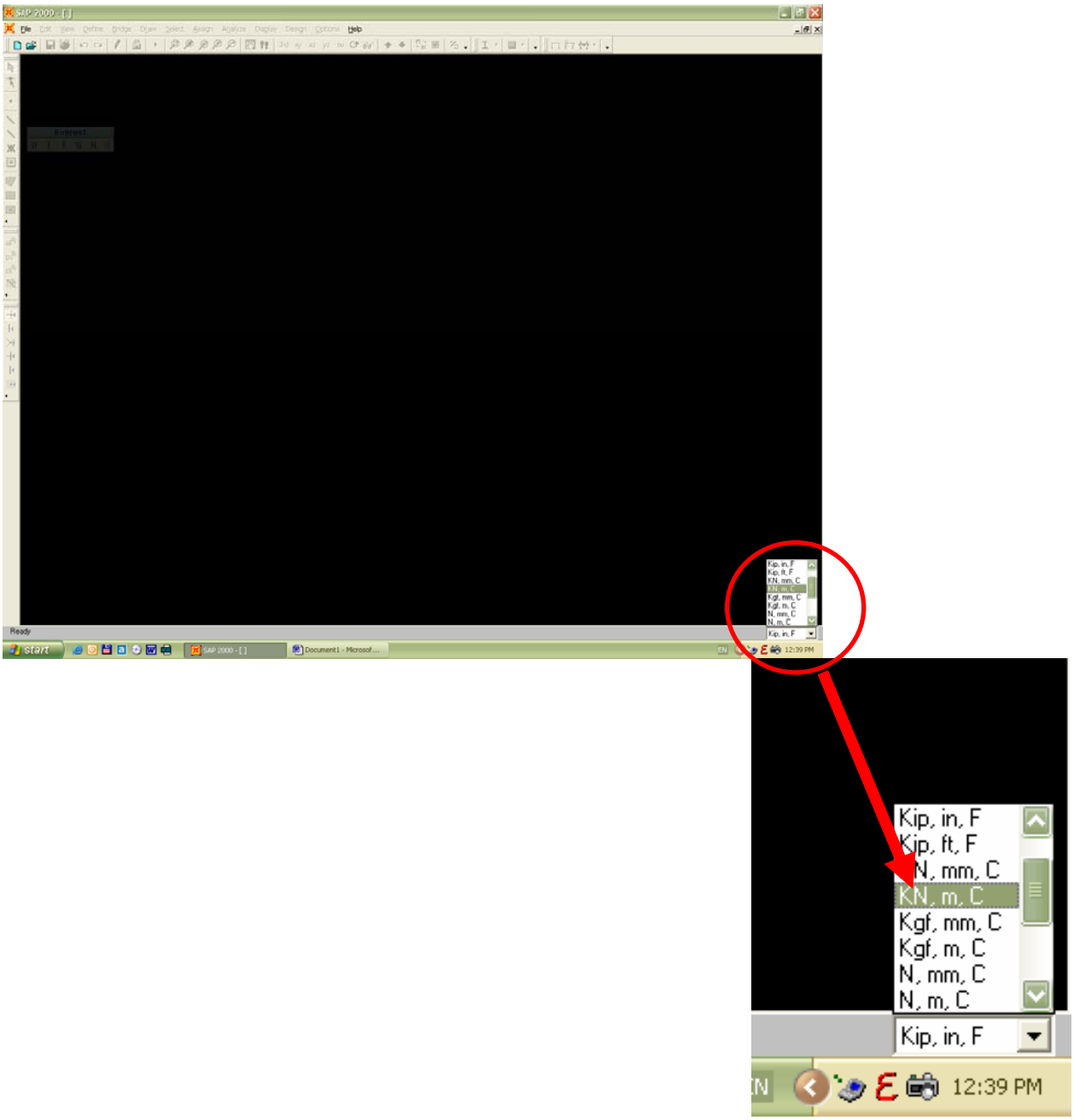
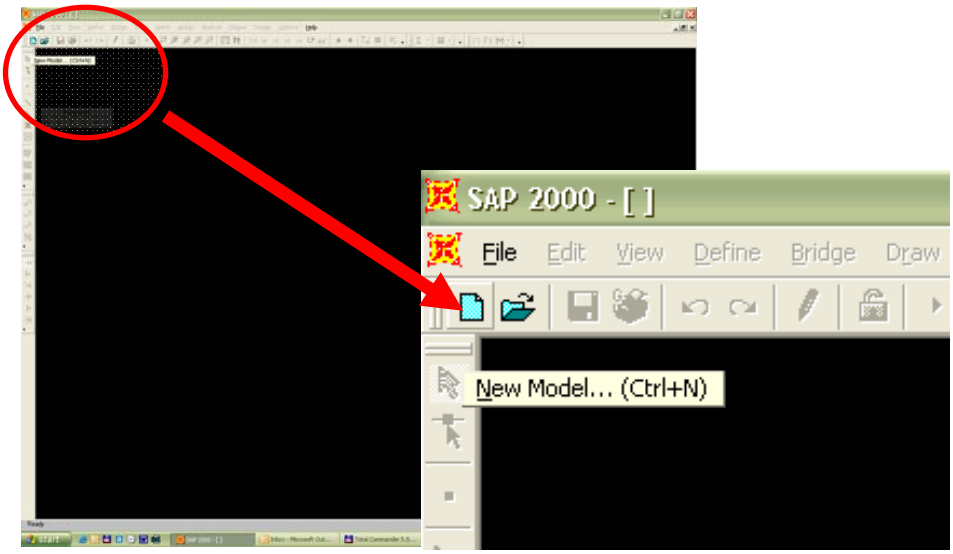
PROCEDURA DE INTRODUCERE A DATELOR ÎN SAP2000

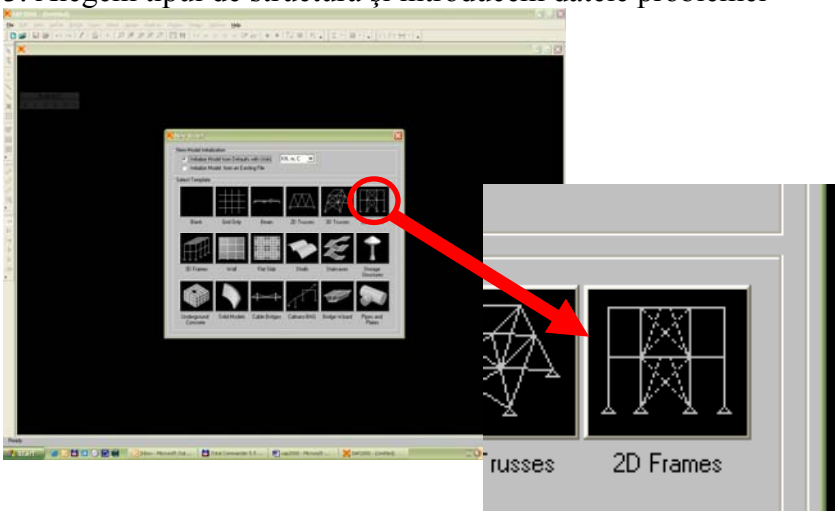
1. Setarea unităților de măsură din **Kip, in, F** în **KN, m, C**

