

Civil**FEM**[®]

INTRO ESSENTIAL

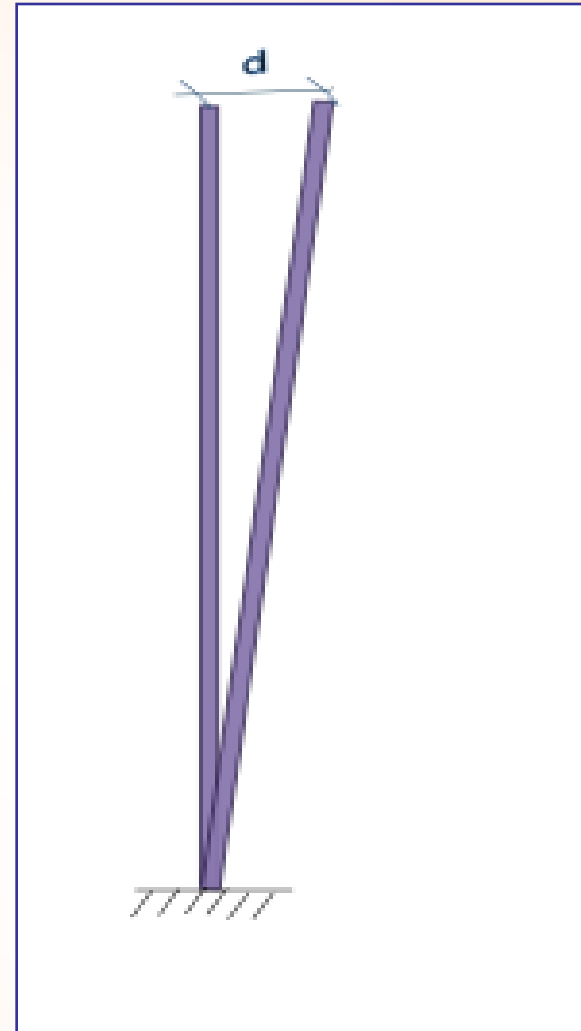
Practice 19 – Push-Over Analysis
of a Cantilever Beam

PUSH OVER APPLIED TO A CANTILEVER BEAM

The example provides an illustration of a push-over analysis applied to a cantilever beam.

In Push-Over Analysis, a load is increased up to the collapse of the structure.

Loading is applied controlling displacements instead of forces because the plasticity process converges better.



Given

Material Properties:

- S355 steel

Element Type:

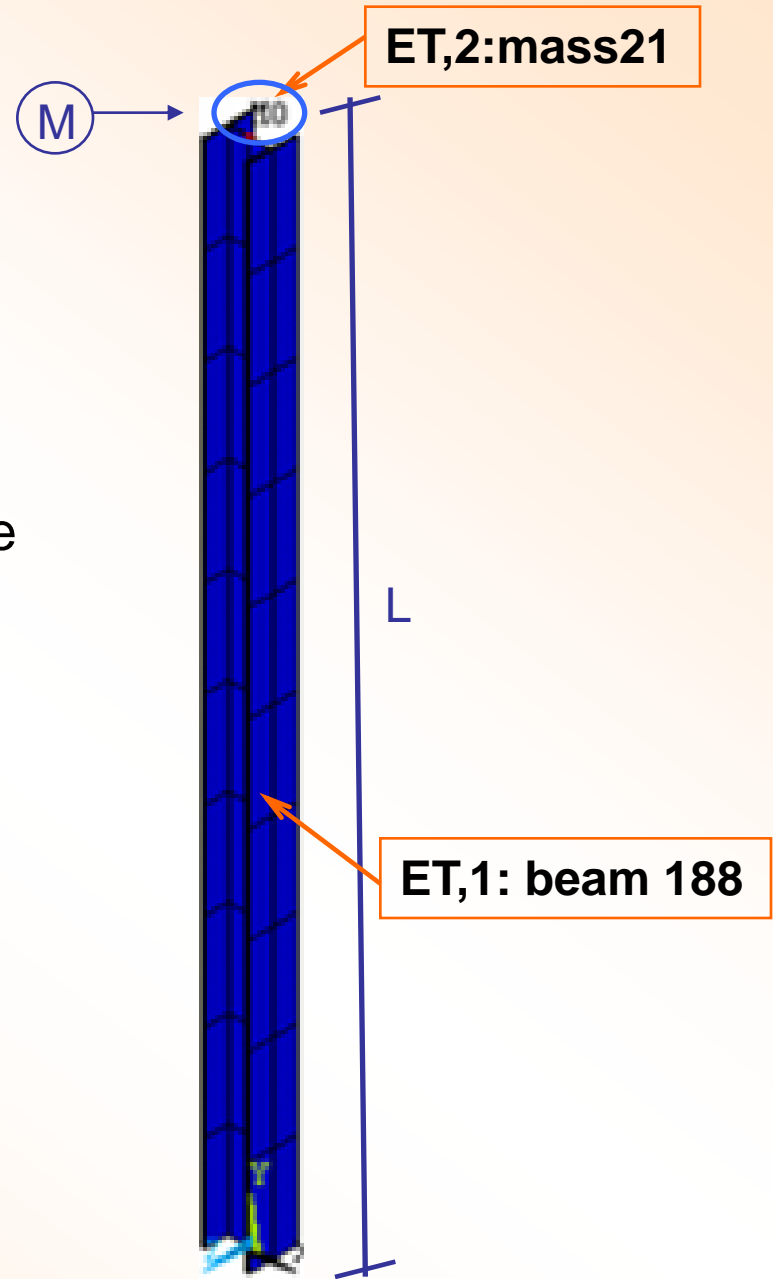
- BEAM188
- MASS21 for a mass located at the end of the beam (element 11)

