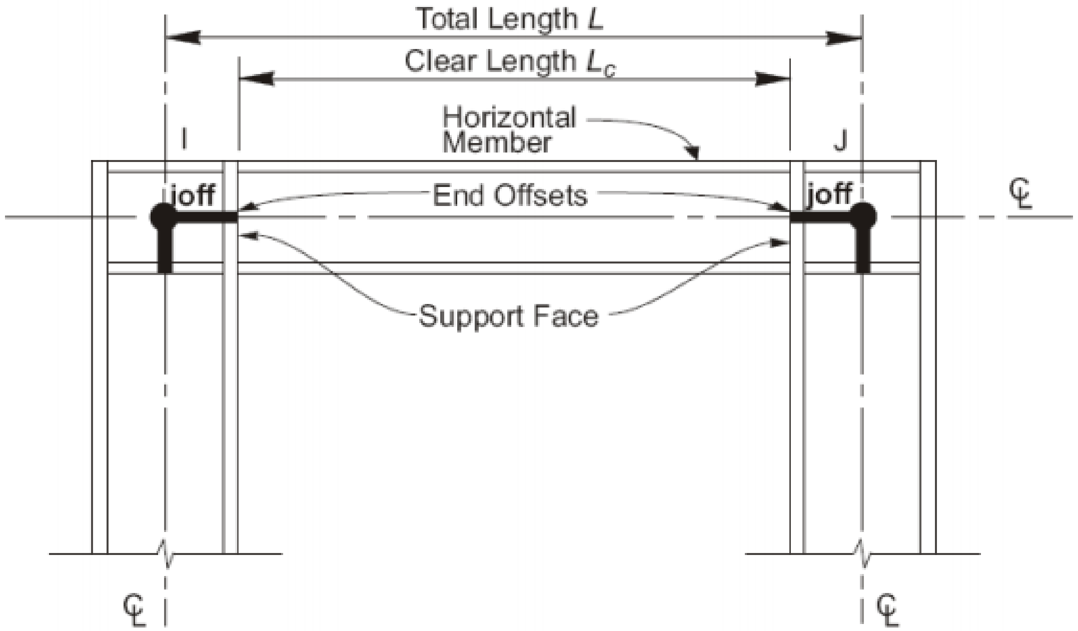
Define Rigid End Zones



<u>Automatic from connectivity:</u> Choose this option to have SAP2000 calculate the end offsets from the connectivity of the model. The program will automatically calculate the end offsets from the Depth (Major) and Width (Minor) specified for the frame section properties.

<u>Define Lengths:</u> Type values for the offsets at End I and End J

<u>Rigid Zone Factor:</u> This is a factor used to define the percentage of the zone specified through end offsets to be taken as fully rigid. **0 means no rigid zone and 1 means that the entire zone is taken as rigid.**



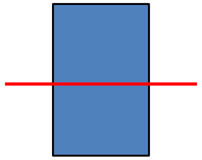
FRAME ELEMENTS IN SAP2000

Real Structural Member :

Beam / Column

Centroidal axis

Centroidal axis



Mathematical Idealization :

Frame Element

=

=

Axial Deformation

Area

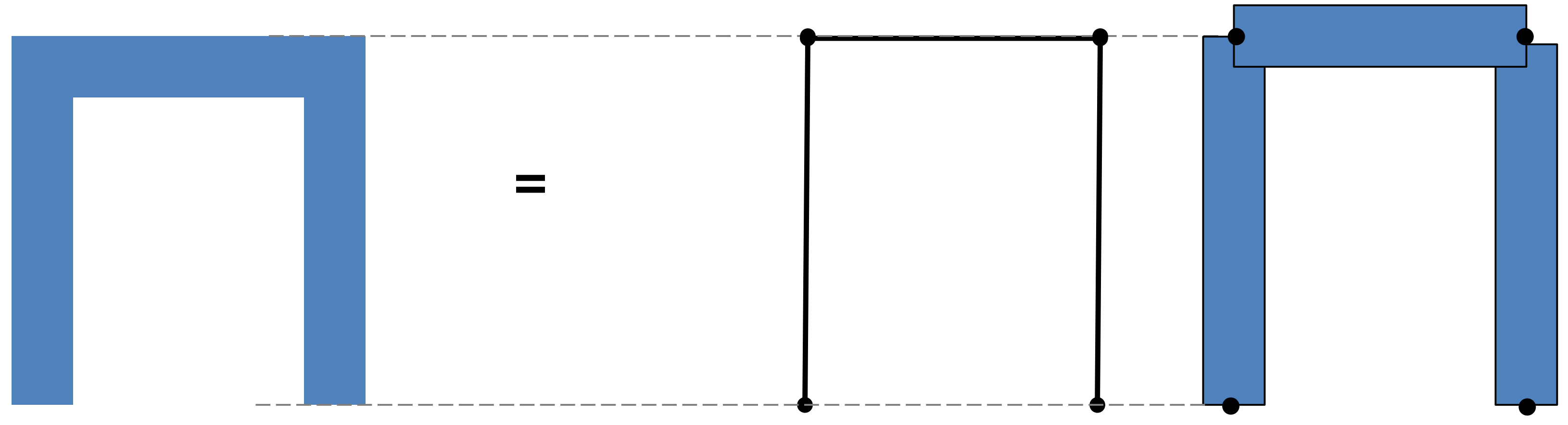
+

Bending Deformation

Moment of Inertia

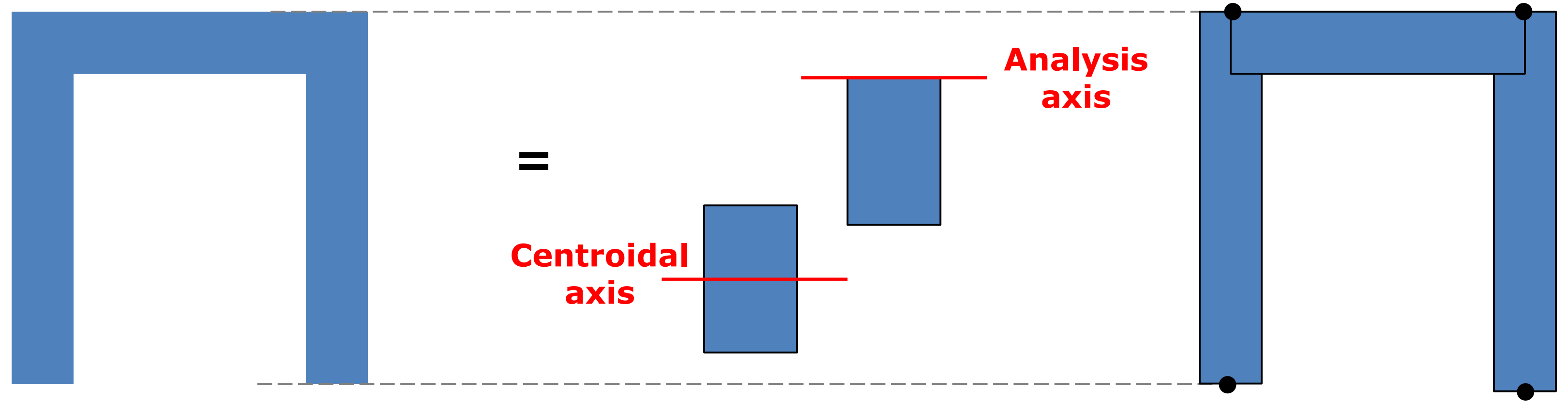
Real Structural Member : **Mathematical Idealization:**

