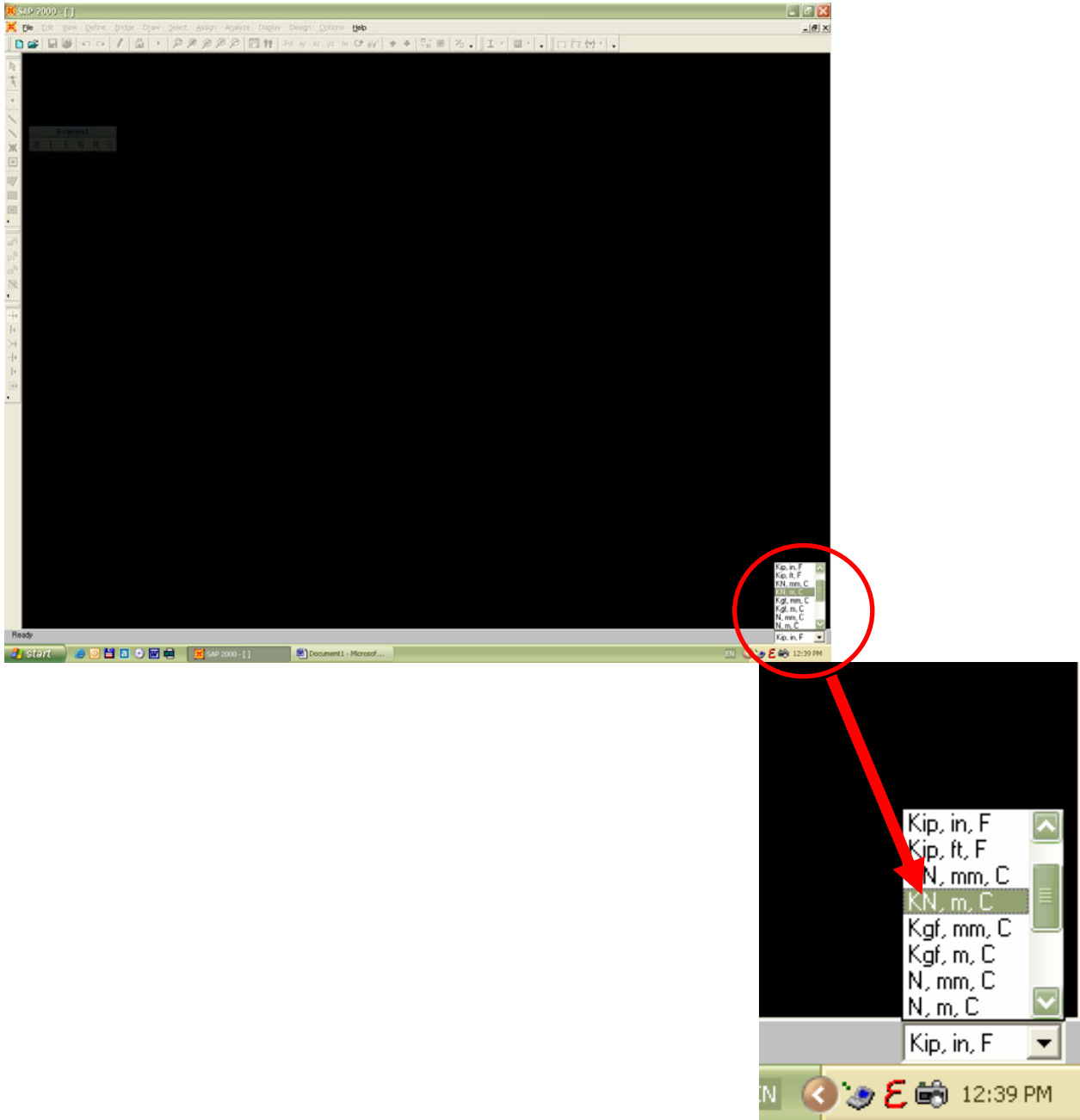
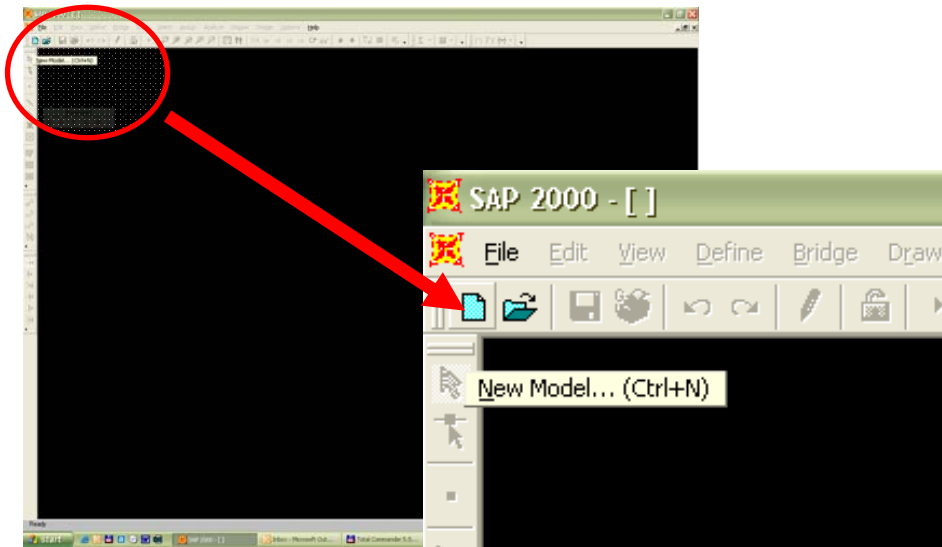