Geometric Properties:

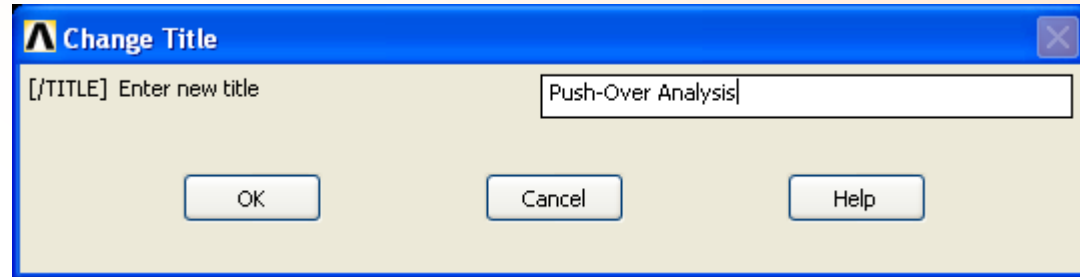
- $L = 4 \text{ m}$
- $M = 1500\text{kg}$
- Section: HE A 100

Imposed displacement:

- $d = 0.1 \text{ m}$



1. Enter the title: Push-Over Analysis
Utility Menu> File> Change Title...



Or issue:

`/TITLE, Push-Over Analysis`

2. CivilFEM setup: Code & Units

- Main Menu> Civil Setup
 - Select International System units.
 - Select codes: EA (steel), EHE (concrete, not used) and NCSE02 (seismic)

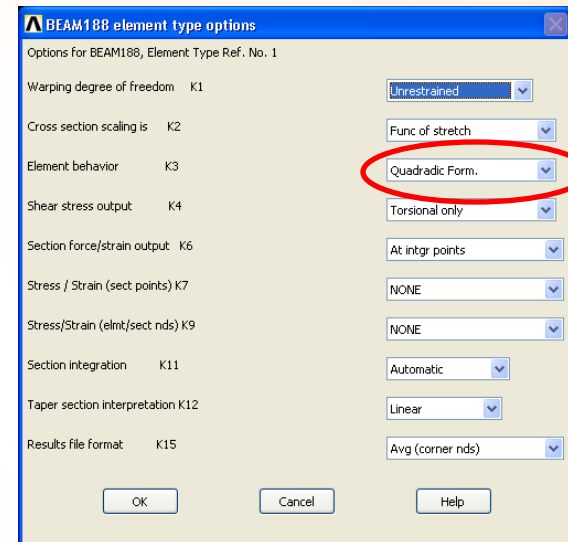
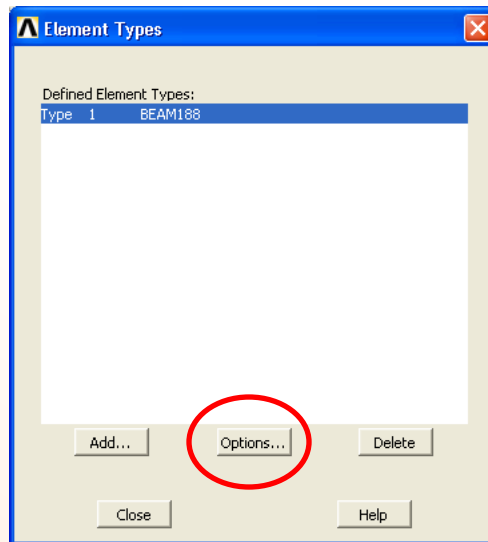
Or issue:

`~UNITS, SI`

`~CODESEL, EA, EHE-98, EHE-98, , NCSE02`

3. Define the element type.

- Add BEAM188 Element : Main Menu> Civil Preprocessor> Element types> Other Elements > Add/ Edit/ Delete > BEAM188
- Modify Keyoption 3 to quadratic form function:



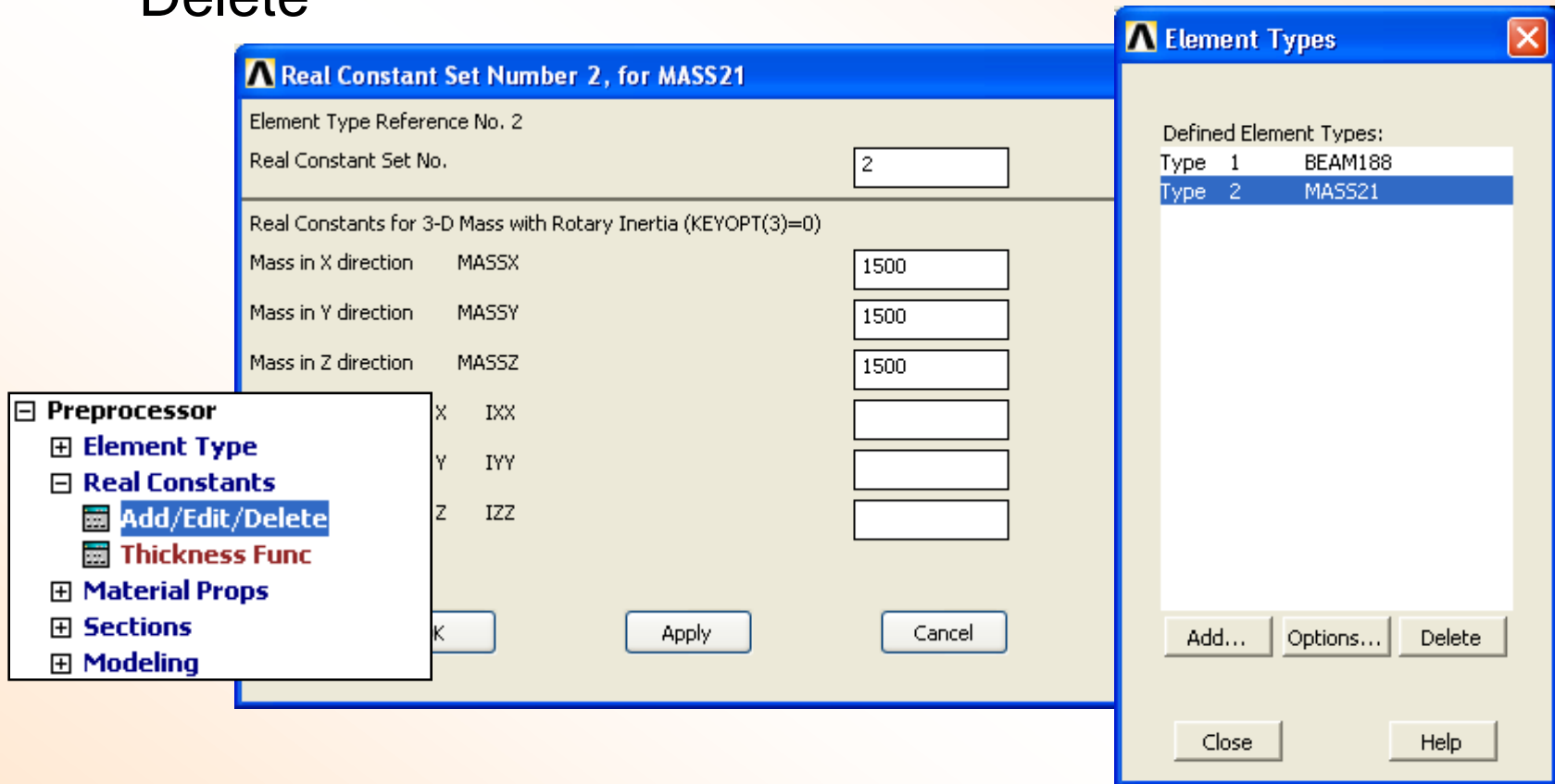
- Add MASS21 Element: Main Menu> Civil Preprocessor> Element types> Other Elements > Add/ Edit/ Delete > MASS21