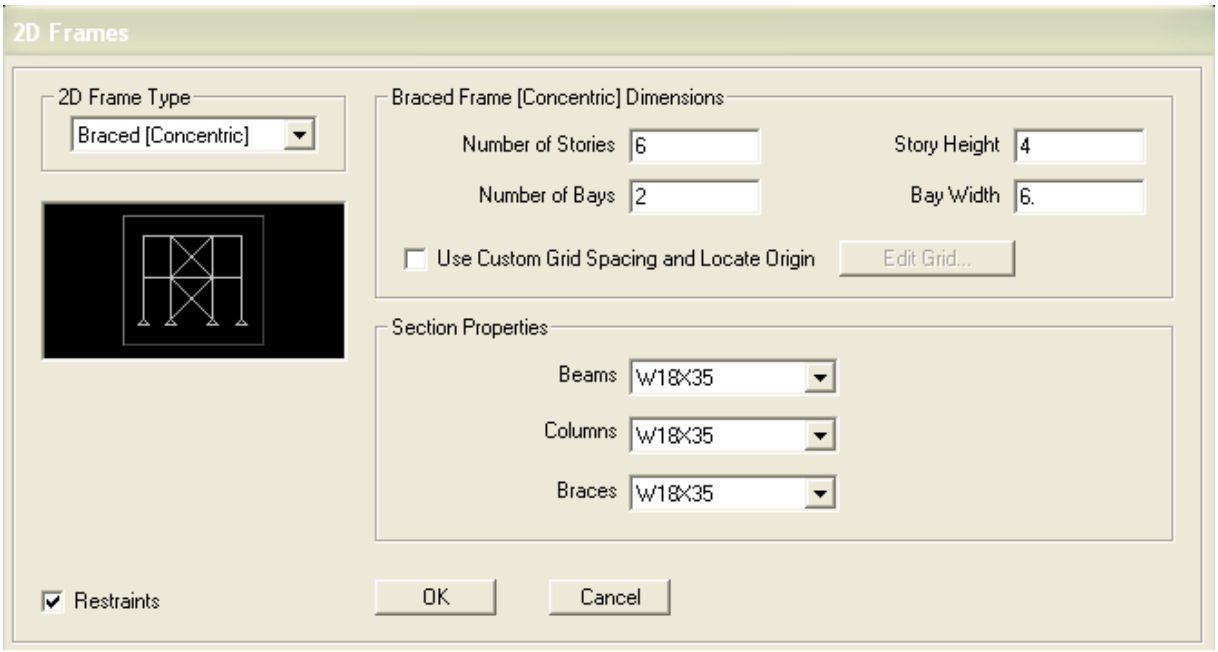


2. Deschiderea unei planşe noi de lucru: **File → New Model**

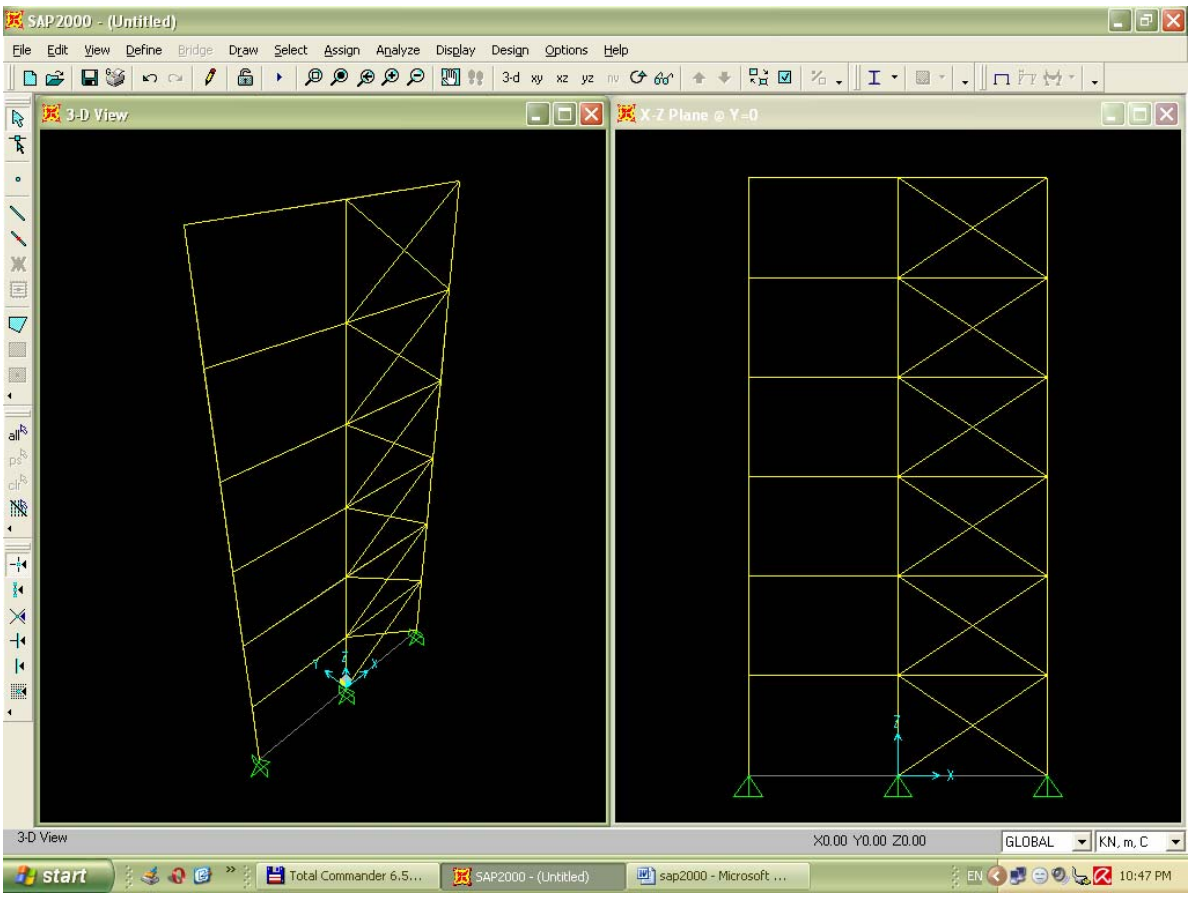


3. Alegem tipul de structură şi introducem datele problemei

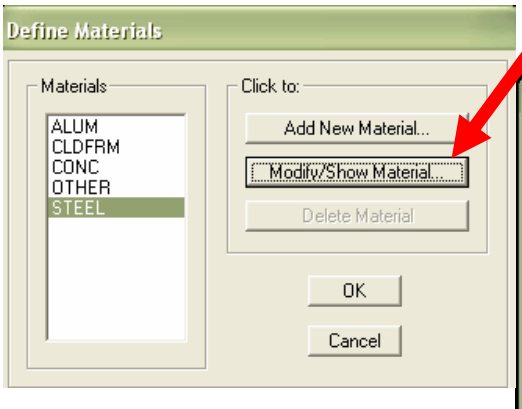


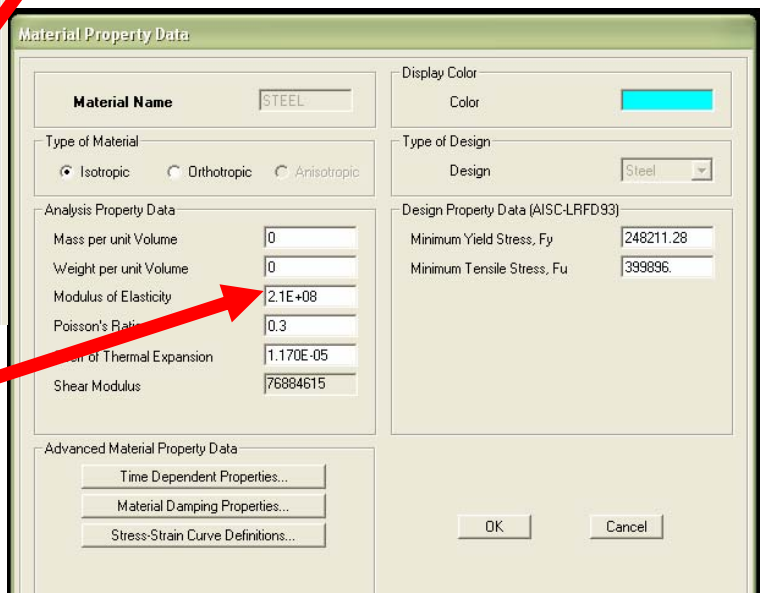


Rezultatul:



3. Definirea materialelor: **Define → Materials**

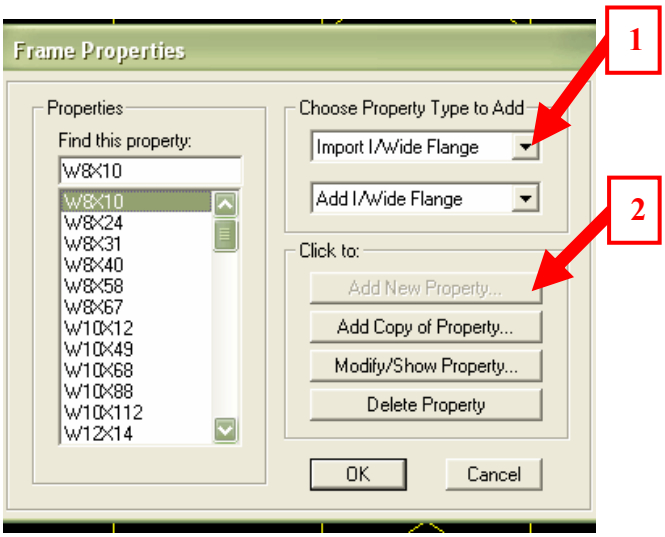


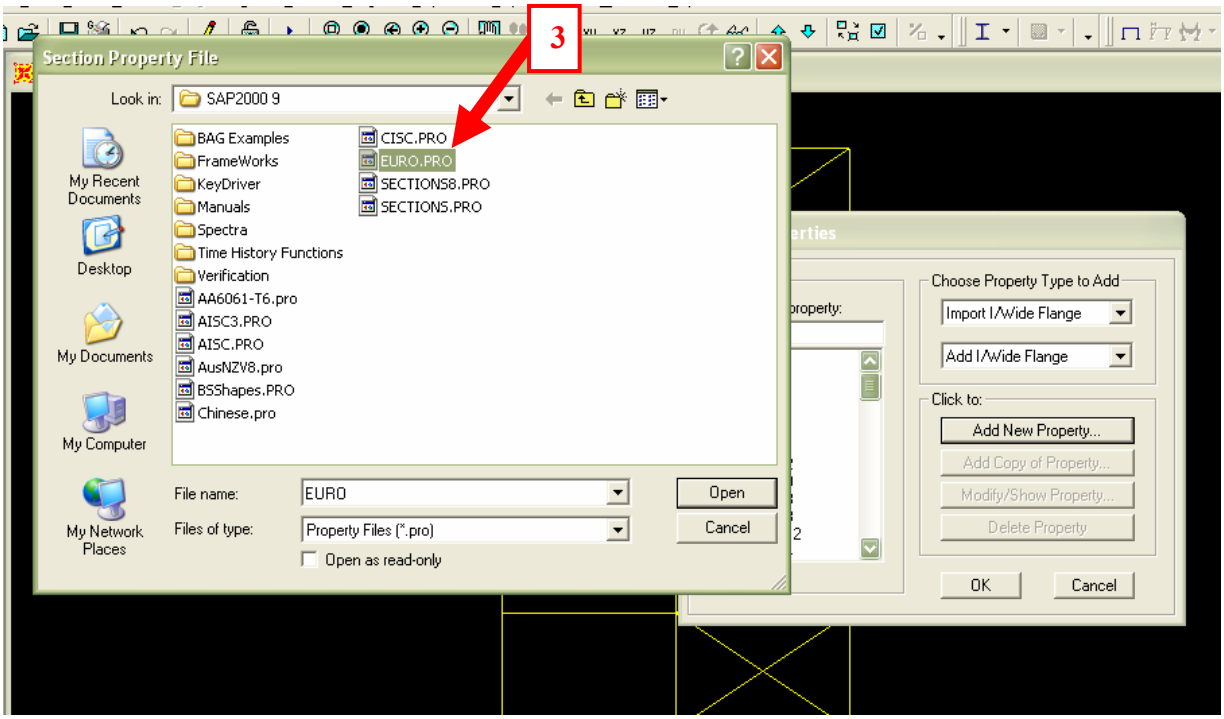


Definim modulul de elasticitate pentru oțel

Masa şi greutatea specifică le trecem cu 0.

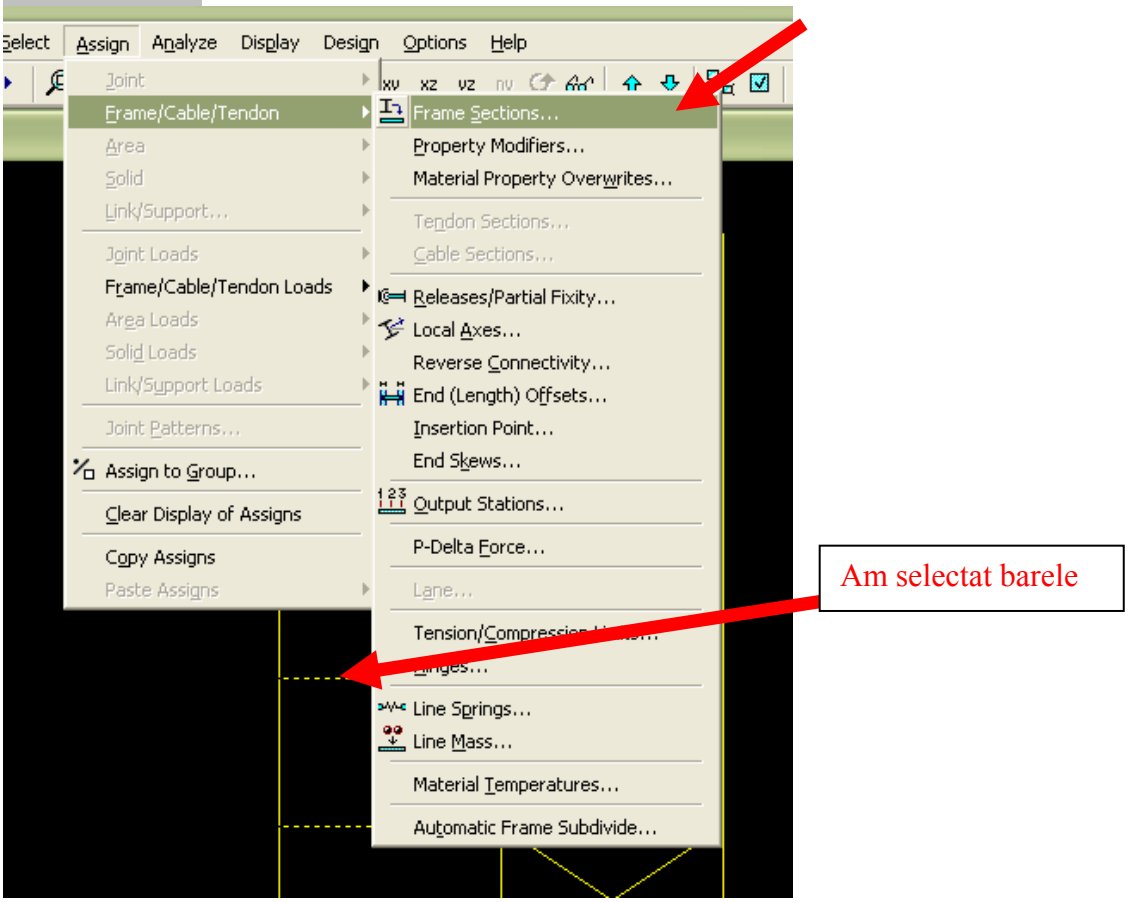
3. Definirea secțiunilor: **Define → Frame sections**



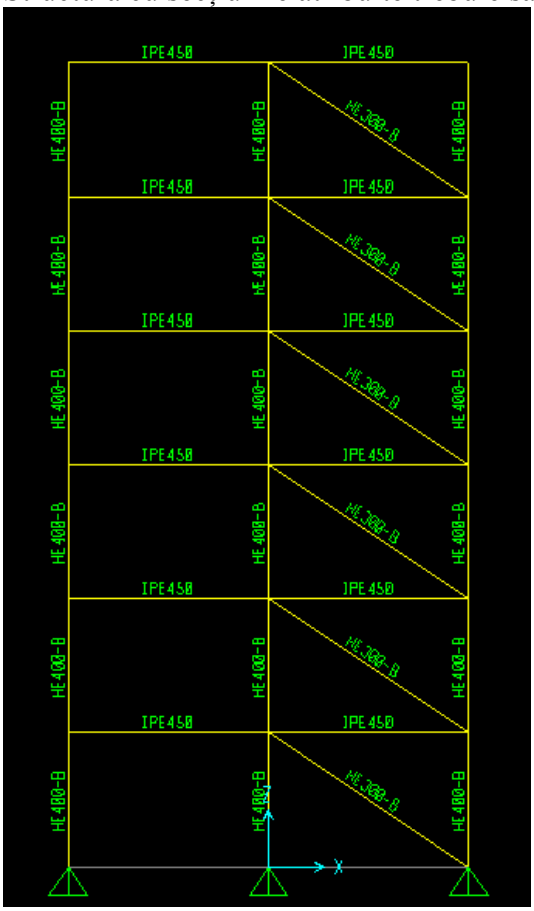


Alegem cu **CTRL** **HE 400B** pentru stâlpi, **IPE 450** pentru rigle şi **HE 300B** pentru contravântuiri.