# PROCEDURA DE INTRODUCERE A DATELOR ÎN SAP2000

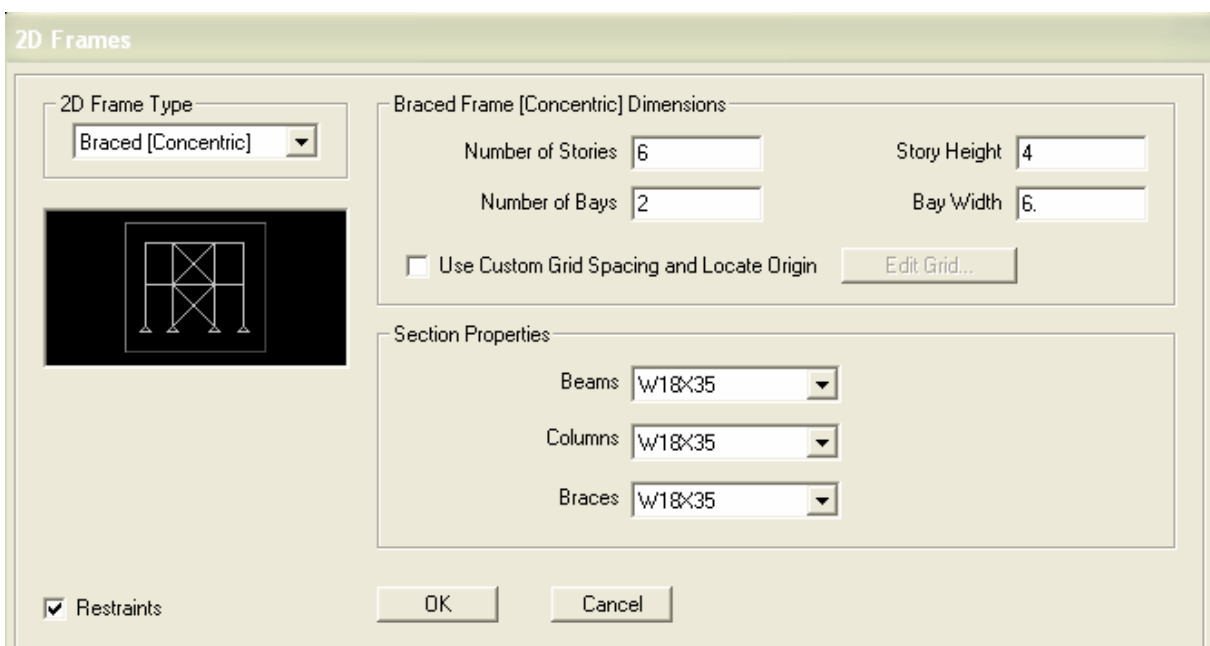
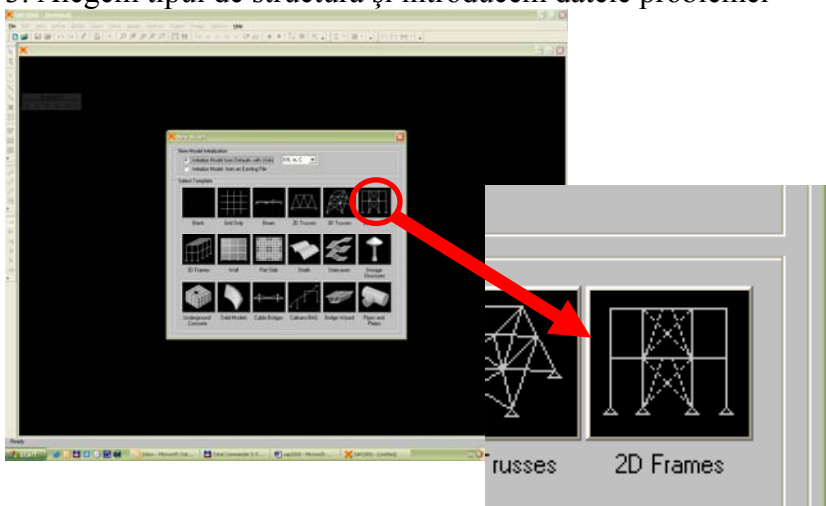
## 1. Setarea unităților de măsură din **Kip, in, F** în **KN, m, C**



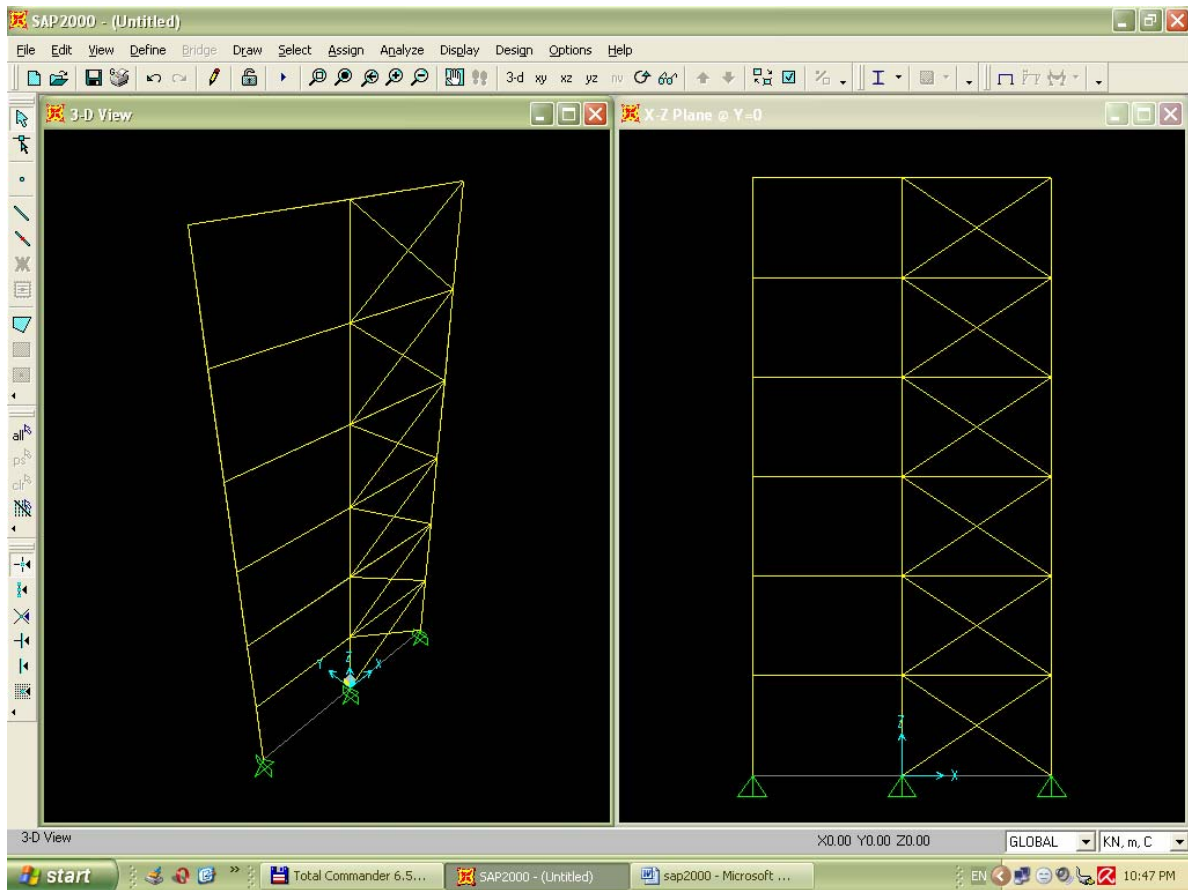
2. Deschiderea unei planșe noi de lucru: **File → New Model**



3. Alegem tipul de structură și introducem datele problemei



## Rezultatul:



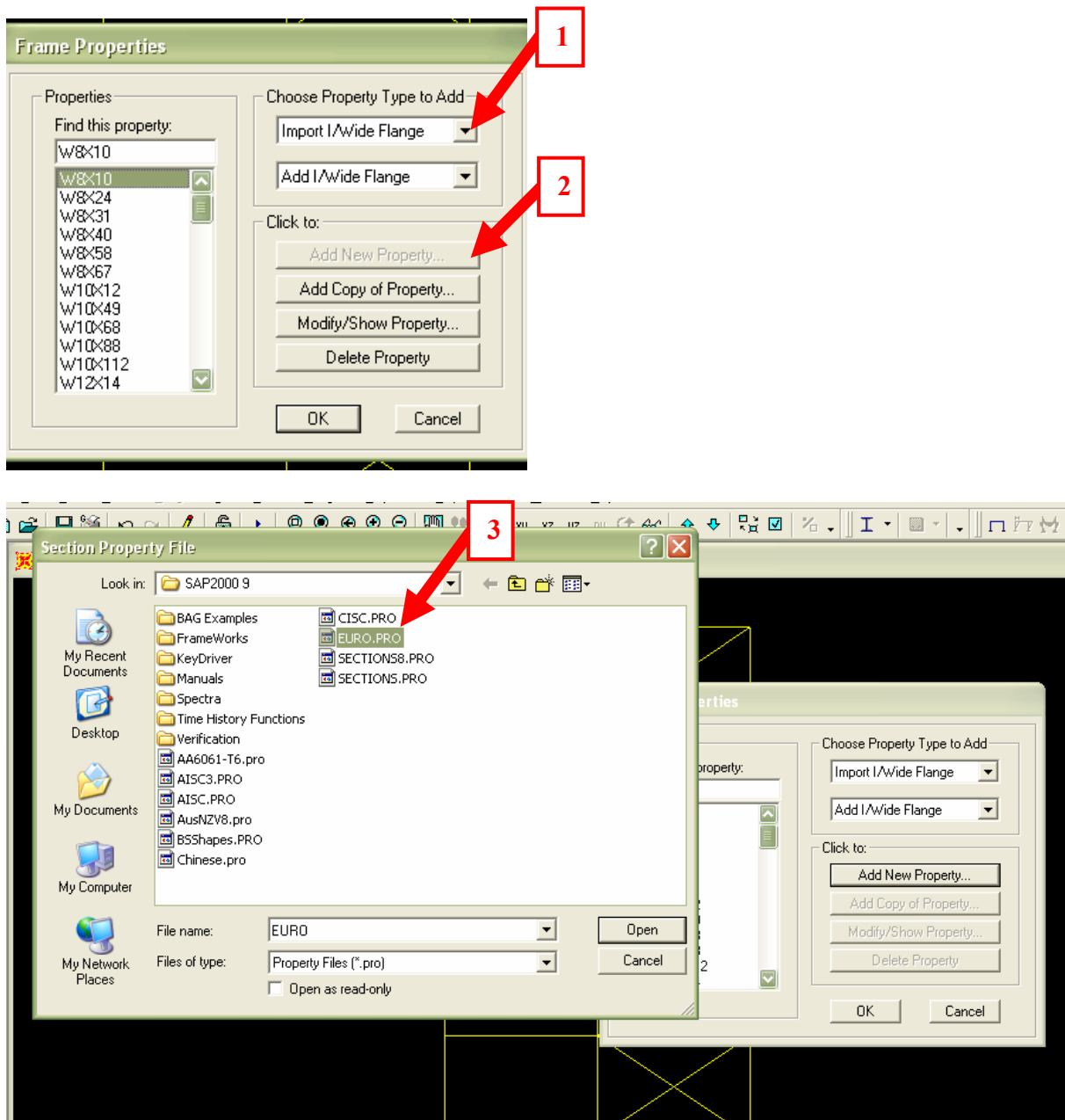
### 3. Definirea materialelor: **Define** → **Materials**