I

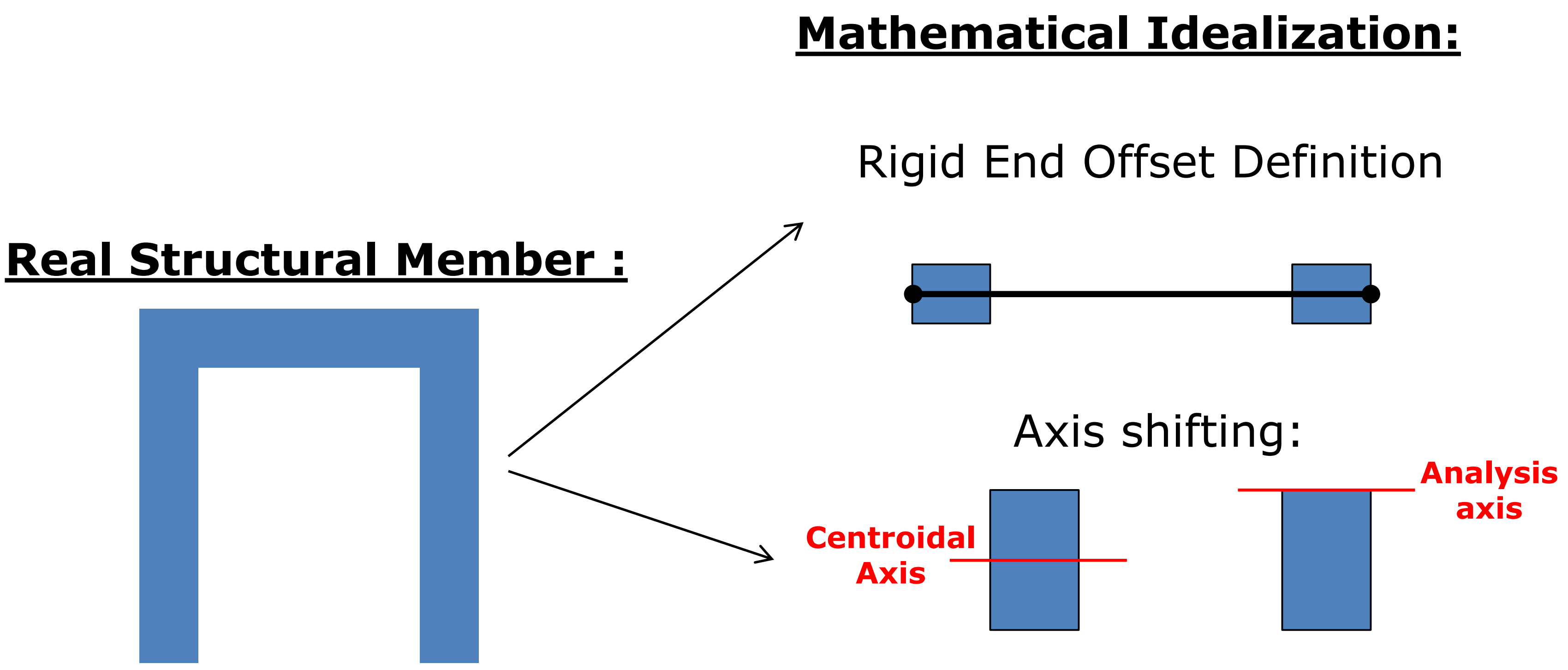


Axis Shifting:

II

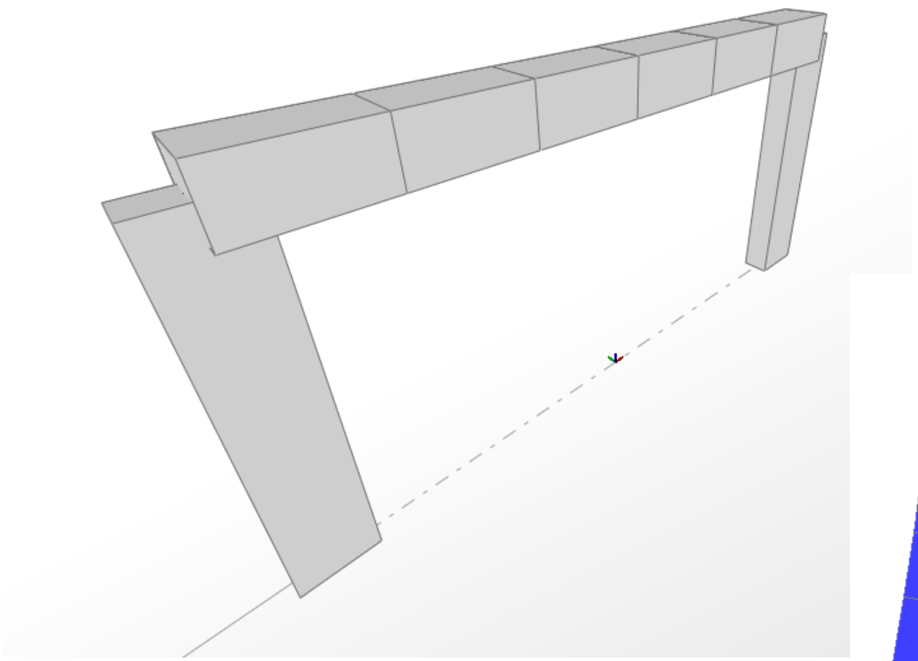


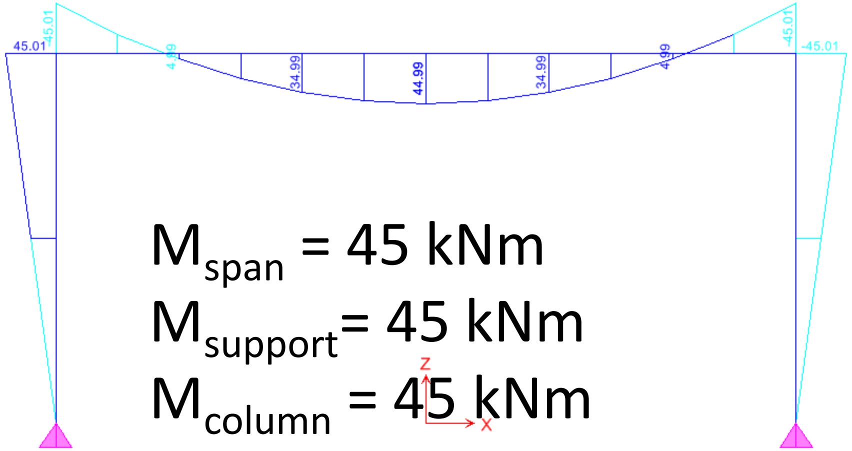
One Dimensional Modelling Alternatives



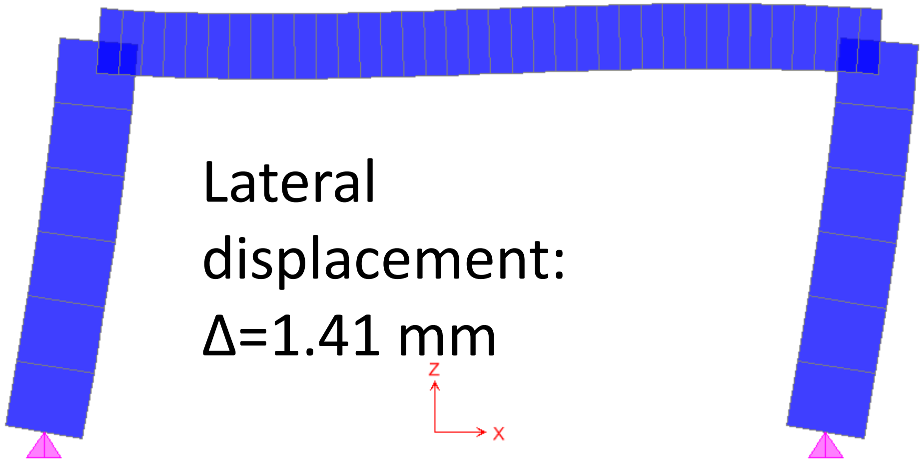
Example Frame
Centroidal Axis Modelling without Rigid End Offsets:

- Beam: 50 x 50 cm
- Columns : 30 x 60 cm



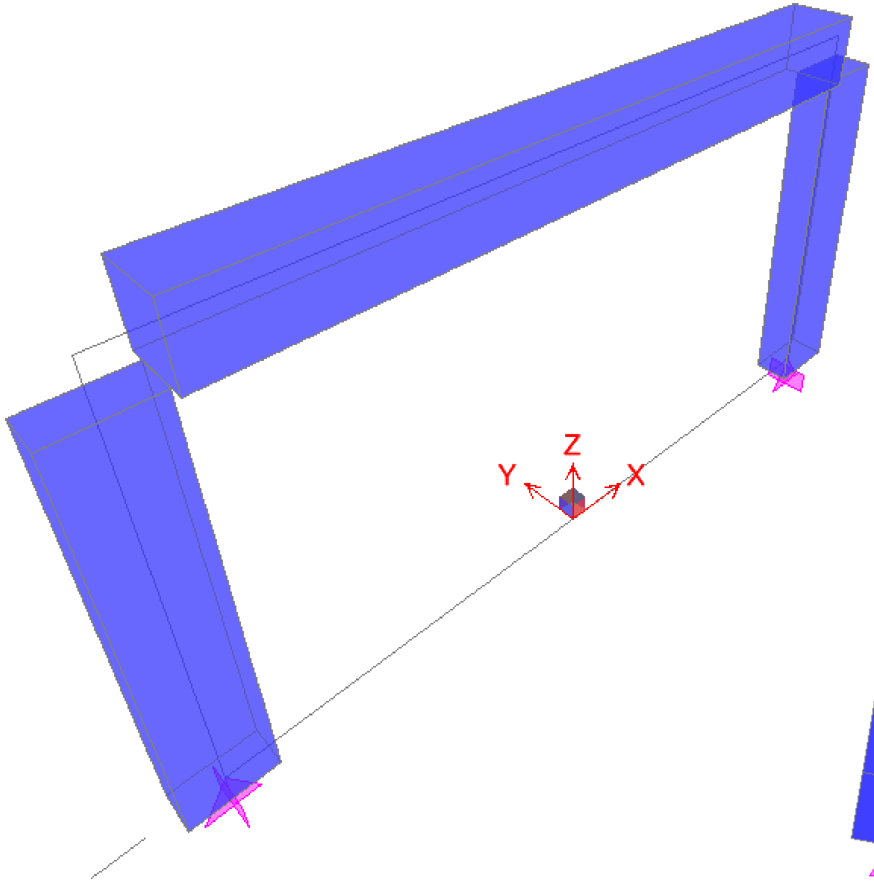


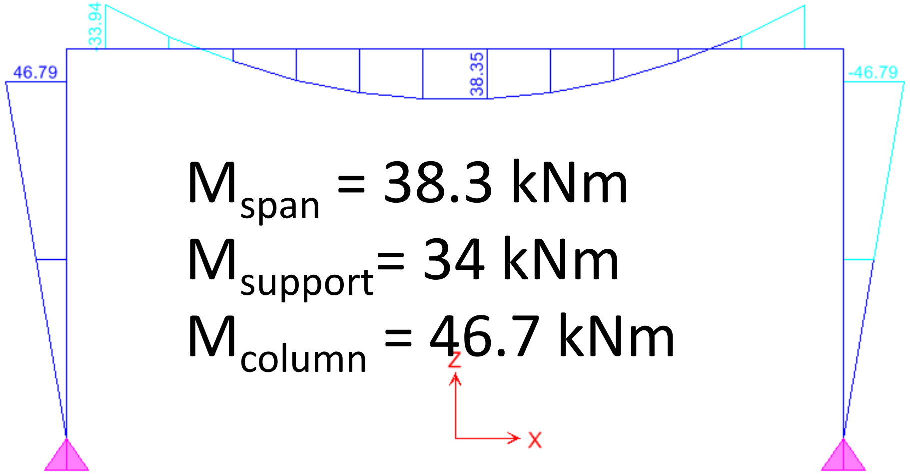
M_{span} = 45 kNm
$M_{support}$= 45 kNm
M_{column} = 45 kNm



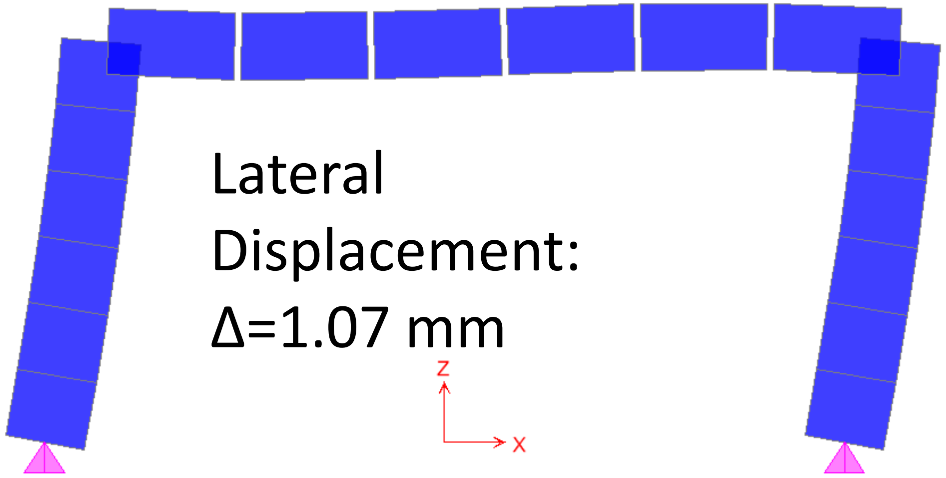
Lateral displacement:
Δ=1.41 mm

Centroidal Axis Modelling with Rigid End Offsets:





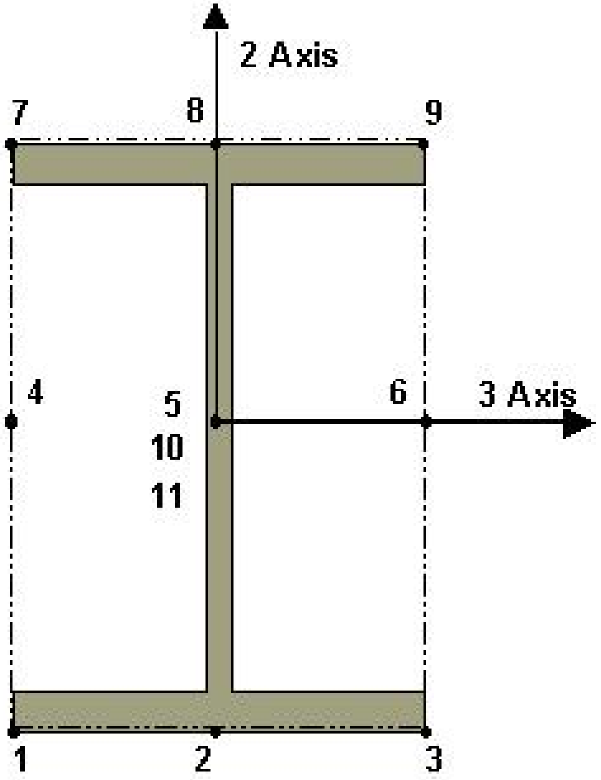
M_{span} = 38.3 kNm
$M_{support}$= 34 kNm
M_{column} = 46.7 kNm