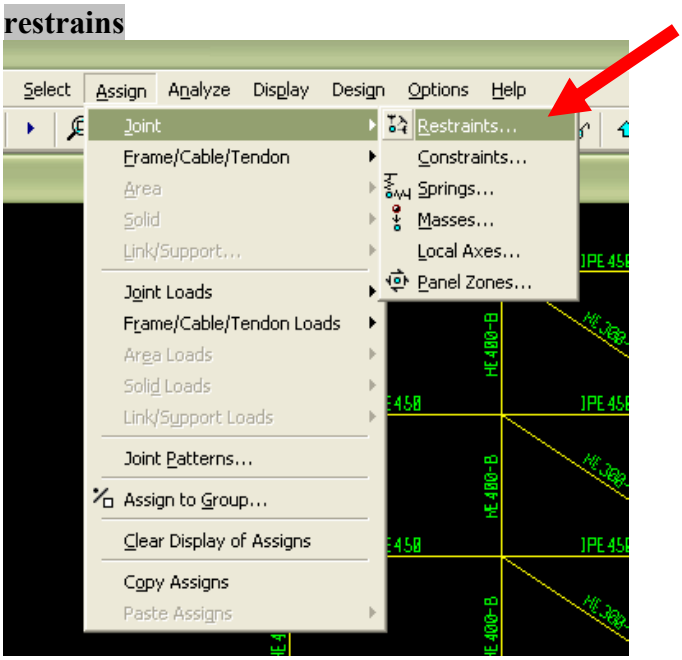
4. Atribuirea secțiunilor: selectăm barele şi atribuim prin **Asign → Frame/Cable/Tendon→ Frame Section**



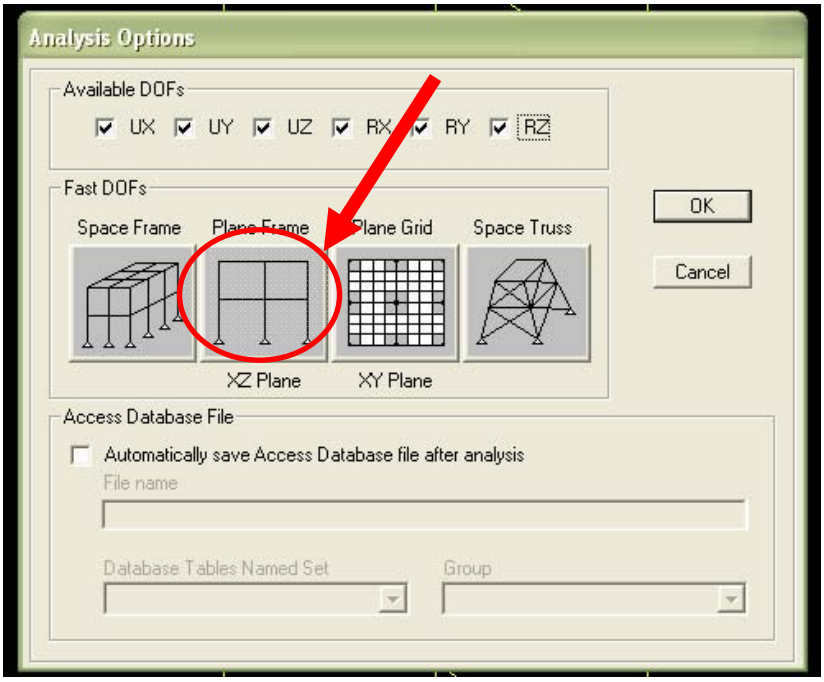
Structura cu secțiunile atribuite trebuie să arate în final aşa:



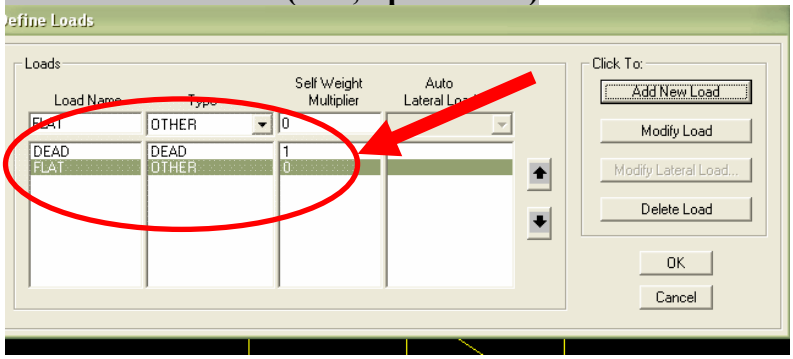
5. Atribuirea reazemelor: selectăm nodurile si atribuim reazeme blocate: **Asign → Joints→ restrains**



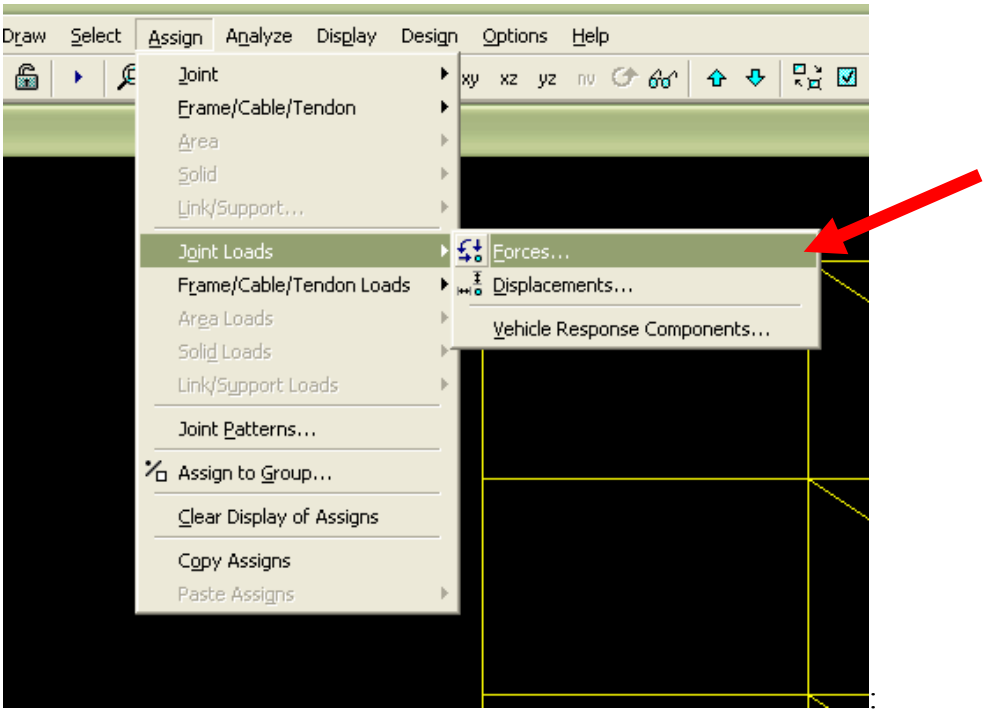
6. Definirea gradelor de libertate (structură plană):
Analyse → set analysis options→ plane frame (XZ)



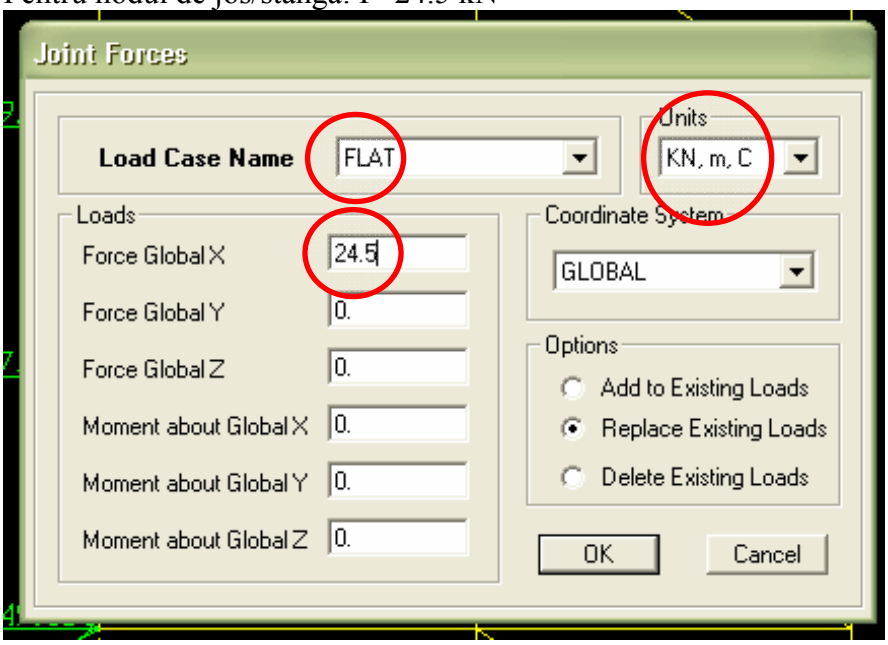
7. Definirea cazului de încărcare statică:
Define → Load case (Flat, tip OTHER)