The image shows the 'Define Materials' dialog box in SAP2000. The dialog is divided into two main sections. The left section, titled 'Materials', contains a list of material types: ALUM, CLDFRM, CONC, OTHER, and STEEL. The 'STEEL' option is selected. Below the list are buttons for 'Add New Material...', 'Modify/Show Material...', and 'Delete Material'. A red arrow points to the 'Modify/Show Material...' button. The right section, titled 'Material Property Data', contains fields for 'Material Name' (STEEL), 'Type of Material' (Isotropic, Orthotropic, Anisotropic), 'Analysis Property Data' (Mass per unit Volume: 0, Weight per unit Volume: 0, Modulus of Elasticity: 2.1E+08, Poisson's Ratio: 0.3, Coefficient of Thermal Expansion: 1.170E-05, Shear Modulus: 76884615), 'Design Property Data (AISC-LRFD93)' (Minimum Yield Stress, Fy: 248211.28, Minimum Tensile Stress, Fu: 399896), and 'Advanced Material Property Data' (Time Dependent Properties..., Material Damping Properties..., Stress-Strain Curve Definitions...). A red arrow points to the 'Modulus of Elasticity' field. The dialog also includes 'OK' and 'Cancel' buttons.

Definim modulul de elasticitate pentru oțel

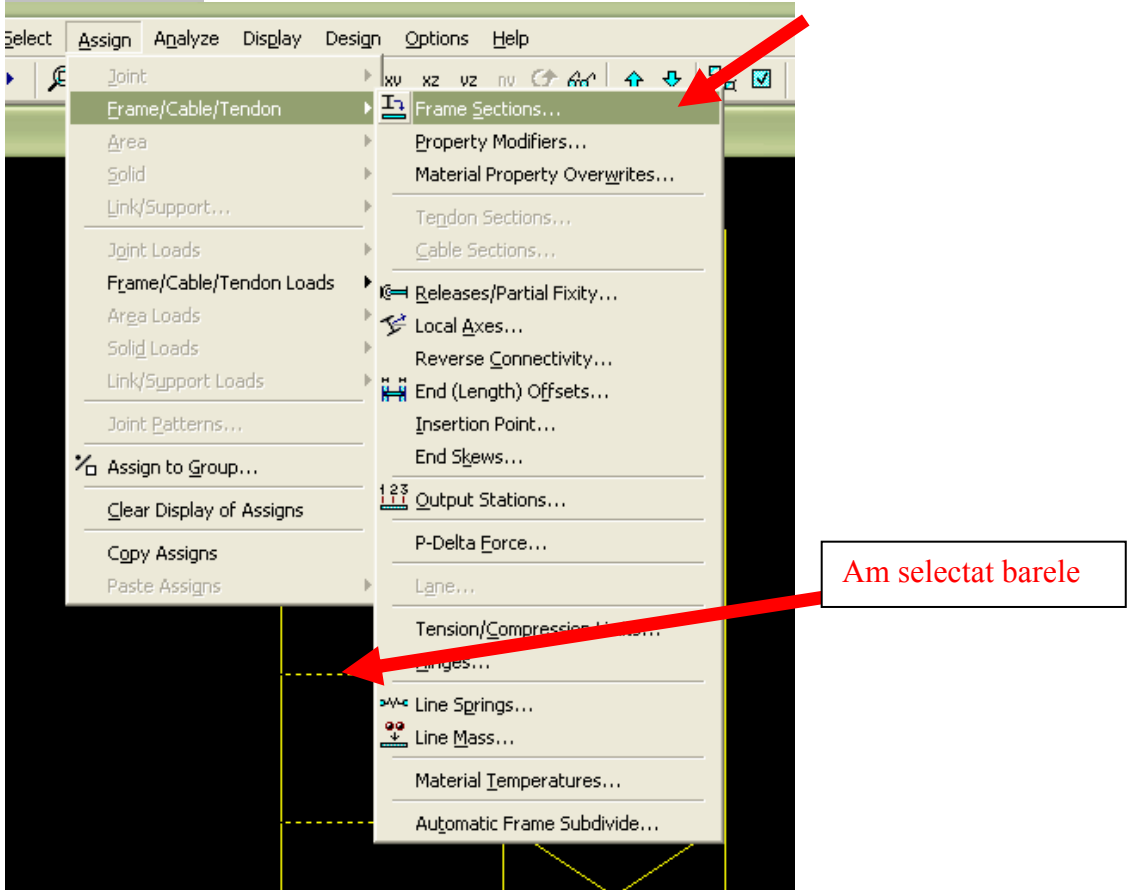
Masa și greutatea specifică le trecem cu 0.

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

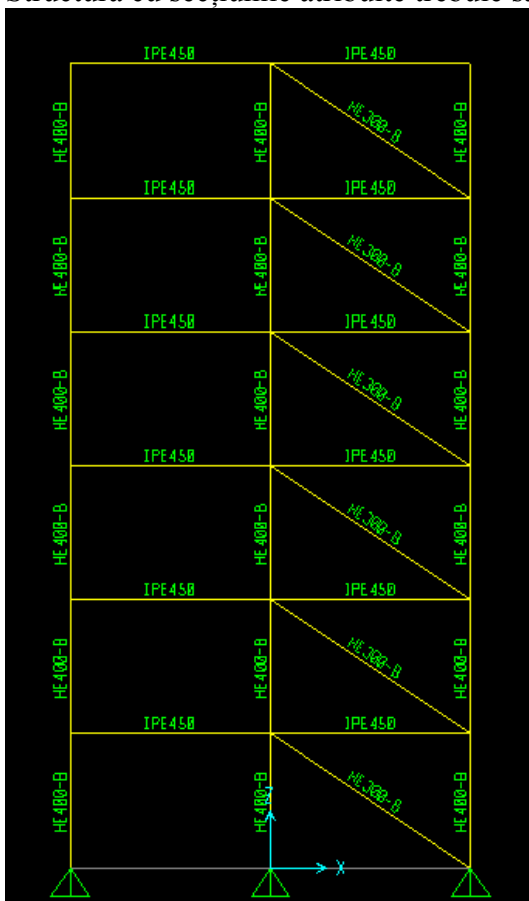


Alegem cu **CTRL** **HE 400B** pentru stâlpi, **IPE 450** pentru rigle și **HE 300B** pentru contravântuiri.