Lateral Displacement:
Δ=1.07 mm

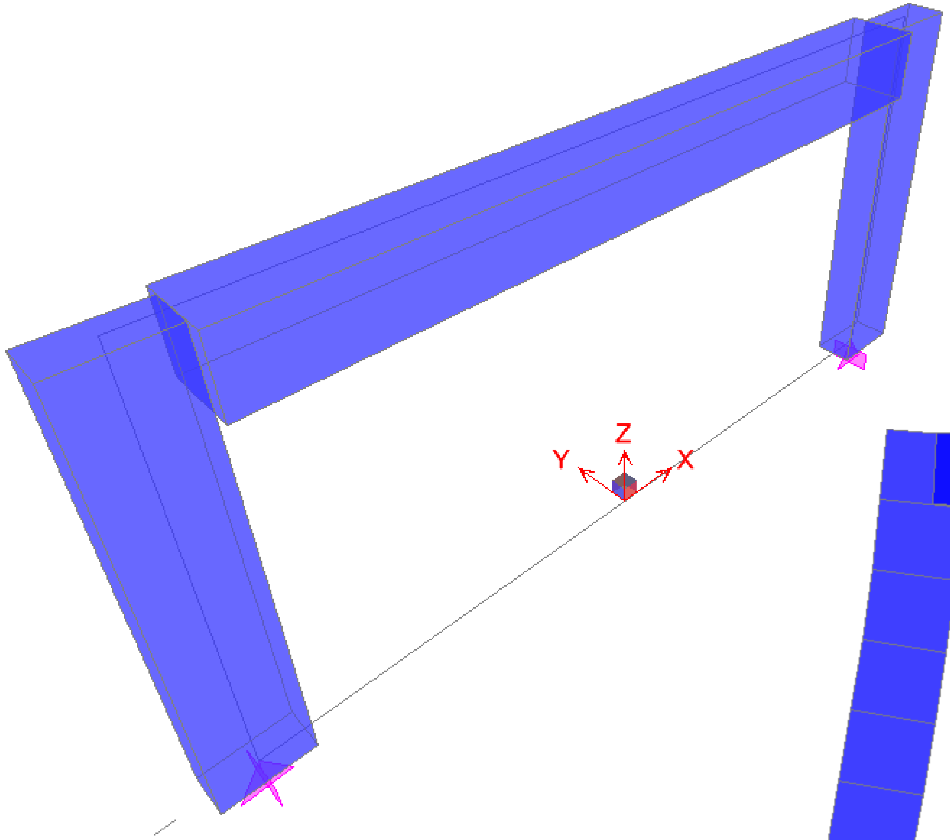
Axis Shifting
Frame Insertion Point

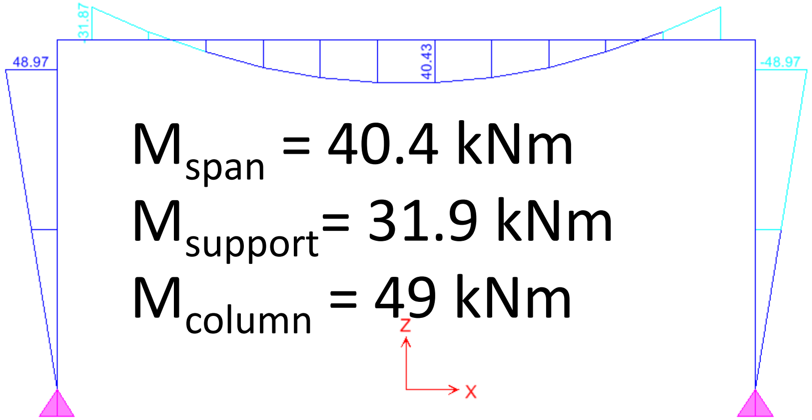
Cardinal Point:



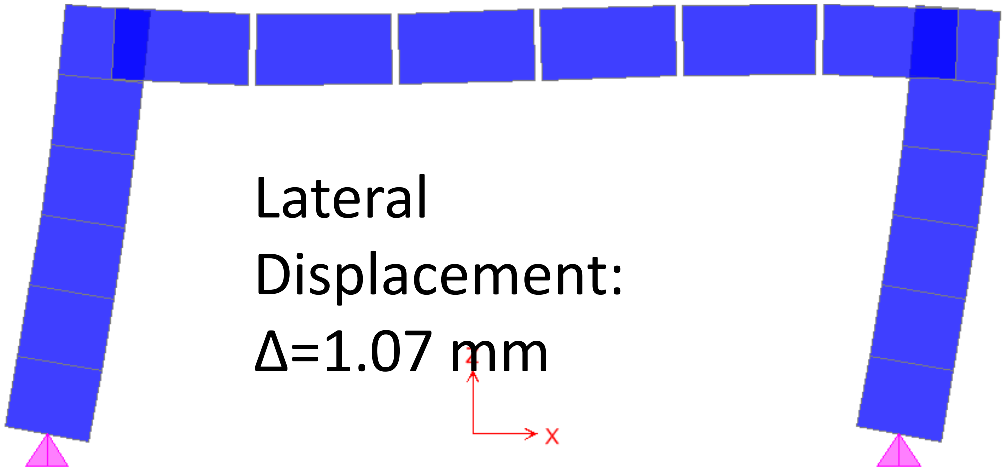
- Bottom left
- Bottom center
- Bottom right
- Middle left
- Middle center
- Middle right
- Top left
- Top center
- Top right
- Centroid
- Shear center

Member Axis Shifted Modelling with Rigid End Offsets:





M_{span} = 40.4 kNm
$M_{support}$ = 31.9 kNm
M_{column} = 49 kNm

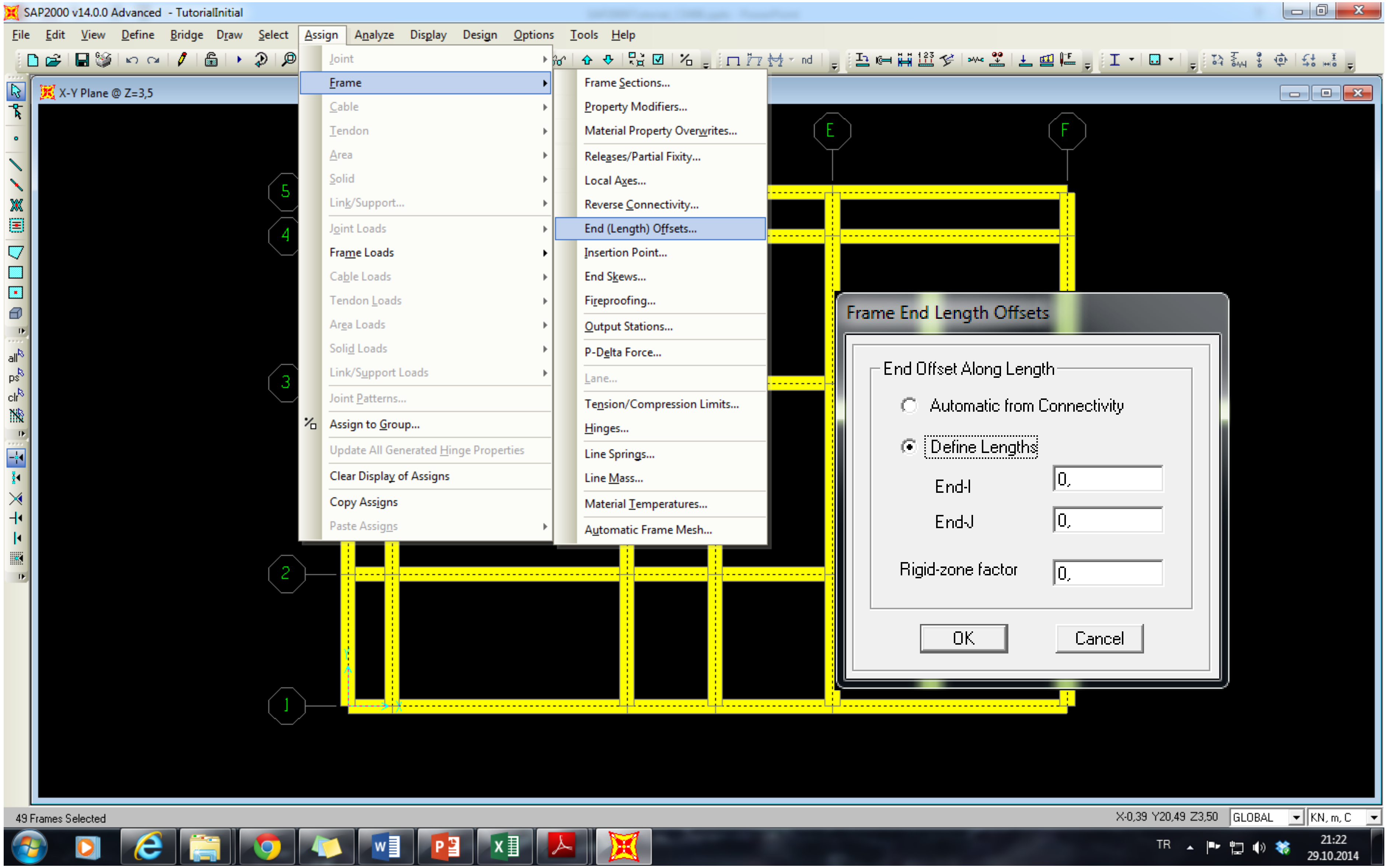


Lateral Displacement:
Δ=1.07 mm

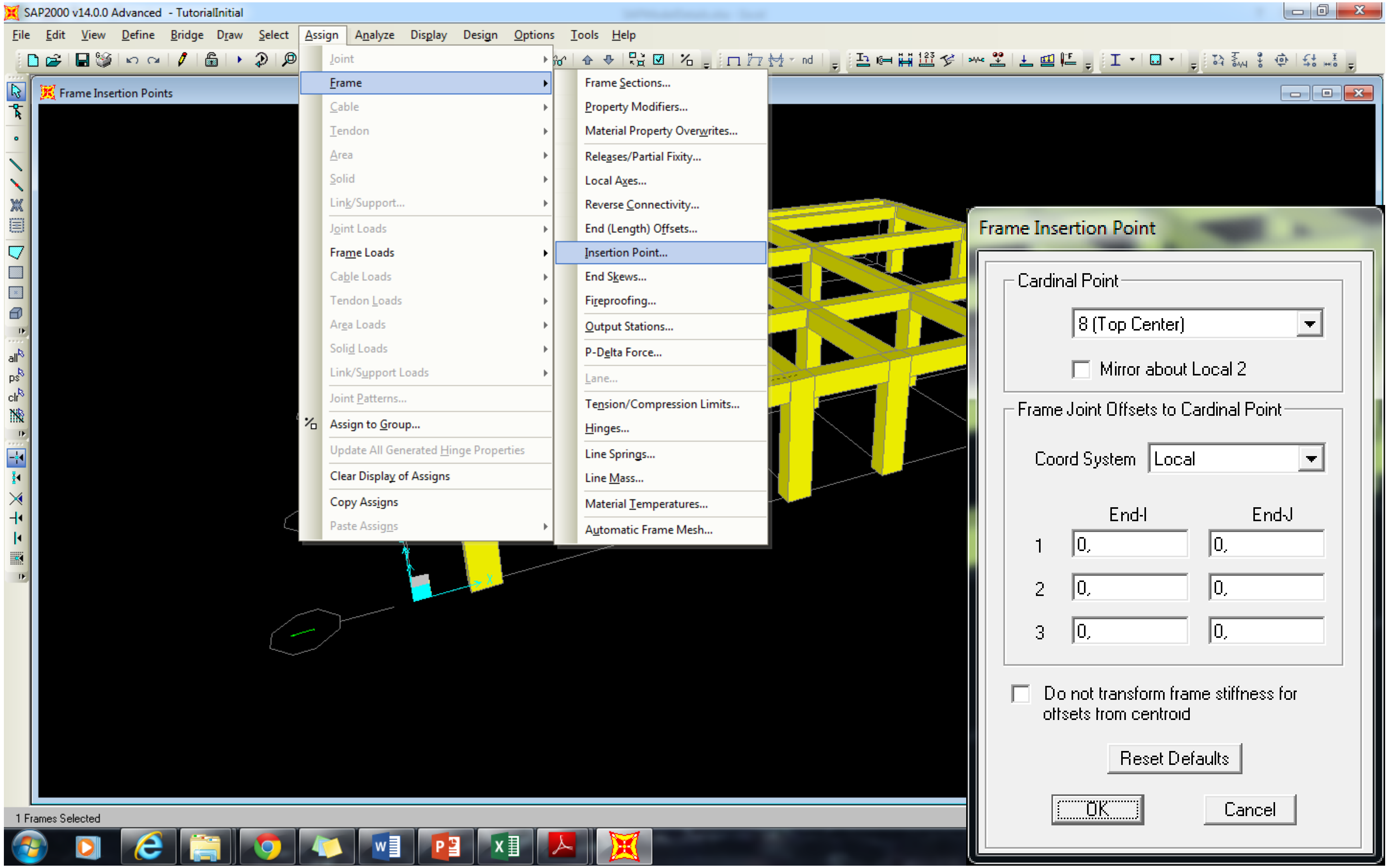
Summary of Results:

	Δ	M_{span}	$M_{support}$	M_{column}
Centroidal without Rigid End Zone	1,41	45	45	45
Centroidal with Rigid End Zone	1,07	38,3	34	46,7
Rigid End Zone Axis Shifting	1,07	40,4	31,9	49