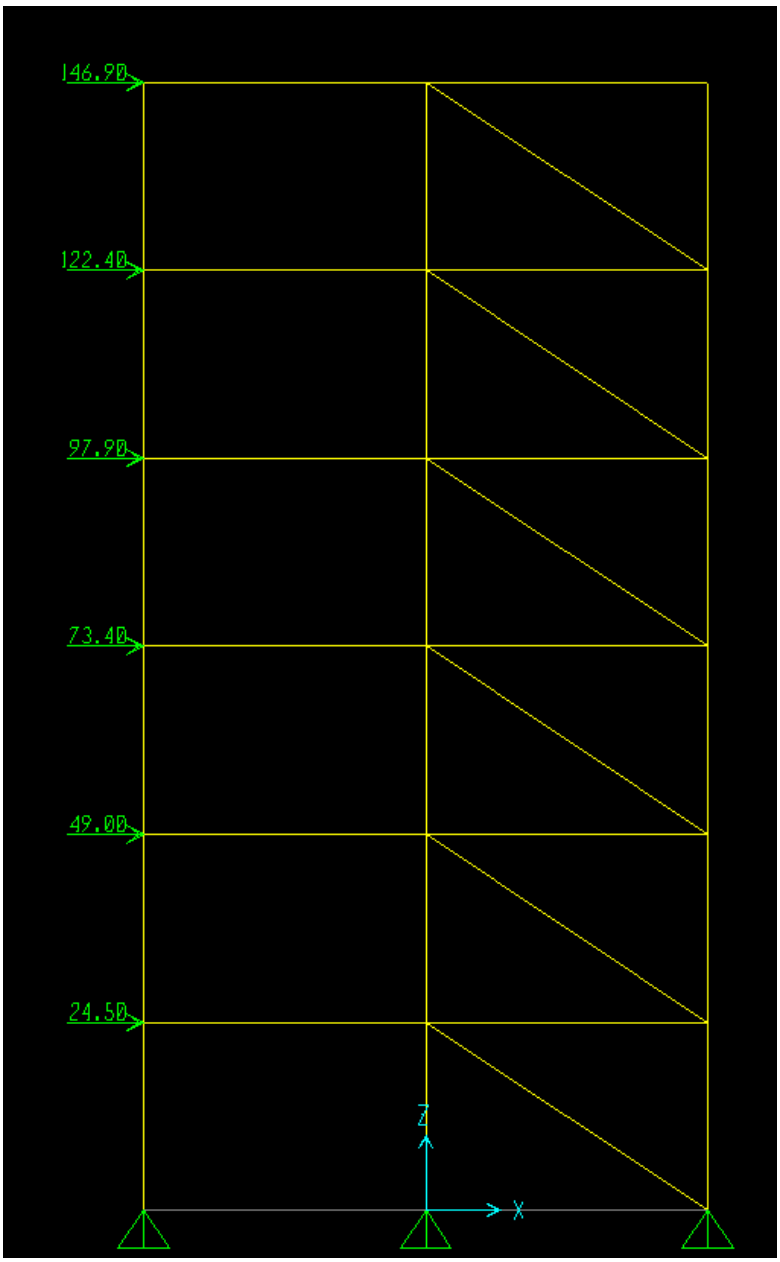
8. Definirea încărcări în noduri (forțe laterale): selectare nod
Asign → Joint Loads → Forces (în cazul de încărcare FLAT)



Pentru nodul de jos/stânga: F=24.5 kN

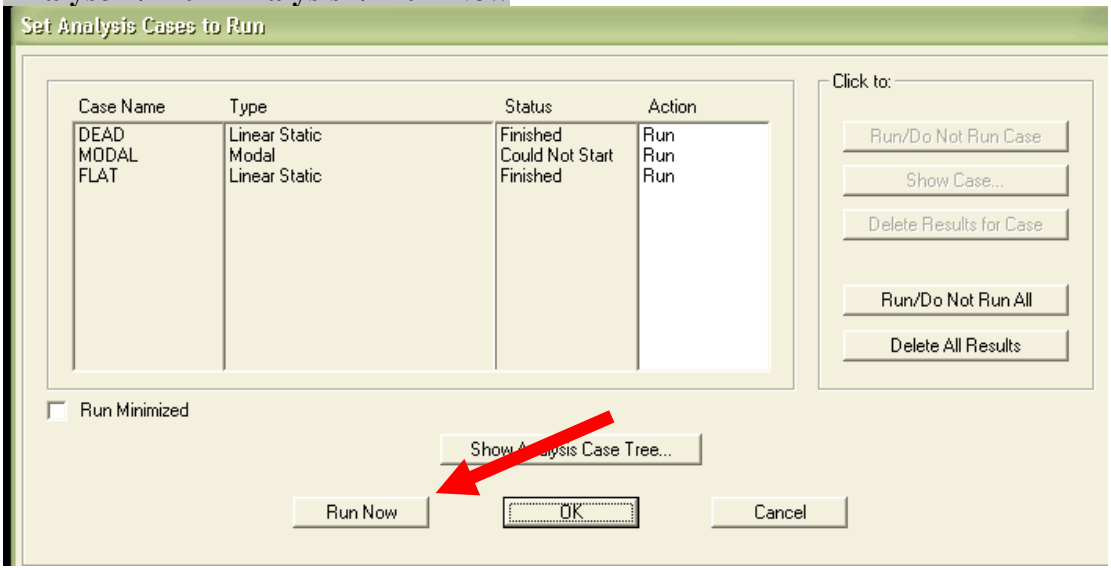


Structura cu forțele aplicate va fi:



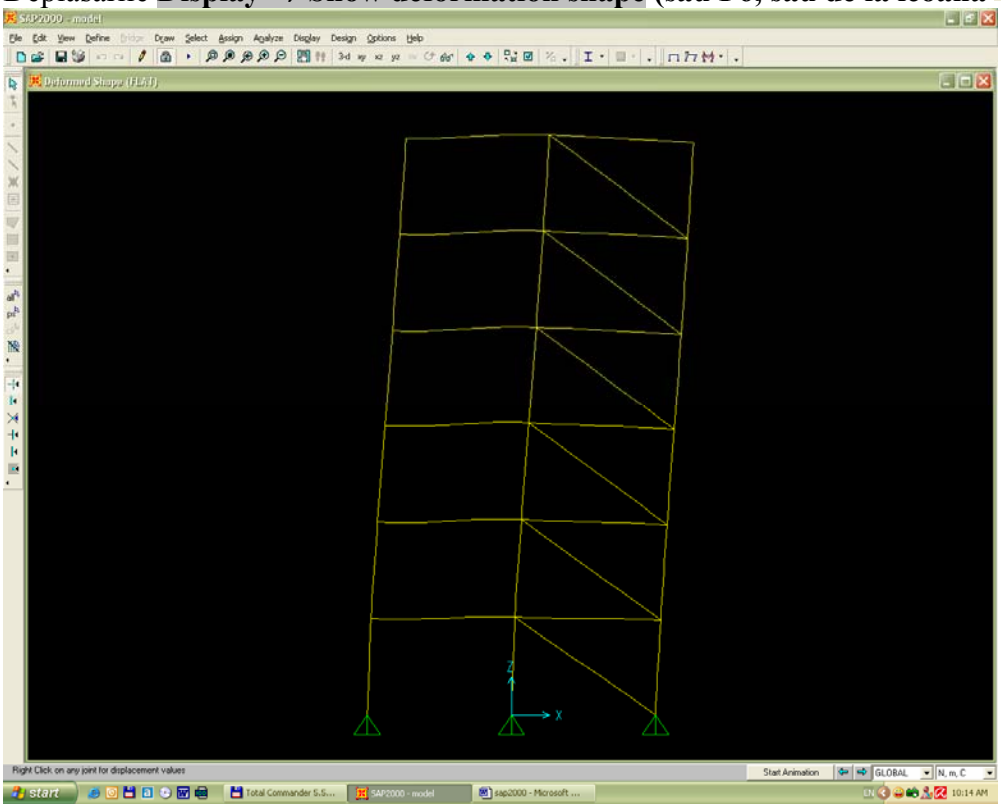
9. Efectuarea calculului:

Analyse → Run Analysis→ Run Now



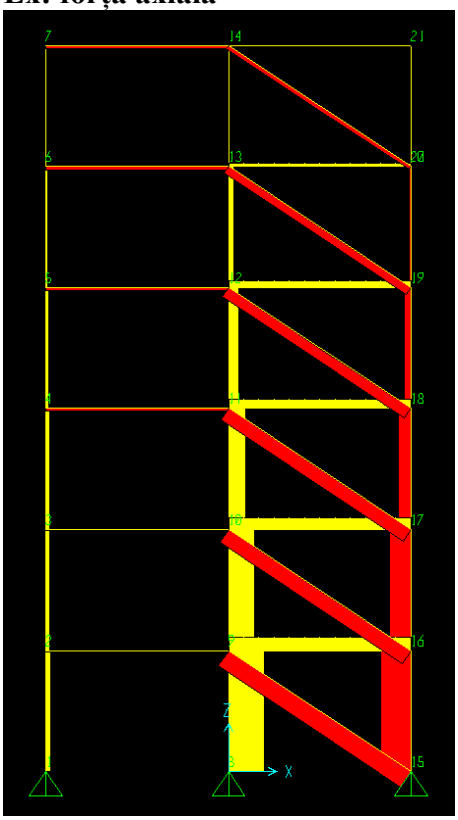
10. Vizualizarea rezultatelor:

Deplasările **Display → Show deformation shape (sau F6, sau de la icoană**  **)**

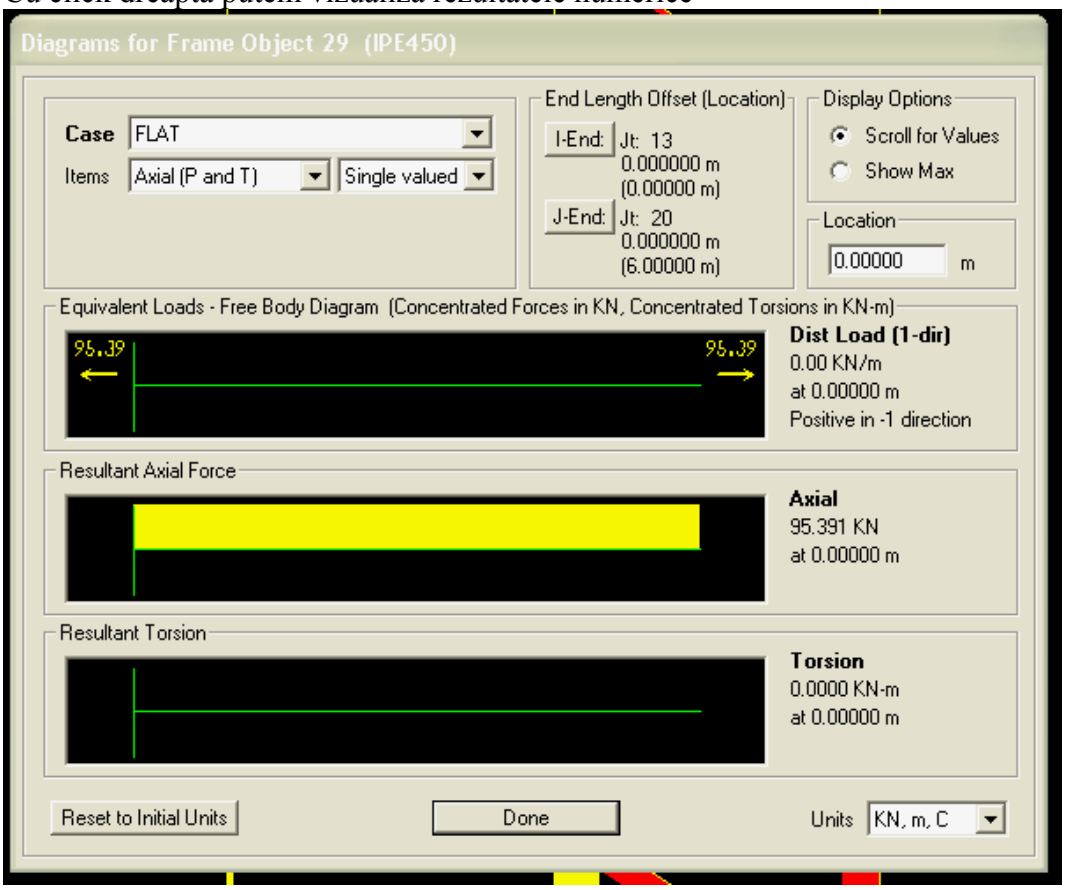


Eforturile: **Display → Show forces/stresses → Frames/Cables**
Pentru moment: **Moment 3-3**
Pentru forța axială: **Axial force**
Pentru forța tăietoare: **Shear 2-2**

Ex: forța axială

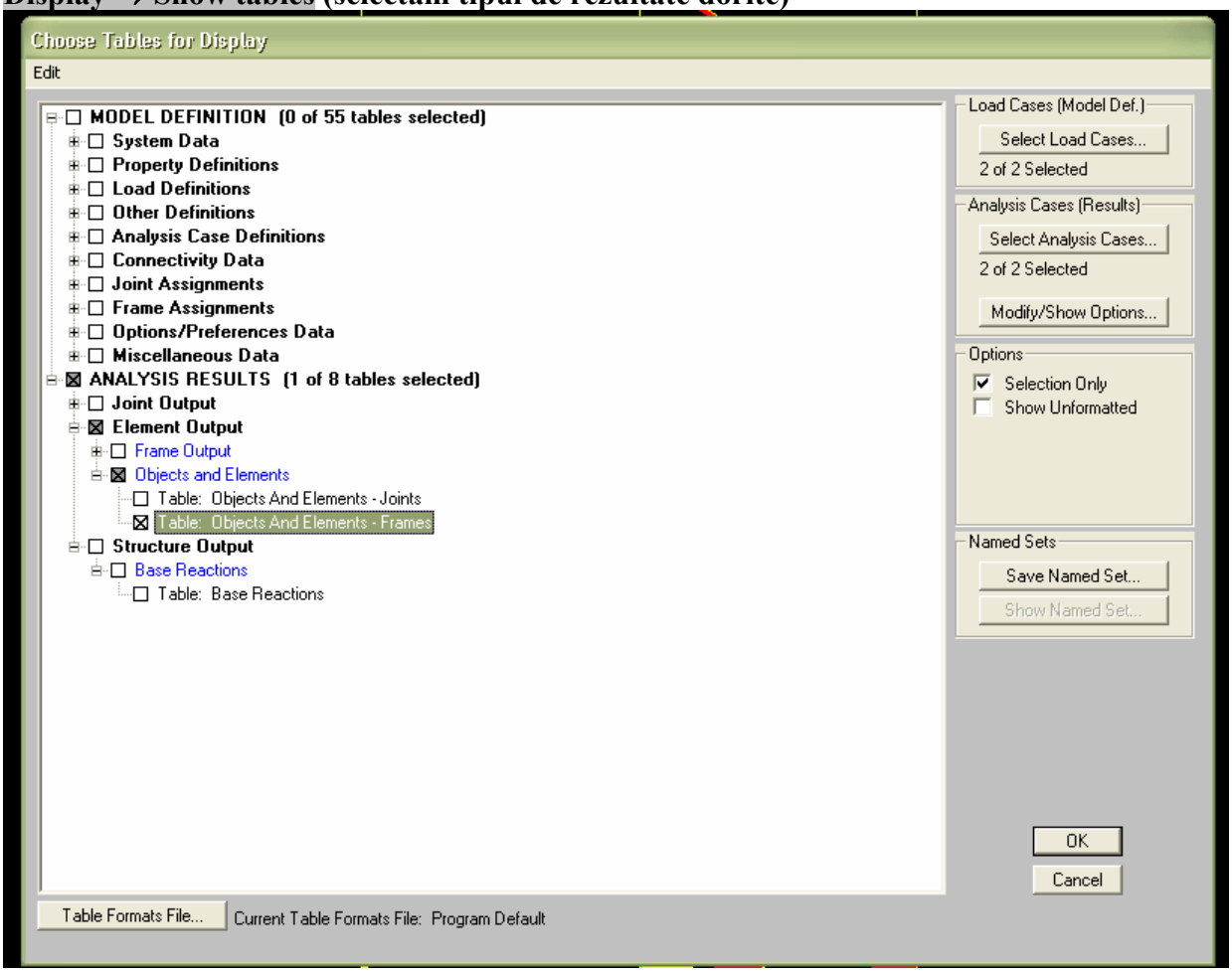


Cu click dreapta putem vizualiza rezultatele numerice



Vizualizarea rezultatelor, tabelar: selectăm elementele

Display → Show tables (selectăm tipul de rezultate dorite)