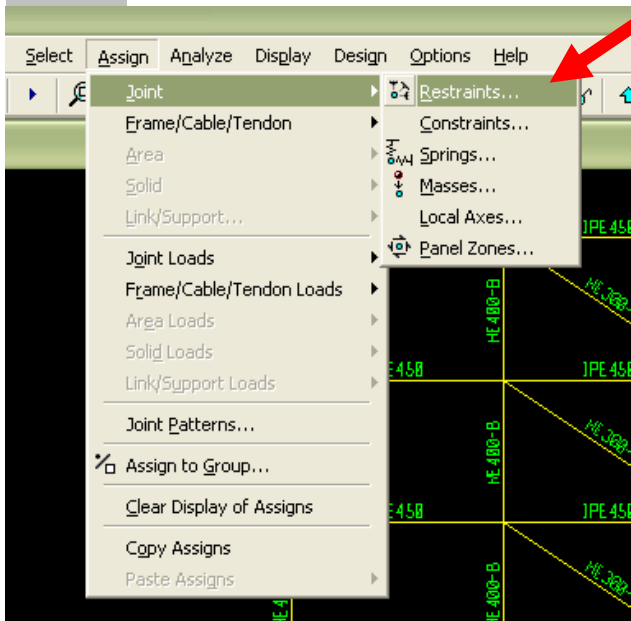
4. Atribuirea secțiunilor: selectăm barele și atribuim prin **Assign → Frame/Cable/Tendon→ Frame Section**



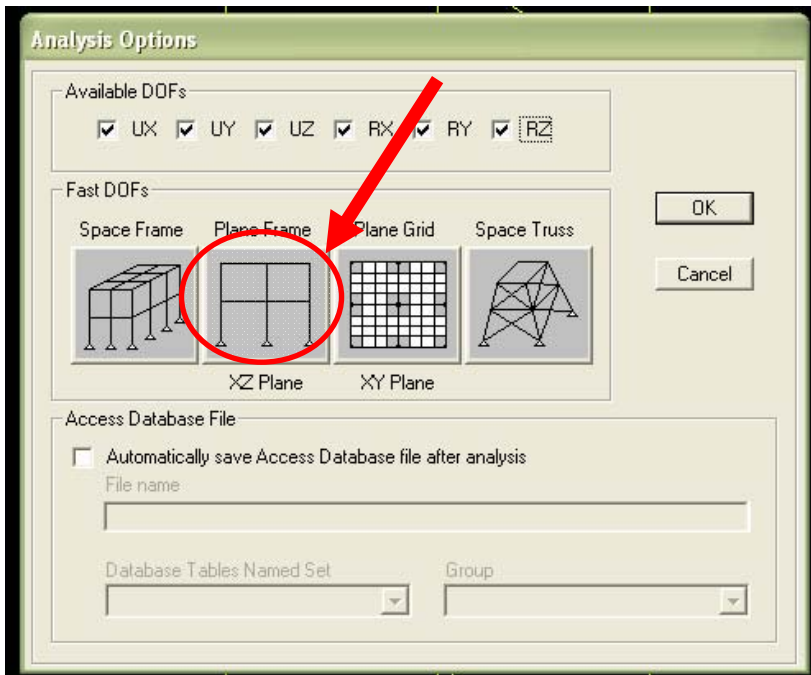
Structura cu secțiunile atribuite trebuie să arate în final așa:



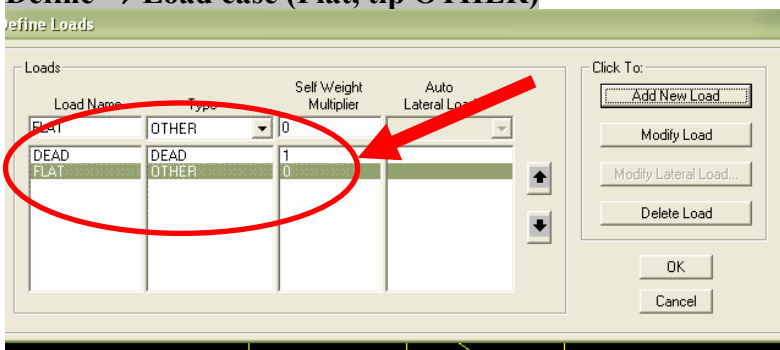
5. Atribuirea reazemelor: selectăm nodurile și atribuim reazeme blocate: **Assign → Joints → restrains**



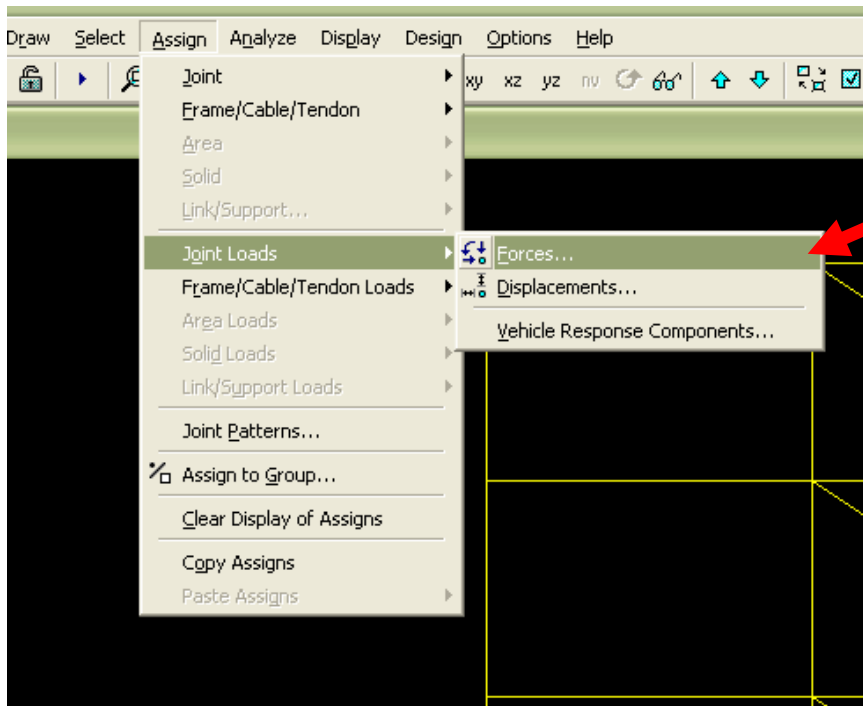
6. Definirea gradelor de libertate (structură plană): **Analyze → set analysis options → plane frame (XZ)**