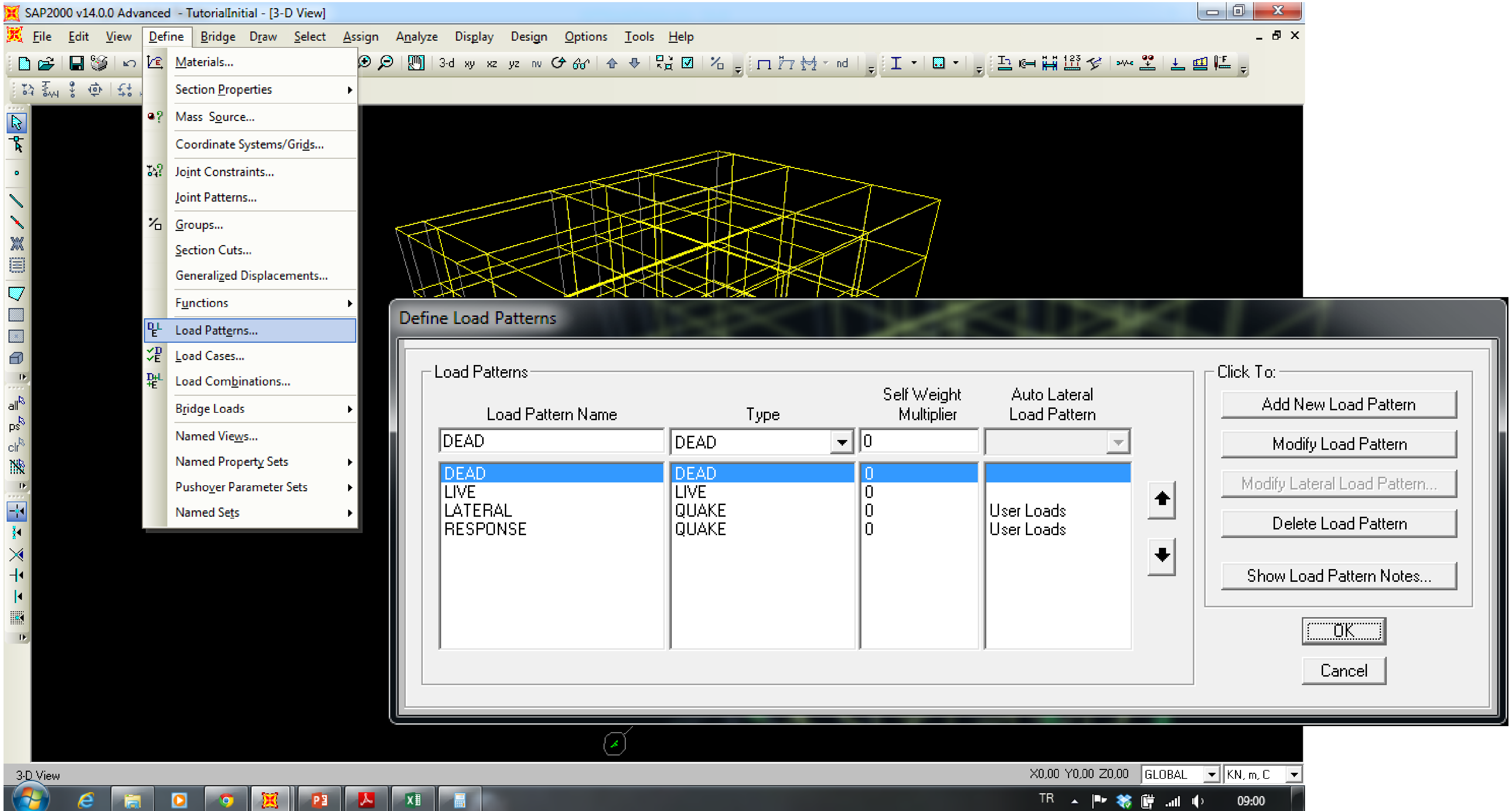
Define Rigid End Zones



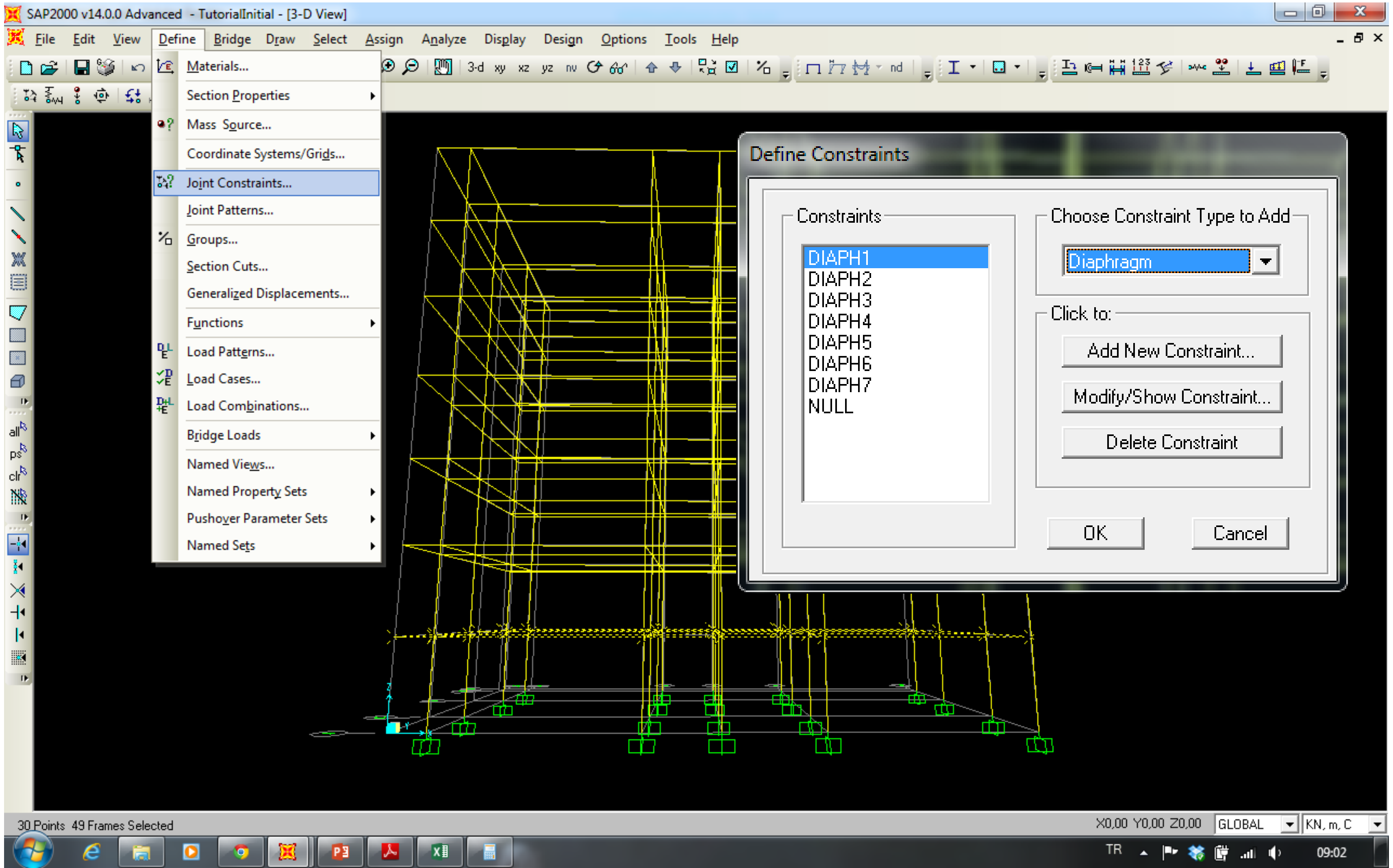
Axis shift


SAP2000 v14.0.0 Advanced - TutorialInitial
File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Tools Help
Frame Insertion Points
Joint
Frame
Cable
Tendon
Area
Solid
Link/Support...
Joint Loads
Frame Loads
Cable Loads
Tendon Loads
Area Loads
Solid Loads
Link/Support Loads
Joint Patterns...
Assign to Group...
Update All Generated Hinge Properties
Clear Display of Assigns
Copy Assigns
Paste Assigns
Frame Sections...
Property Modifiers...
Material Property Overwrites...
Releases/Partial Fixity...
Local Axes...
Reverse Connectivity...
End (Length) Offsets...
Insertion Point...
End Skews...
Fireproofing...
Output Stations...
P-Delta Force...
Lane...
Tension/Compression Limits...
Hinges...
Line Springs...
Line Mass...
Material Temperatures...
Automatic Frame Mesh...
Frame Insertion Point
Cardinal Point
8 (Top Center)
Mirror about Local 2
Frame Joint Offsets to Cardinal Point
Coord System
Local
End-I
End-J
1
0.
0.
2
0.
0.
3
0.
0.
Do not transform frame stiffness for offsets from centroid
Reset Defaults
OK
Cancel
1 Frames Selected