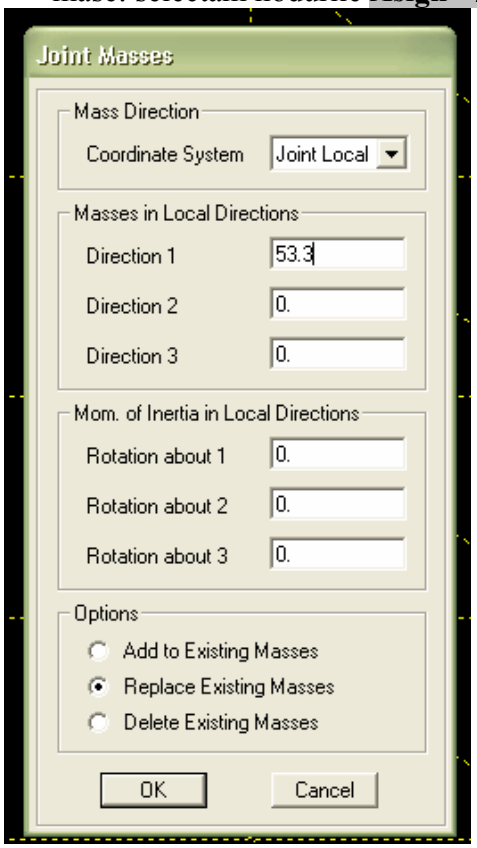


11. Procedura de introducere a datelor pentru analiză modală/spectrală:

ANALIZA MODALĂ

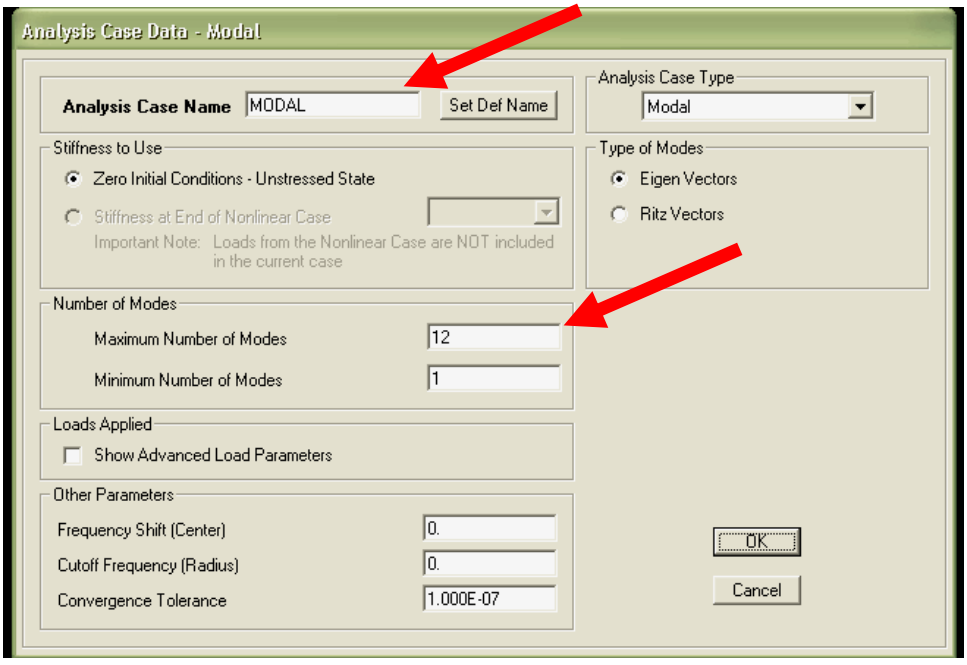
Întâi deblocăm modelul de la  **→** 

- mase: selectăm nodurile **Asign → Joints → Masses**



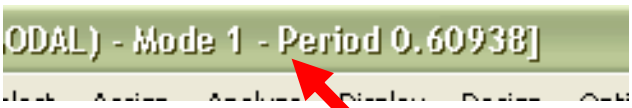
- definire analiză modală

Define → Analysis Cases → Modal

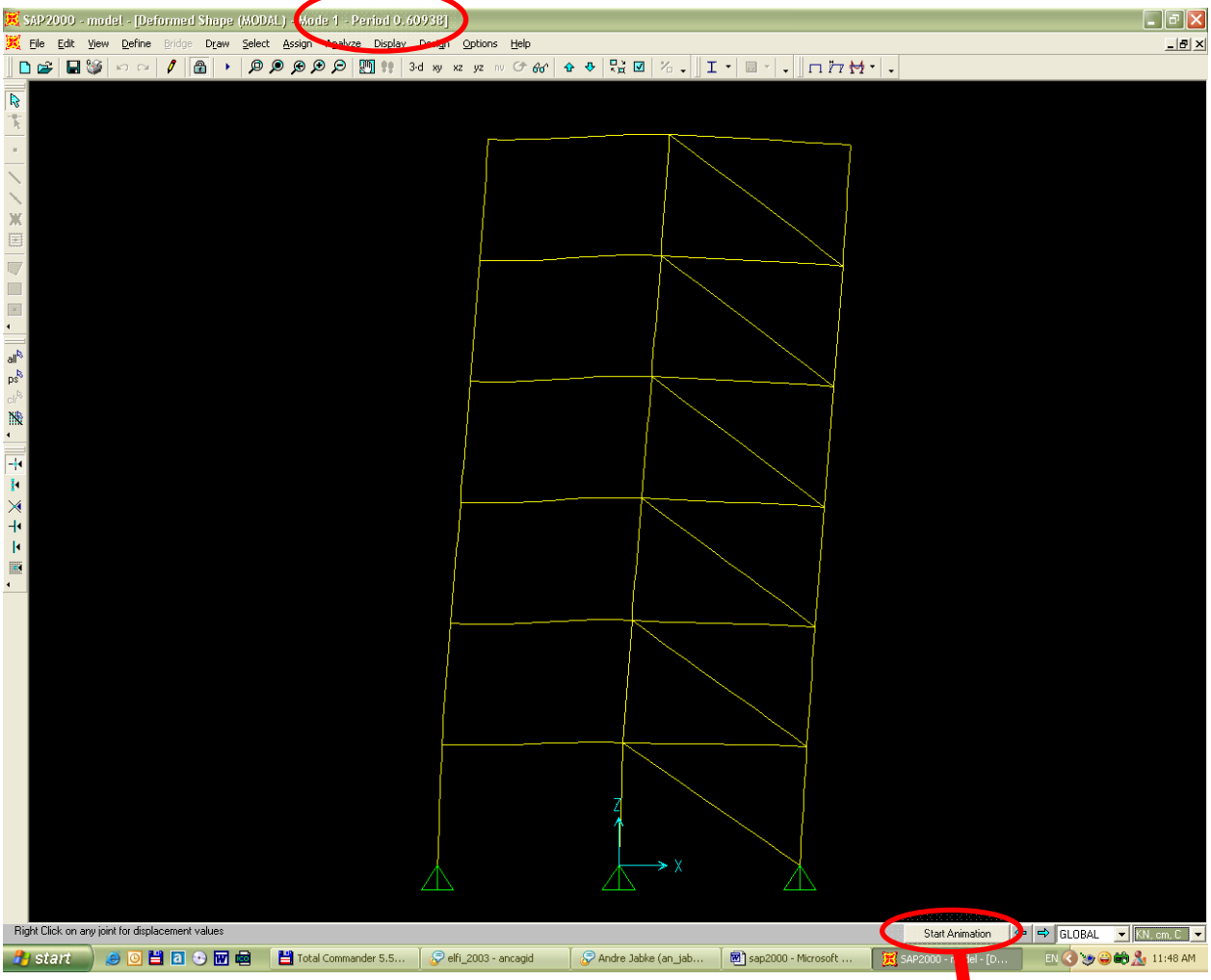


Efectuare analiză: RUN 

Vizualizarea rezultatelor:



ex.: deplasări



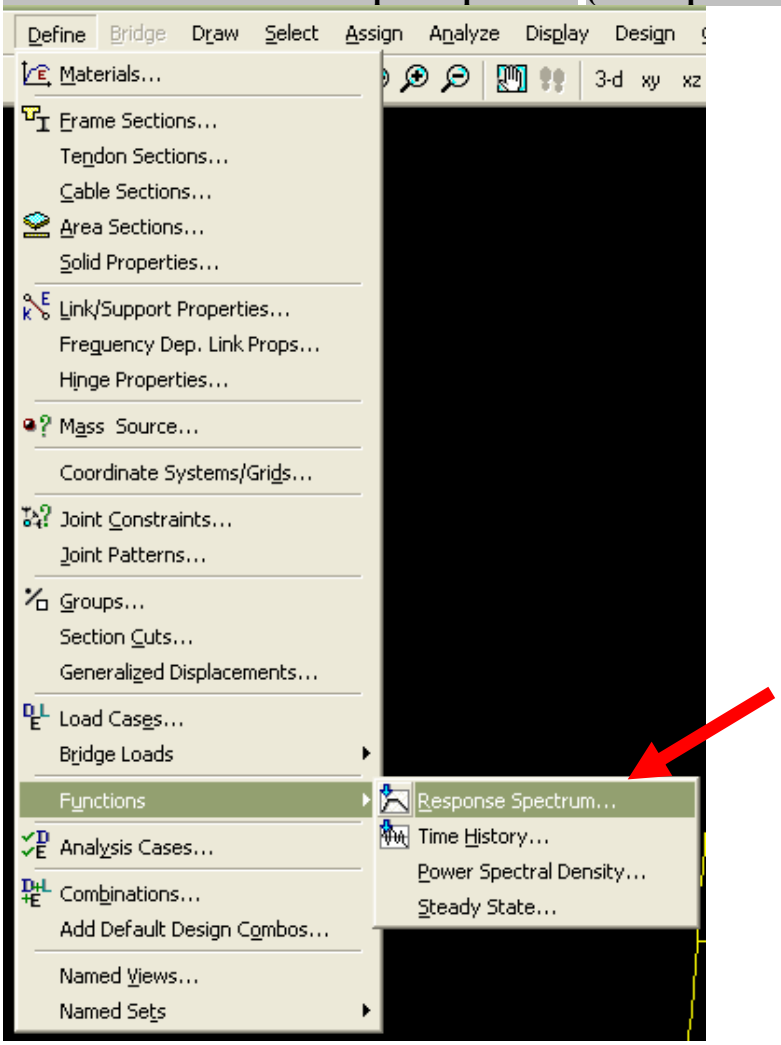
modelul poate fi şi animat

ANALIZA SPECTRALĂ

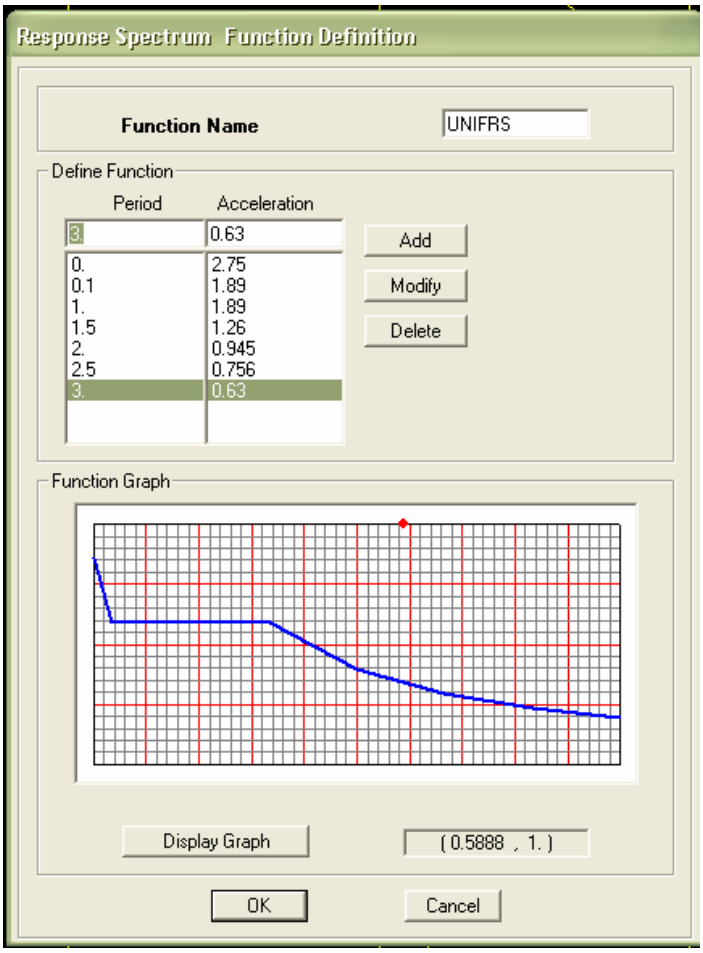
Întâi deblocăm modelul de la  **→**

Definirea spectrului

Define → Functions → Respons Spectrum (User Spectrum, Add new function)

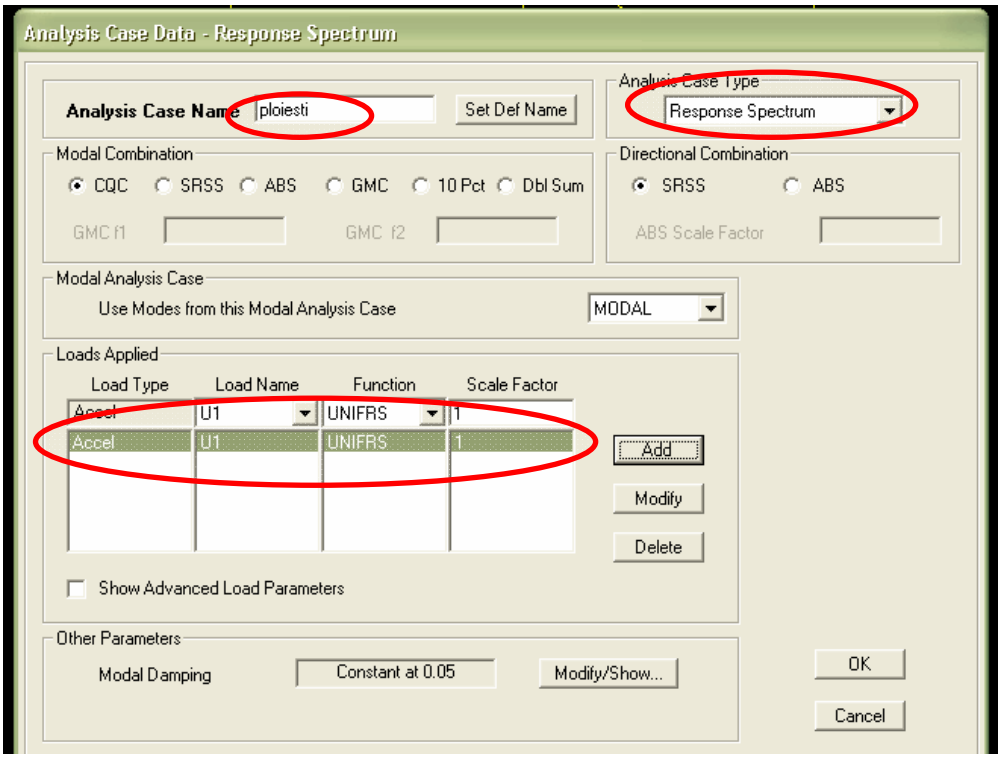


Introducem datele dorite



Definim calculul spectral

Define → Analysis Cases → Add new → Respons Spectrum



Efectuare analiză: RUN 

Şi vizualizăm după metodele descrise mai sus