Or issue:

```
ET, 1, BEAM188, , , 2 $ ET, 2, MASS21
```

4. Define real constant for the mass.

- Main Menu > Preprocessor > Real Constants > Add/ Edit/ Delete

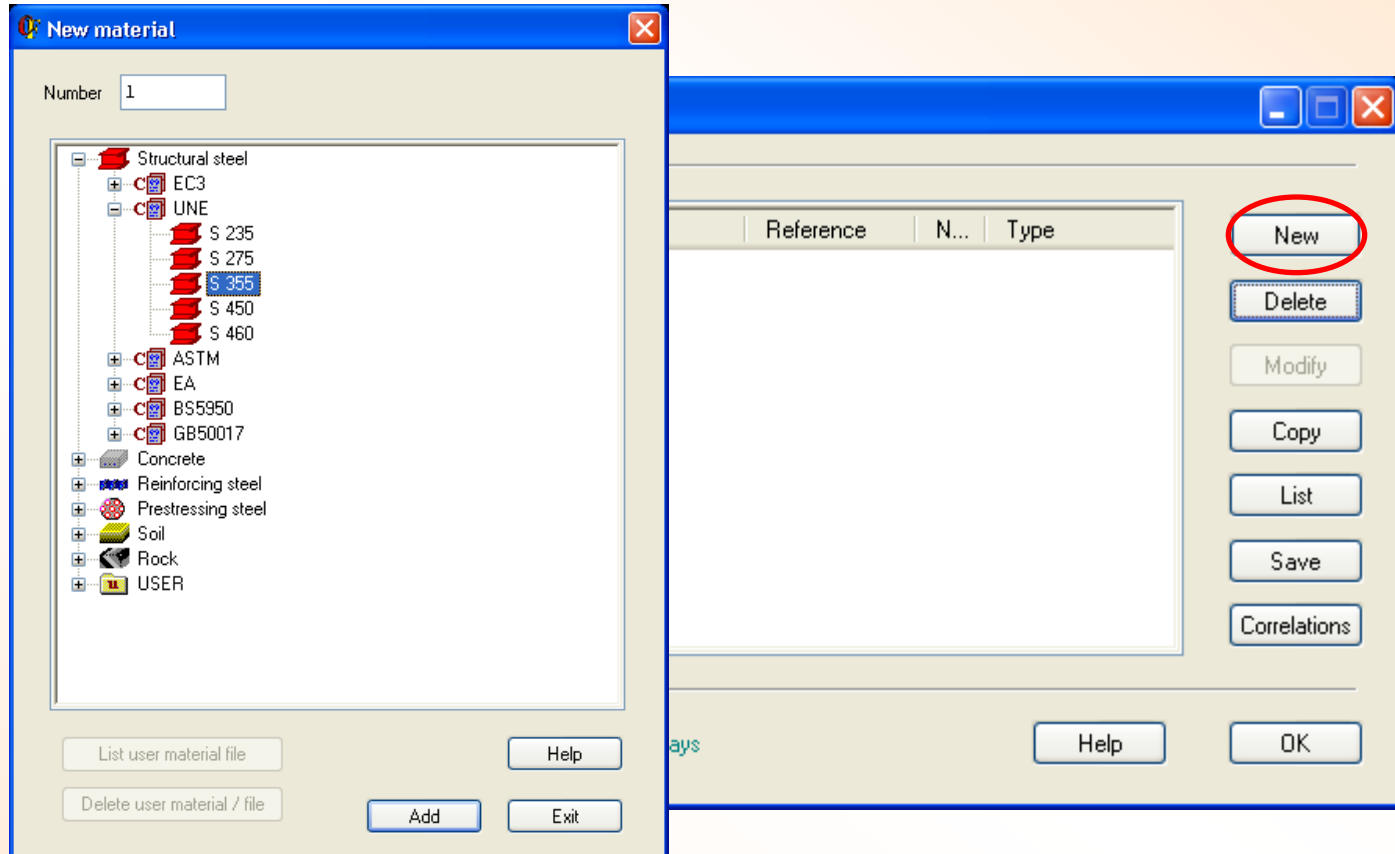


Or issue:

R, 2, 1500, 1500, 1500

5. Then define the material.

- Main Menu > Civil Preprocessor > Materials



Or issue:

~CFMP, 1, LIB, STEEL, UNE, S 355

Bilinear Analysis Diagram for Plasticity

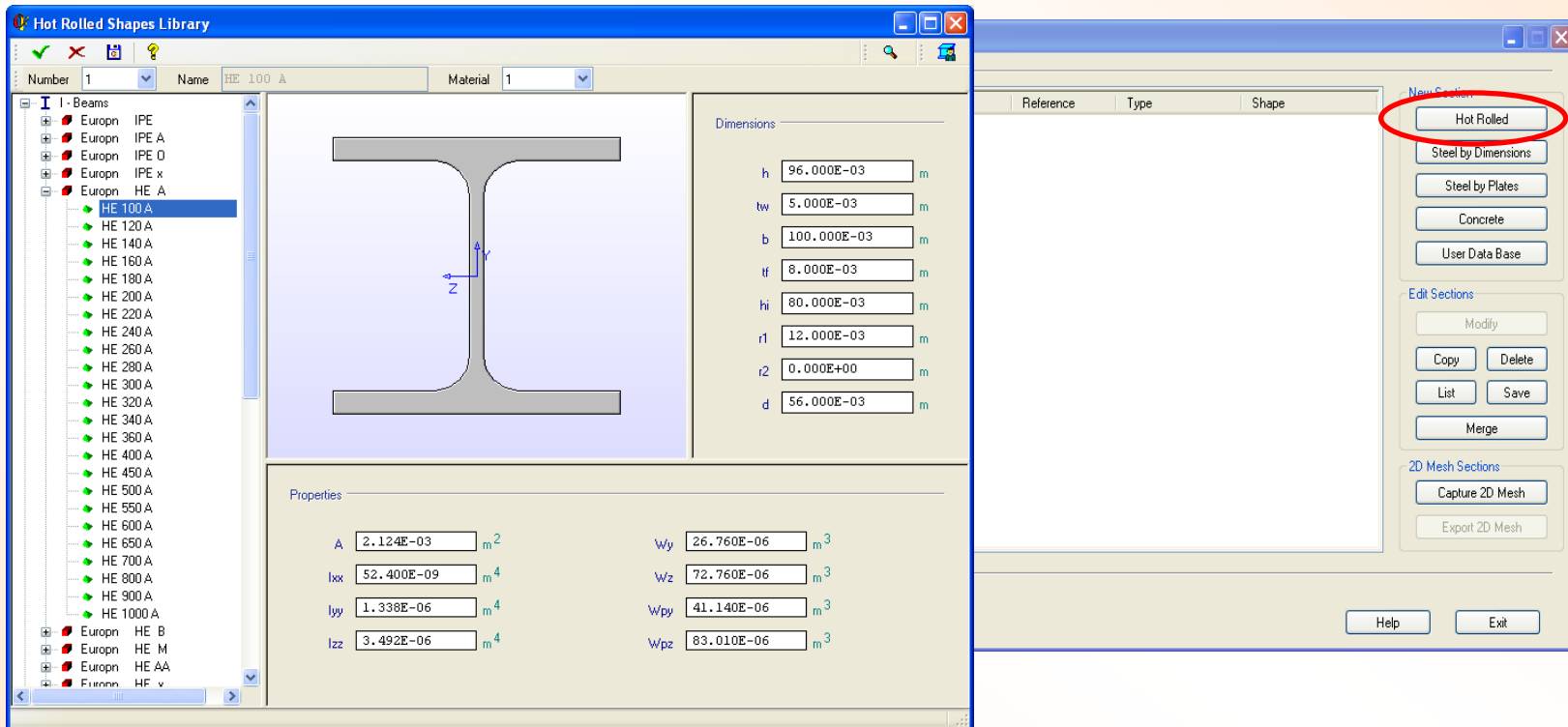
The image shows two overlapping windows from a software application. The primary window is titled "Material 1: Fe E355" and has several tabs: "General", "Analysis Diagram", "Design Diagram", "Steel", "EA", and "FLAC3D". The "Analysis Diagram" tab is active, displaying a stress-strain diagram for a bilinear material. The diagram shows a linear elastic region followed by a perfectly plastic plateau. The y-axis is labeled $\sigma (+)$ and the x-axis is labeled $\varepsilon (+)$. To the right of the diagram, there are input fields for "Point" (set to 4), "Strain" ($10.000E-03$), and "Stress" ($355.170E+06$ Pa). Below these are "Add point" and "Delete point" buttons. Further down, there are fields for "KPLA" (set to "1: BKIN - Bilinear kinemat") and "PLRAT" ($10.000E+03$). At the bottom of the dialog, there are "OK", "Apply", "Cancel", and "Help" buttons.