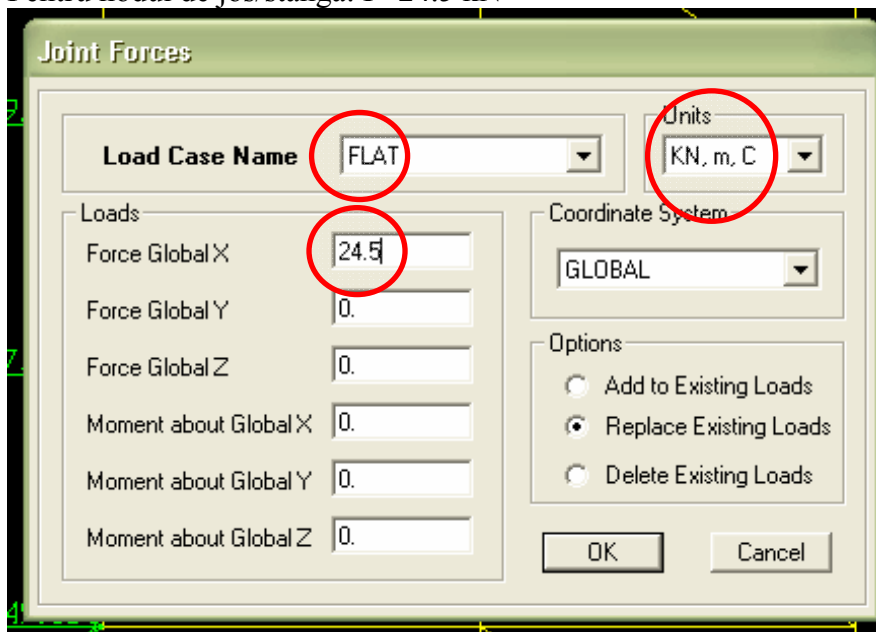
7. Definirea cazului de încărcare statică: **Define → Load case (Flat, tip OTHER)**



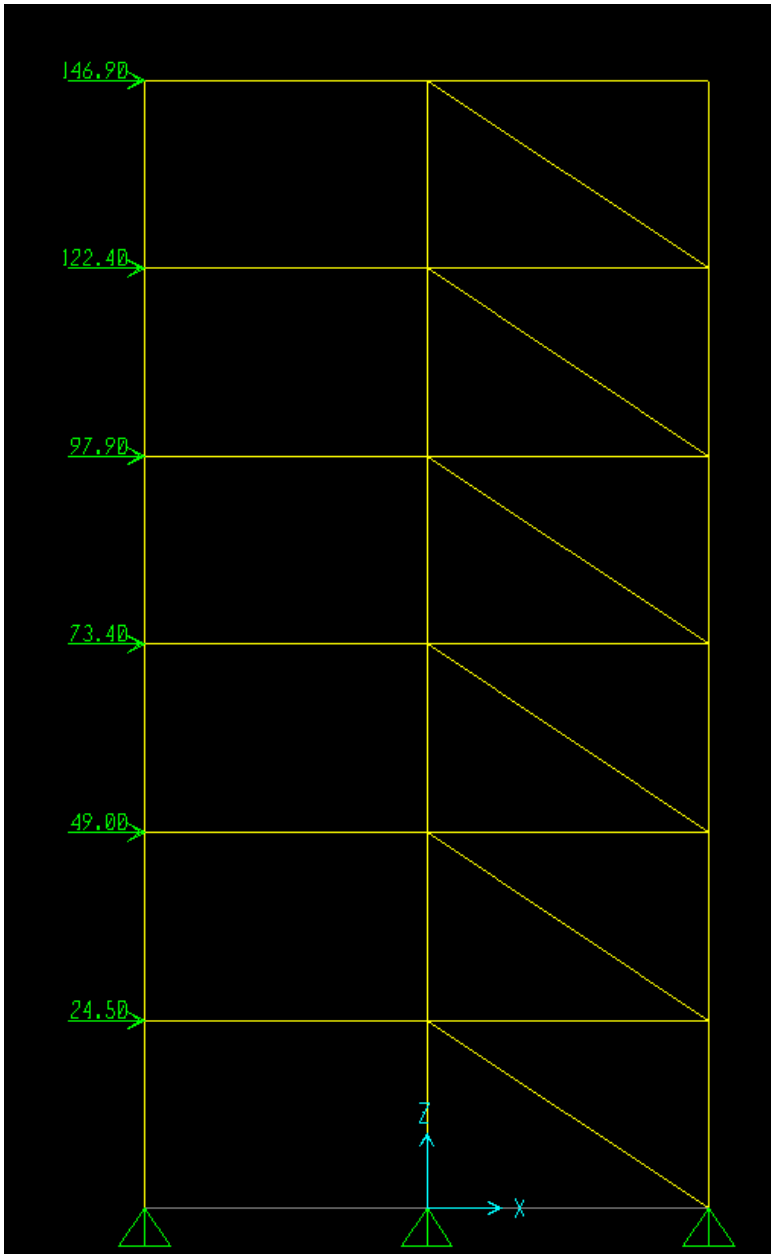
8. Definirea încărcări în noduri (forțe laterale): selectare nod  
**Assign → Joint Loads → Forces** (în cazul de încărcare FLAT)



Pentru nodul de jos/stânga:  $F=24.5$  kN

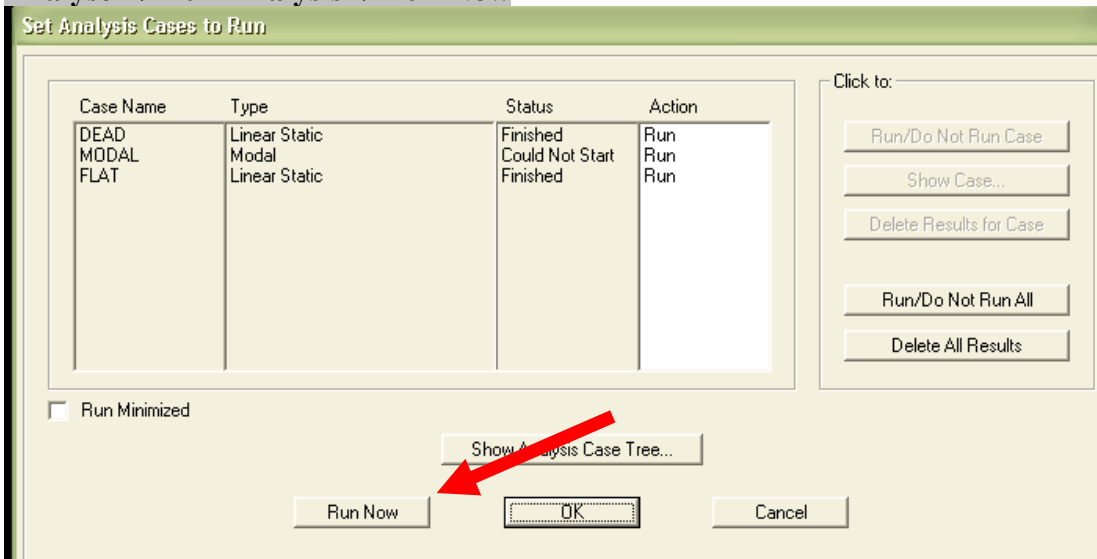


Structura cu forțele aplicate va fi:



9. Efectuarea calculului:

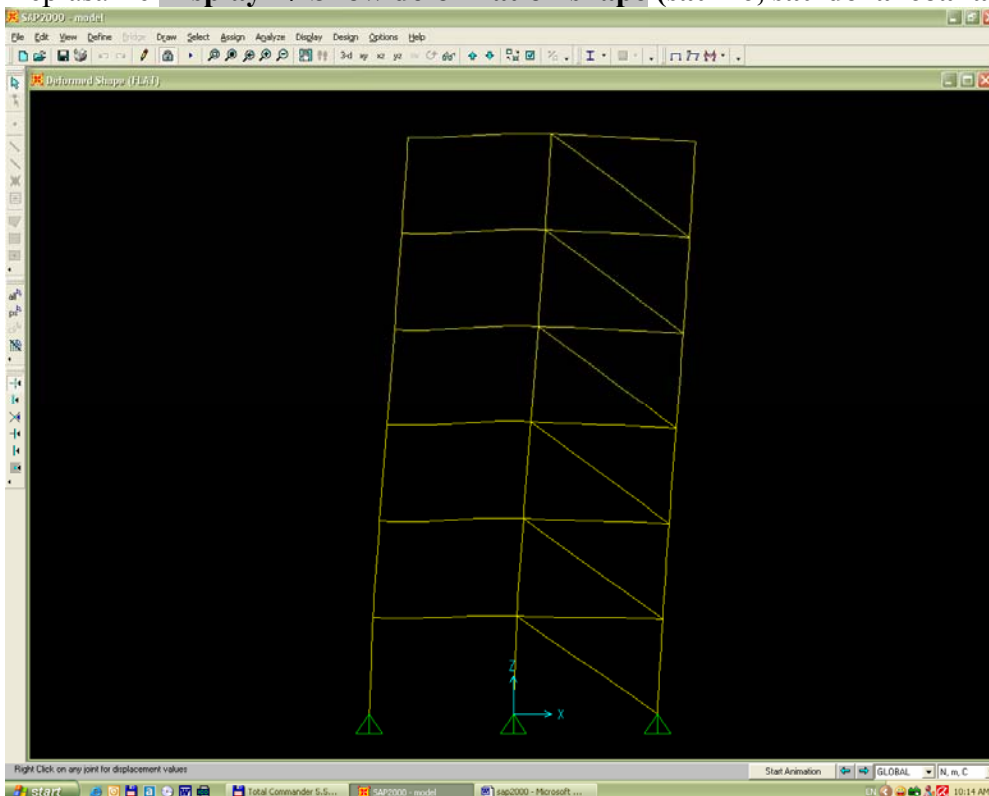
**Analyse → Run Analysis → Run Now**





10. Vizualizarea rezultatelor:

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



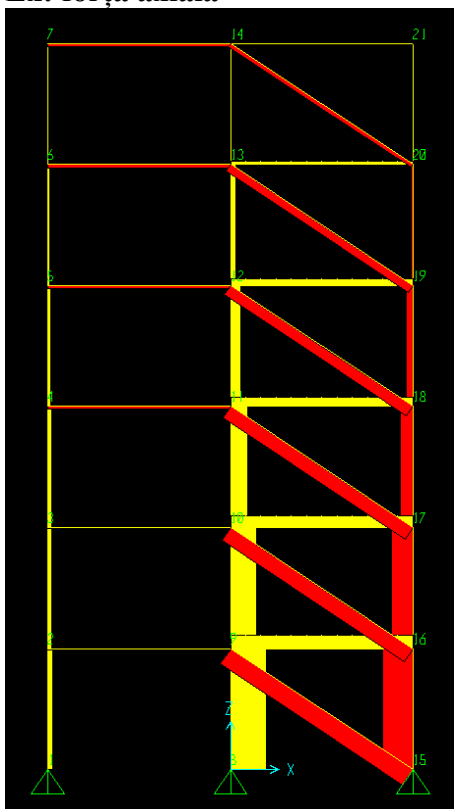
Eforturile: **Display** → **Show forces/stresses** → **Frames/Cables**