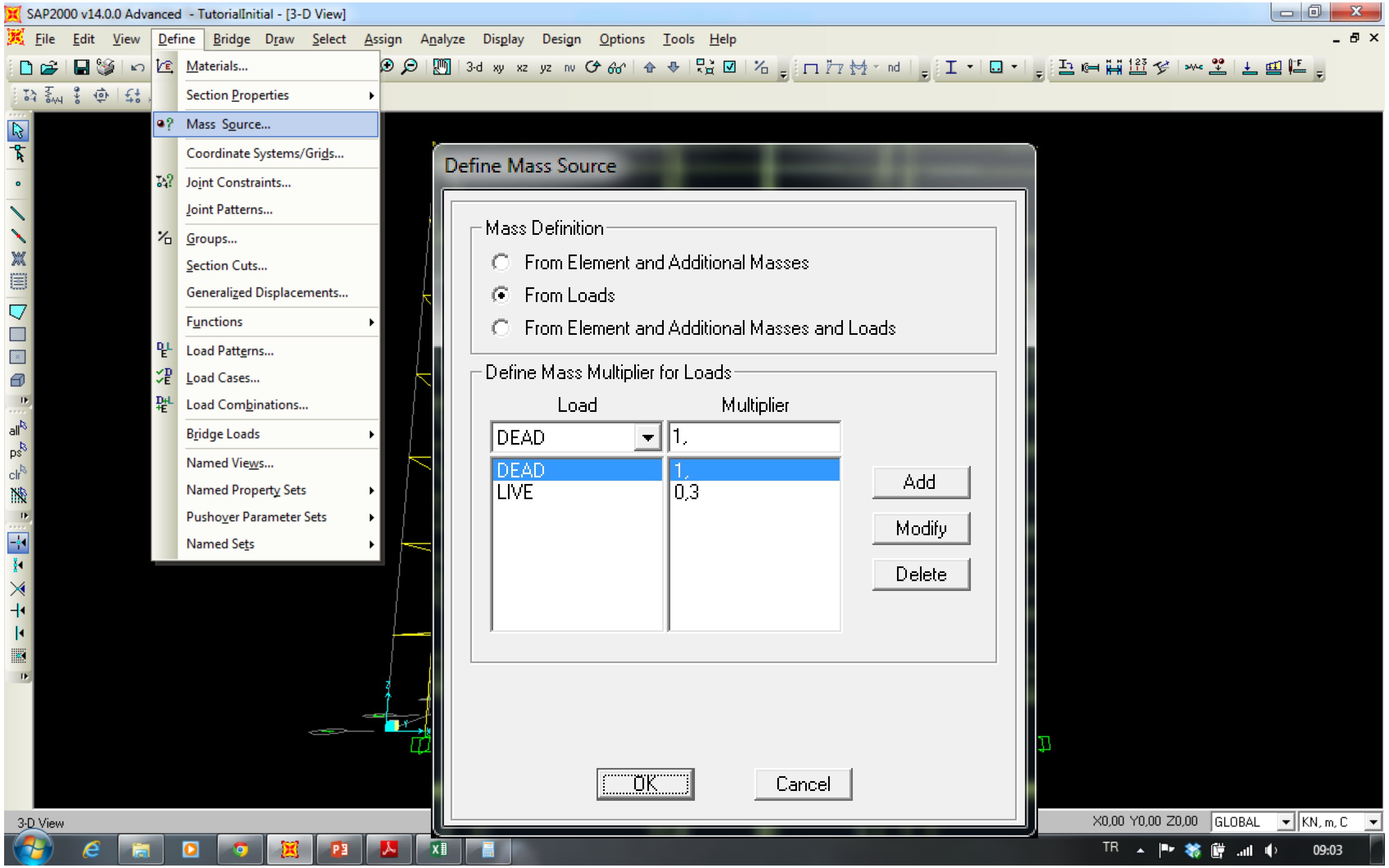
Define the loads (load patterns → load cases → load combinations)



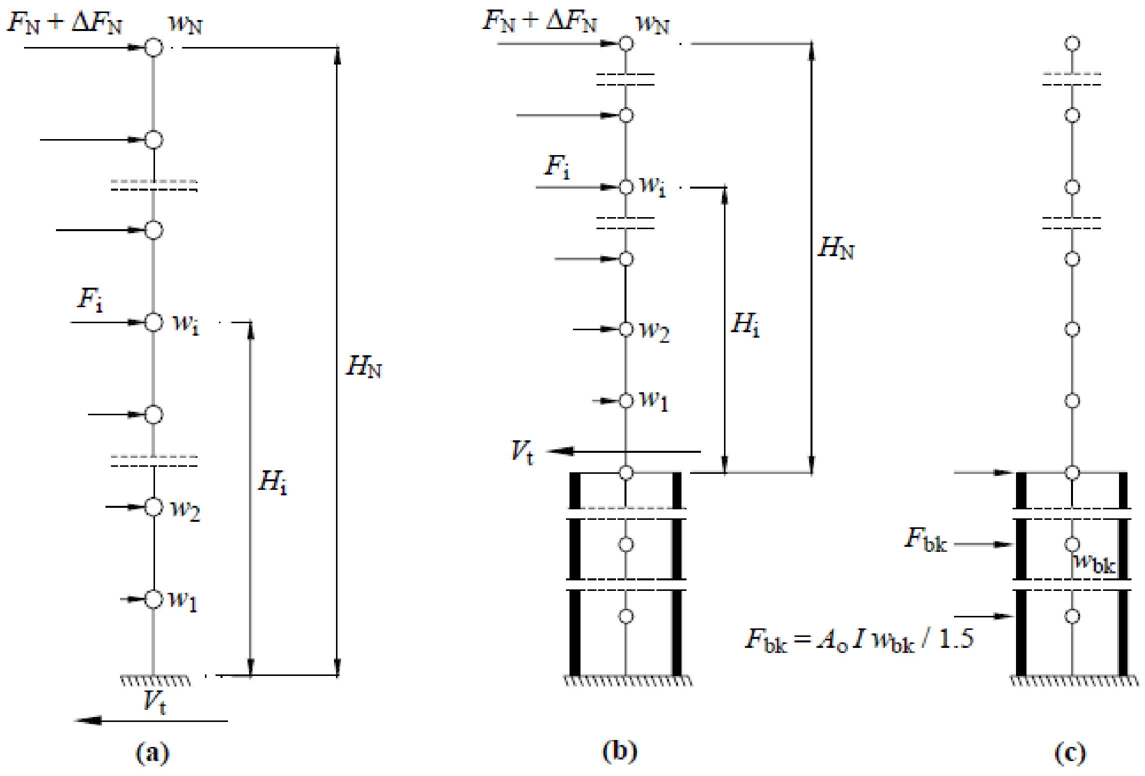
Define rigid diaphragms at floor levels



Define the mass source



Lineer Static Procedure (TEC 2007)



Şekil 2.6

2.7.2.1 – Denk.(2.4) ile hesaplanan toplam eşdeğer deprem yükü, bina katlarına etkiyen eşdeğer deprem yüklerinin toplamı olarak **Denk.(2.7)** ile ifade edilir (**Şekil 2.6a**):

$$V_t = \Delta F_N + \sum_{i=1}^{N} F_i \qquad \textbf{(2.7)}$$

2.7.2.2 – Binanın N'inci katına (tepesine) etkiyen *ek eşdeğer deprem yükü* ΔF_N'in değeri **Denk.(2.8)** ile belirlenecektir.