Pentru moment: **Moment 3-3**

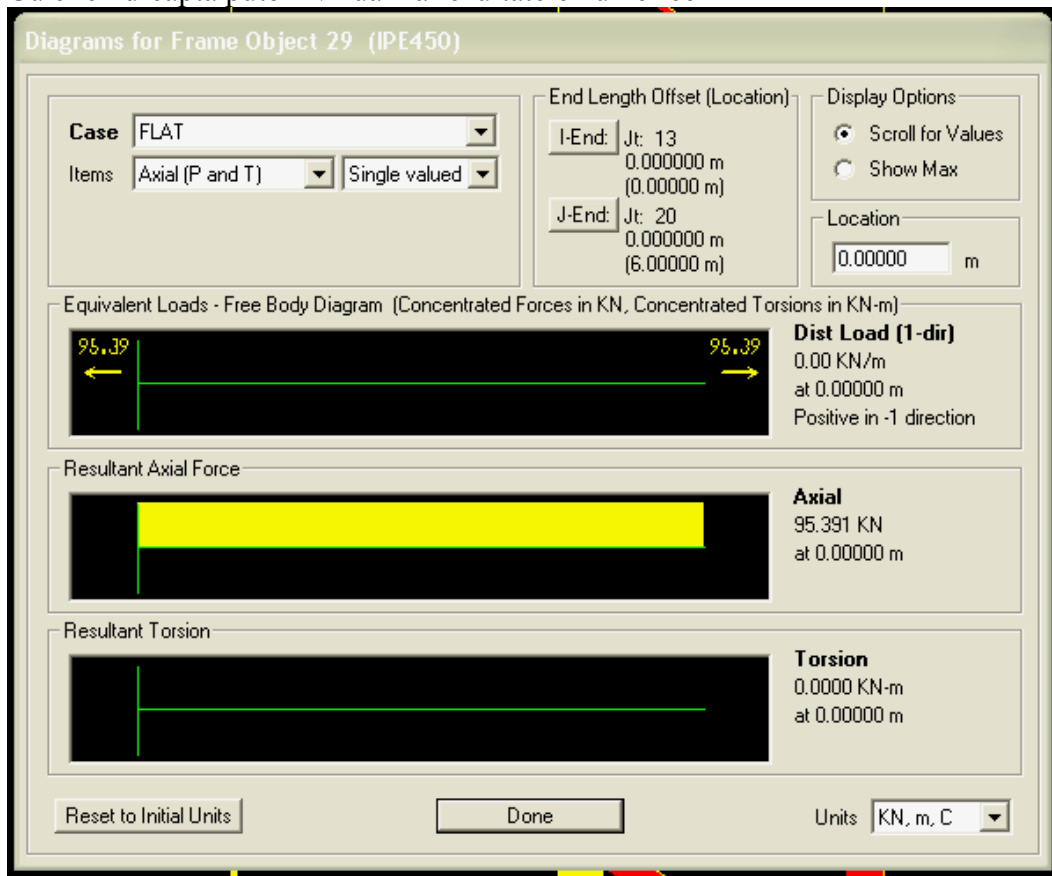
Pentru forța axială: **Axial force**

Pentru forța tăietoare: **Shear 2-2**

**Ex: forța axială**

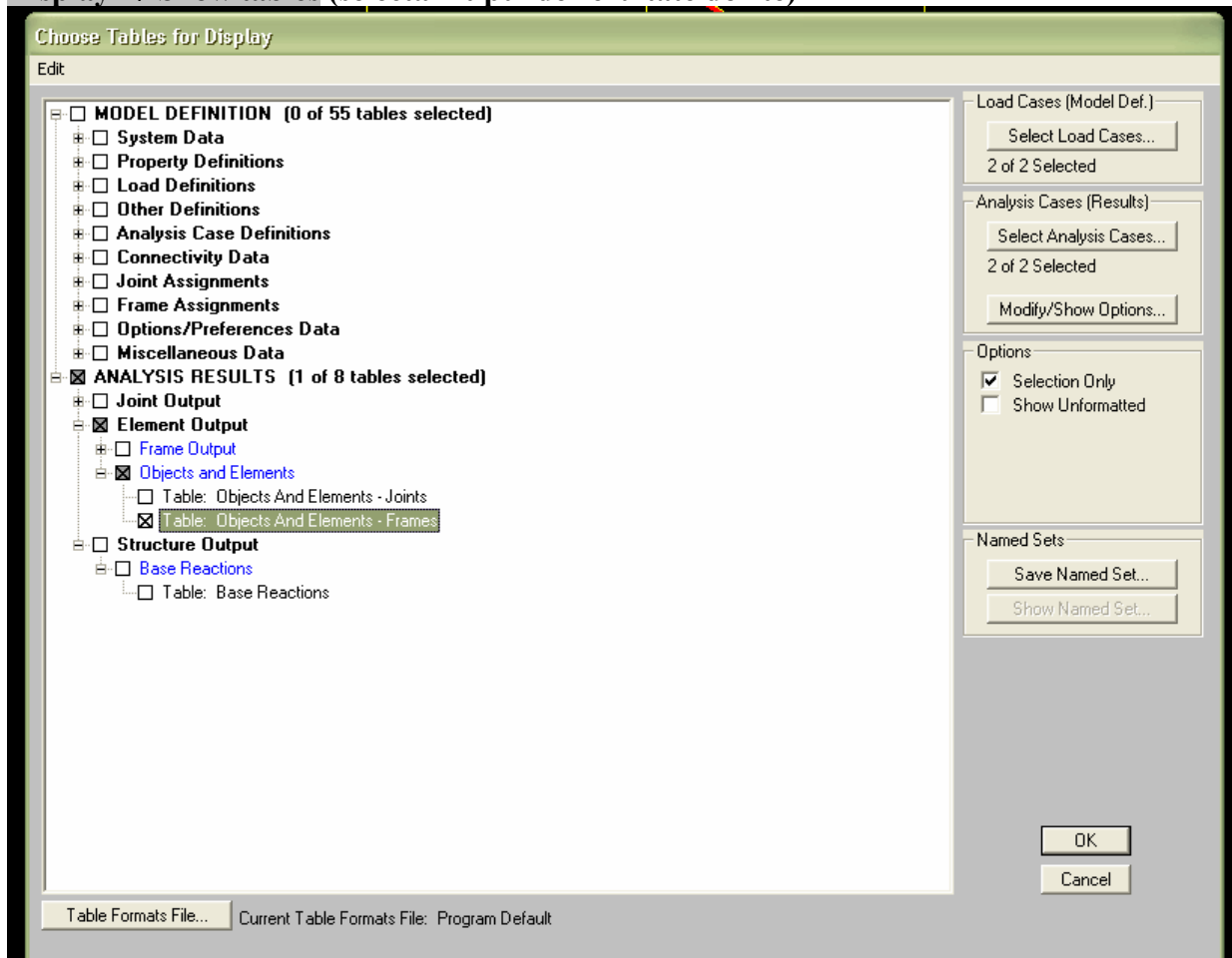


Cu click dreapta putem vizualiza rezultatele numerice



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 **Assign → Joints → Masses**

A screenshot of the 'Joint Masses' dialog box. It has a title bar 'Joint Masses'. The 'Mass Direction' section has a 'Coordinate System' dropdown set to 'Joint Local'. The 'Masses in Local Directions' section has three input fields: 'Direction 1' with '53.3', 'Direction 2' with '0.', and 'Direction 3' with '0.'. The 'Mom. of Inertia in Local Directions' section has three input fields: 'Rotation about 1', 'Rotation about 2', and 'Rotation about 3', all with '0.'. The 'Options' section has three radio buttons: 'Add to Existing Masses', 'Replace Existing Masses' (which is selected), and 'Delete Existing Masses'. At the bottom are 'OK' and 'Cancel' buttons.

- definire analiză modală

**Define → Analysis Cases → Modal**

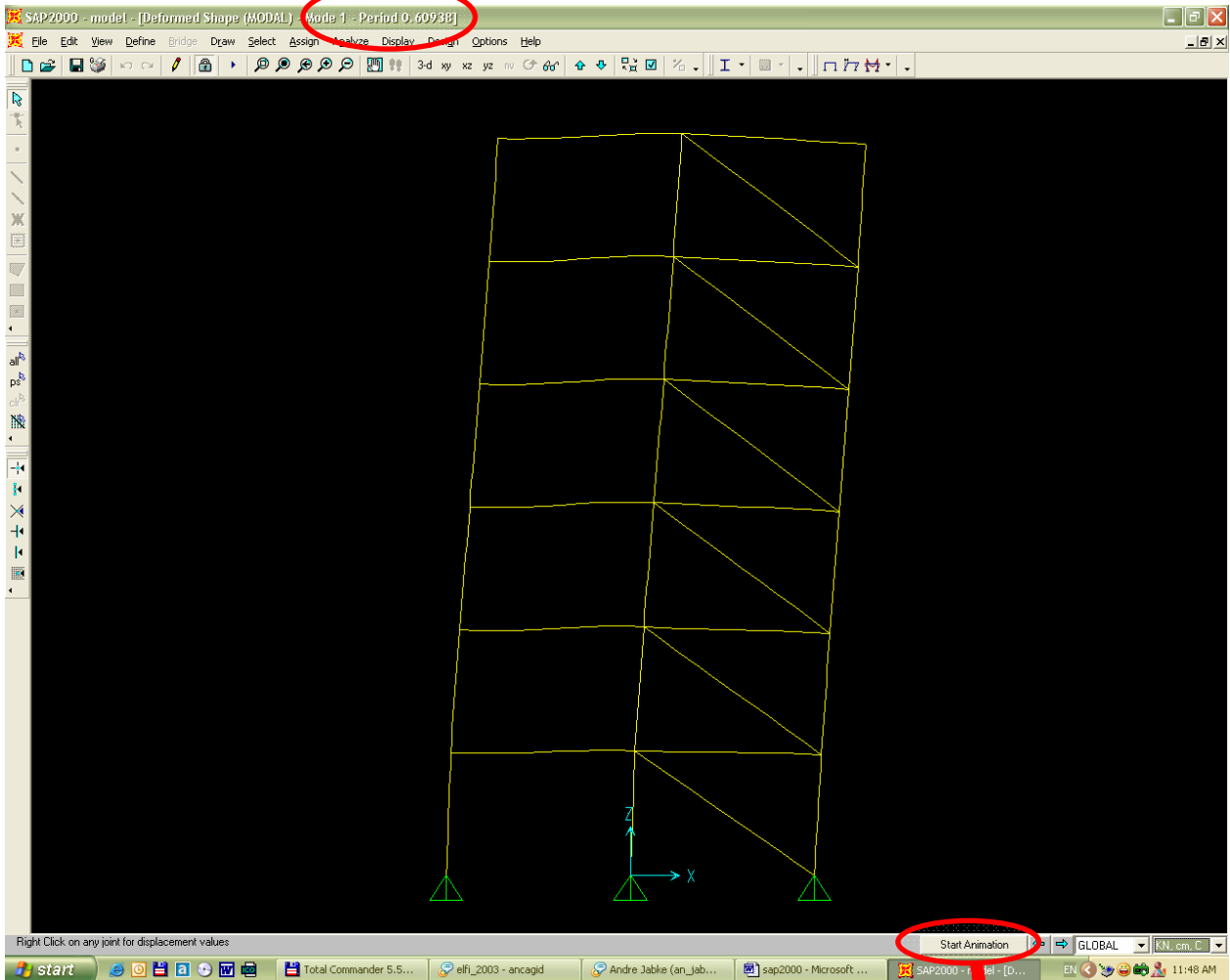
A screenshot of the 'Analysis Case Data - Modal' dialog box. The 'Analysis Case Name' is 'MODAL' and there is a 'Set Def Name' button. The 'Analysis Case Type' dropdown is set to 'Modal'. The 'Stiffness to Use' section has 'Zero Initial Conditions - Unstressed State' selected. The 'Type of Modes' section has 'Eigen Vectors' selected. The 'Number of Modes' section has 'Maximum Number of Modes' set to '12' and 'Minimum Number of Modes' set to '1'. The 'Loads Applied' section has 'Show Advanced Load Parameters' unchecked. The 'Other Parameters' section has 'Frequency Shift (Center)' set to '0.', 'Cutoff Frequency (Radius)' set to '0.', and 'Convergence Tolerance' set to '1.000E-07'. At the bottom right are 'OK' and 'Cancel' buttons. Two red arrows point to the 'Analysis Case Name' field and the 'Maximum Number of Modes' field.