$$\Delta F_N = 0.0075\, N\, V_t \qquad \textbf{(2.8)}$$

2.7.2.3 – Toplam eşdeğer deprem yükünün ΔF_N dışında geri kalan kısmı, N'inci kat dahil olmak üzere, bina katlarına **Denk.(2.9)** ile dağıtılacaktır.

$$F_i = (V_t - \Delta F_N)\, \frac{w_i\, H_i}{\sum_{j=1}^{N} w_j\, H_j} \qquad \textbf{(2.9)}$$

Lineer Static Procedure (TEC 2007)

$$A(T) = A_0\, I\, S(T) \qquad (2.1)$$
$$S_{ae}(T) = A(T)\, g$$

2.4.3. Spektrum Katsayısı

2.4.3.1 – Denk.(2.1)'de yer alan *Spektrum Katsayısı*, $S(T)$, yerel zemin koşullarına ve bina doğal periyodu T'ye bağlı olarak **Denk.(2.2)** ile hesaplanacaktır (**Şekil 2.5**).

$$S(T) = 1 + 1.5\,\frac{T}{T_A} \qquad (0 \le T \le T_A)$$
$$S(T) = 2.5 \qquad (T_A < T \le T_B) \qquad (2.2)$$
$$S(T) = 2.5\left(\frac{T_B}{T}\right)^{0.8} \qquad (T_B < T)$$

Denk.(2.2)'deki *Spektrum Karakteristik Periyotları*, T_A ve T_B, **Bölüm 6**'da **Tablo 6.2** ile tanımlanan *Yerel Zemin Sınıfları*'na bağlı olarak **Tablo 2.4**'te verilmiştir.

TABLO 2.4 – SPEKTRUM KARAKTERİSTİK PERİYOTLARI (T_A, T_B)

Tablo 6.2'ye göre Yerel Zemin Sınıfı	T_A *(saniye)*	T_B *(saniye)*
Z1	0.10	0.30
Z2	0.15	0.40
Z3	0.15	0.60
Z4	0.20	0.90

2.7.4.1 – Eşdeğer Deprem Yükü Yöntemi'nin uygulanması durumunda, binanın deprem doğrultusundaki hakim doğal periyodu, **Denk.(2.11)** ile hesaplanan değerden daha büyük alınmayacaktır.

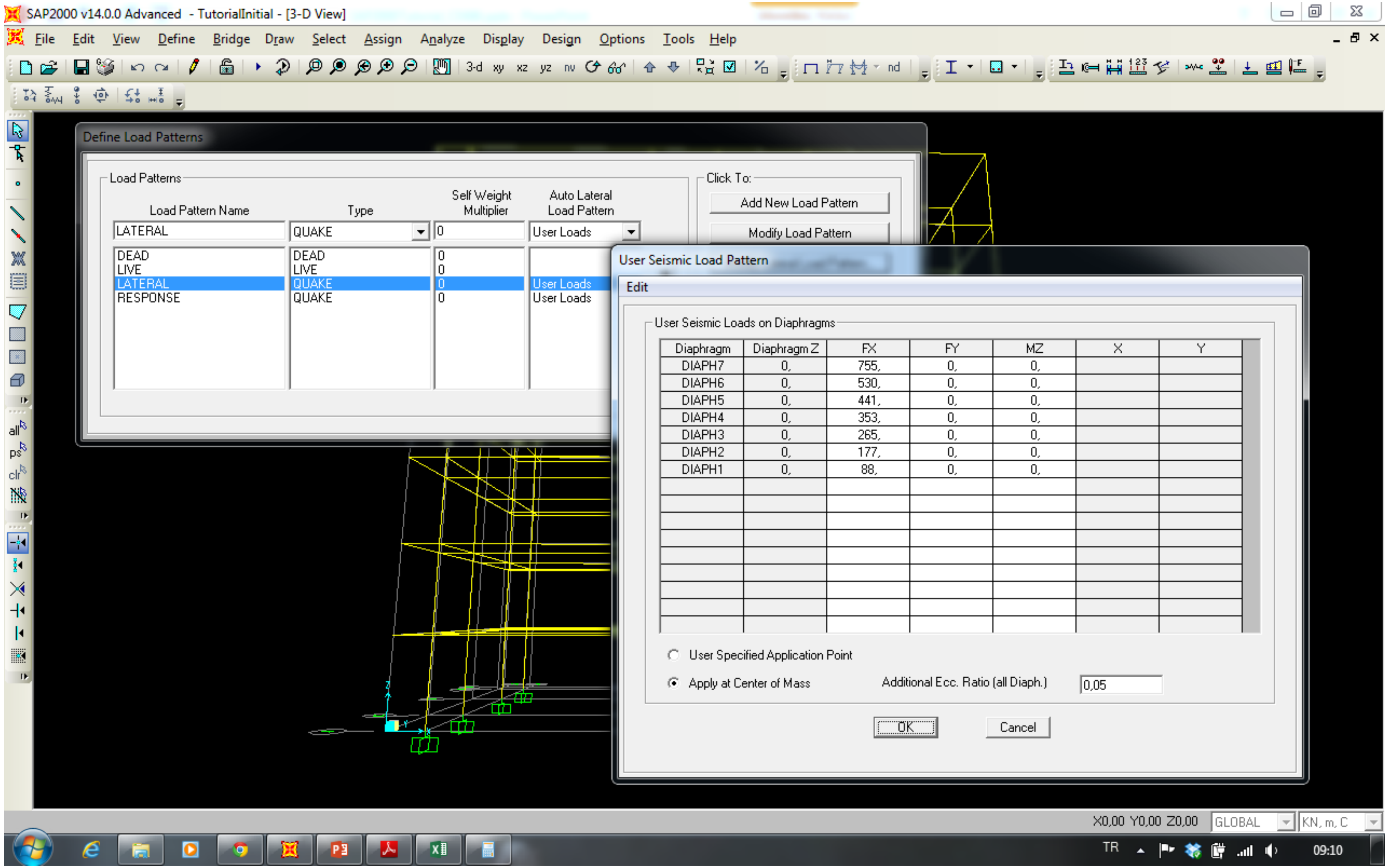
$$T_1 = 2\pi \left(\frac{\sum_{i=1}^{N} m_i\, d_{fi}^2}{\sum_{i=1}^{N} F_{fi}\, d_{fi}} \right)^{1/2}$$

TABLO 2.2 – ETKİN YER İVMESİ KATSAYISI (A_0)

Deprem Bölgesi	A_0
1	0.40
2	0.30
3	0.20
4	0.10

T=	0.88206	s					
Ta=	0.1			W=	32972.1	kN	
Tb=	0.3			Ao=	0.3		
R=	4			I=	1		
N=	7	number of story		0.10*Ao*I*W=	989.163	kN	
S(T)=	1.054966						
Ra(T)=	4						
A(T1)=	0.31649						
Vt=	2608.834	kN	GREATER				
DeltaFn=	136.9638	kN					
Story	weight	height	Fi				
7	4710.3	24.5	755	kN			
6	4710.3	21	530	kN			
5	4710.3	17.5	441	kN			
4	4710.3	14	353	kN			
3	4710.3	10.5	265	kN			
2	4710.3	7	177	kN			
1	4710.3	3.5	88	kN			
			2608.834	kN			

Lineer Static Procedure (TEC 2007)



Response Spectrum

- **Scale factor :** 0.736 (Ao*g/R=(0.3*9.814)/4 = 0.9814)