Efectuare analiză: **RUN** 

Vizualizarea rezultatelor:

ODAL) - Mode 1 - Period 0.60938]

ex.: deplasări



modelul poate fi și animat



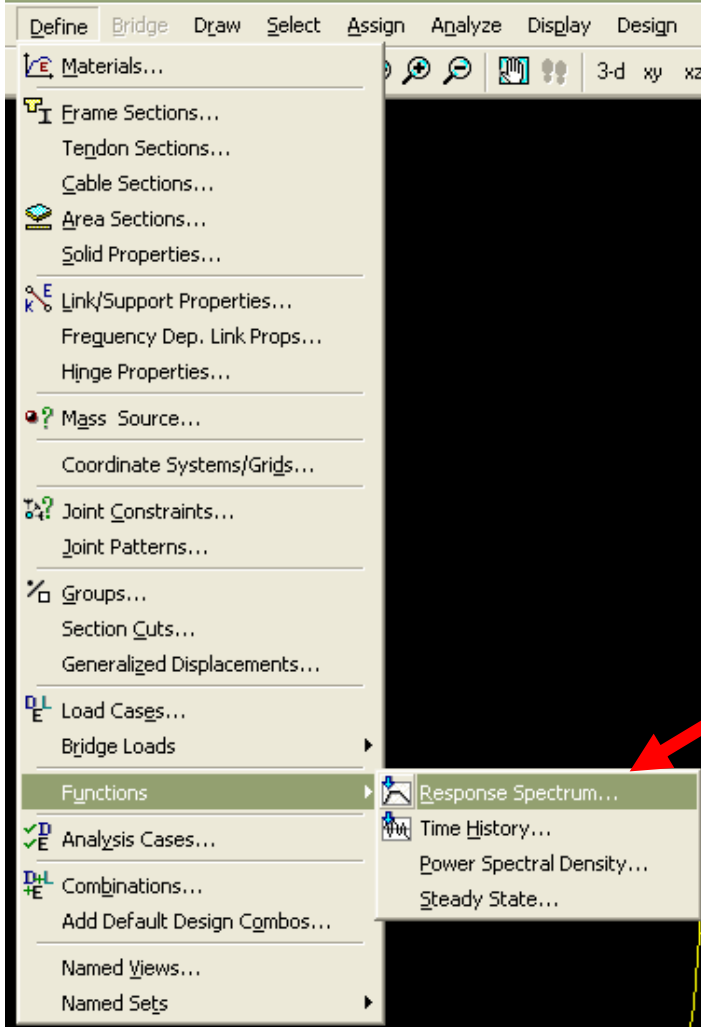
## ANALIZA SPECTRALĂ

Întâi deblocăm modelul de la

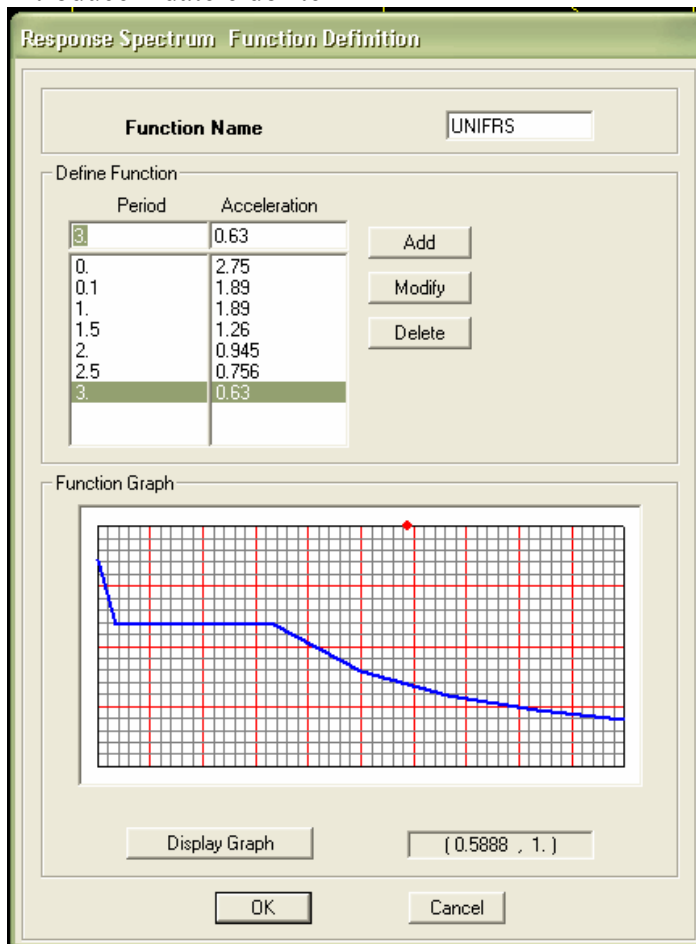


Definirea spectrului

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

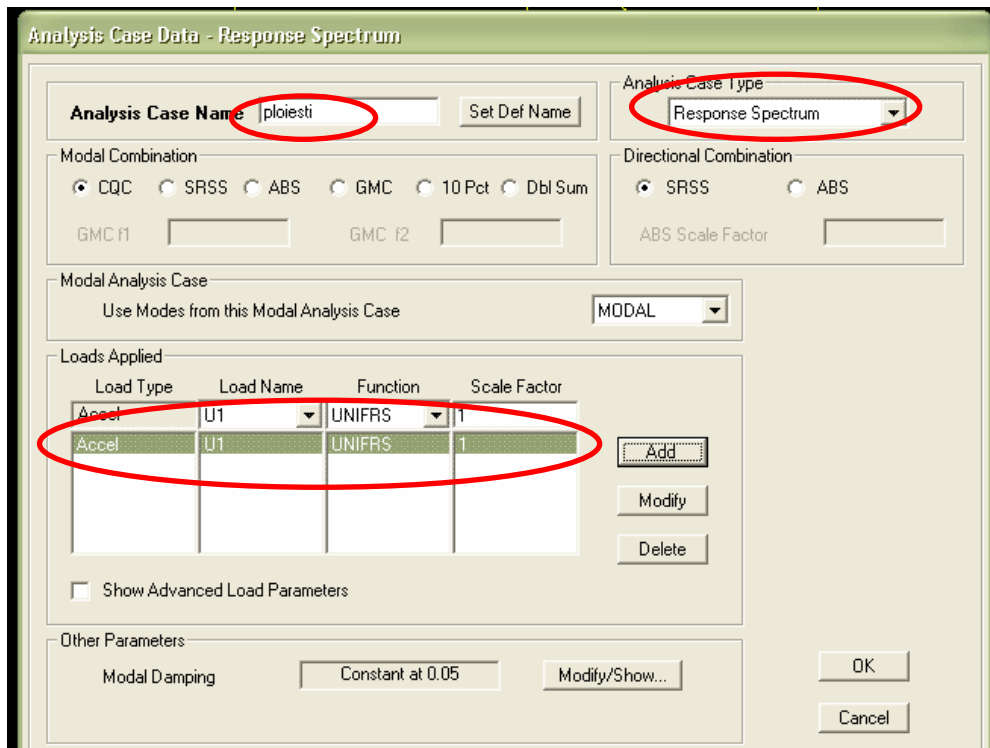


Introducem datele dorite



Definim calculul spectral

**Define → Analysis Cases → Add new → Respons Spectrum**



Efectuare analiză: **RUN**



Și vizualizăm după metodele descrise mai sus