The second window, partially visible on the right, shows a table of material properties. The table has columns for "Reference", "N...", and "Type". The first row contains "S 355", "1", and "Structural steel". To the right of the table are buttons for "New", "Delete", "Modify", "Copy", "List", "Save", and "Correlations". The "Modify" button is circled in red. At the bottom of this window are "Help" and "OK" buttons.

Or issue:
~CFMP,1,Steel,TSASSD,,2

6. Select the cross section of the beam.

- Main Menu > Civil Preprocessor > Cross Sections (Select HEA100)

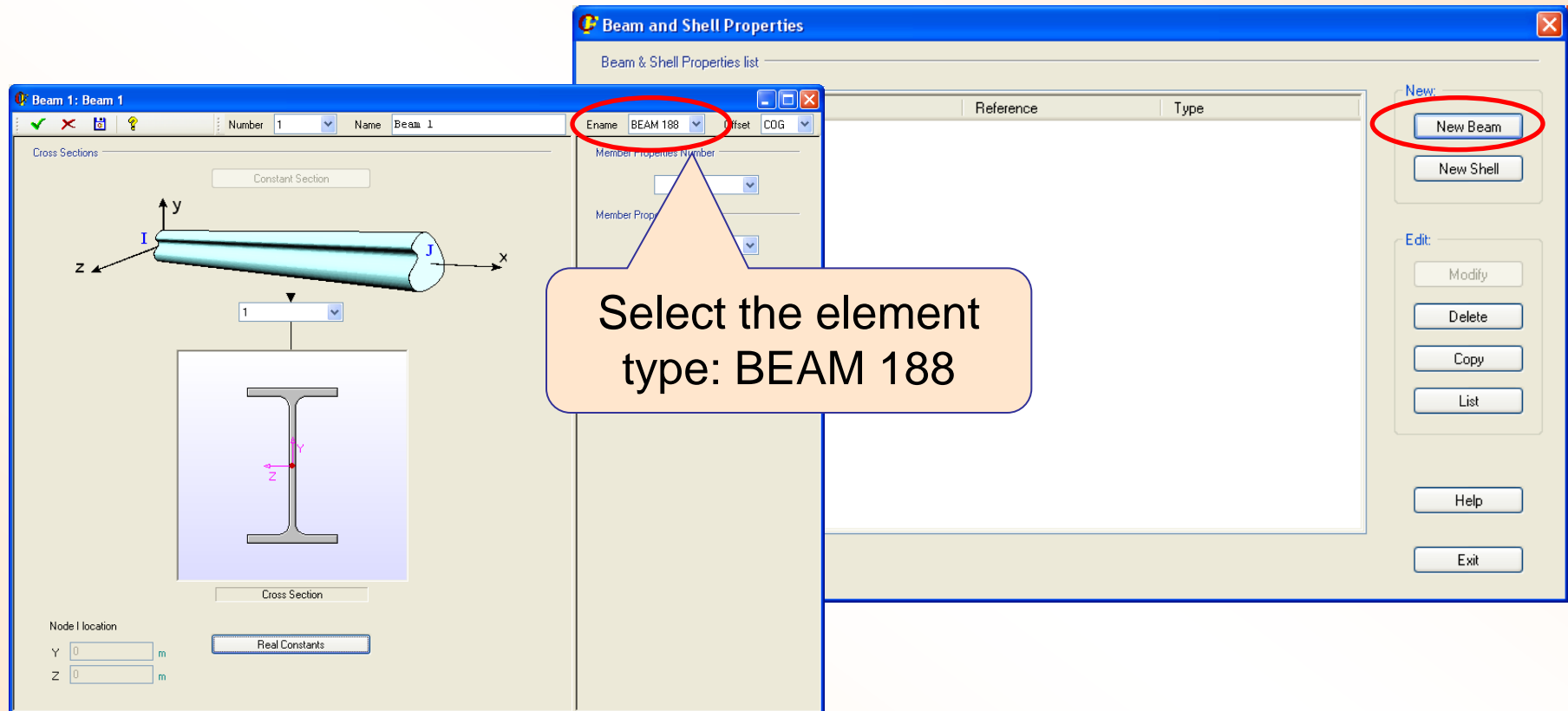


Or issue:

~SSECLIB, 1, 1, 5, 1

7. Define the Beam & Shell properties.

- Main Menu> Civil Preprocessor> Beam & Shell Properties



Or issue:

~BMSHPRO,1,BEAM,1,1,,,188,1,0,,Beam 1

8. Geometry Definition

First, the ending nodes are defined:

- Main Menu> Preprocessor> Modeling> Create> Nodes> In Active CS

Create Nodes in Active Coordinate System

[N] Create Nodes in Active Coordinate System

NODE Node number

X,Y,Z Location in active CS

THXY,THYZ,THZX
Rotation angles (degrees)

Or issue:

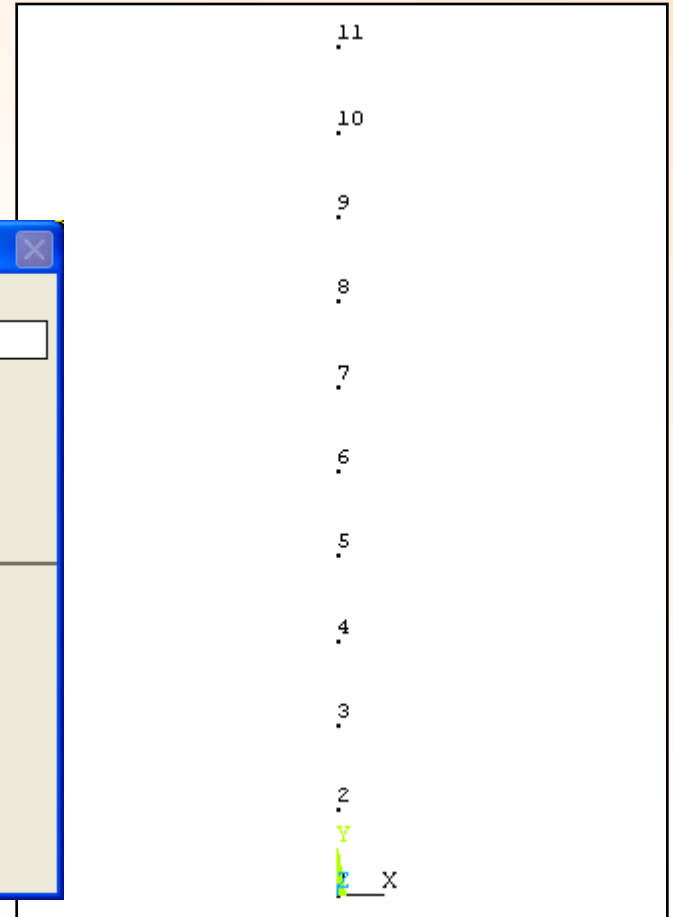
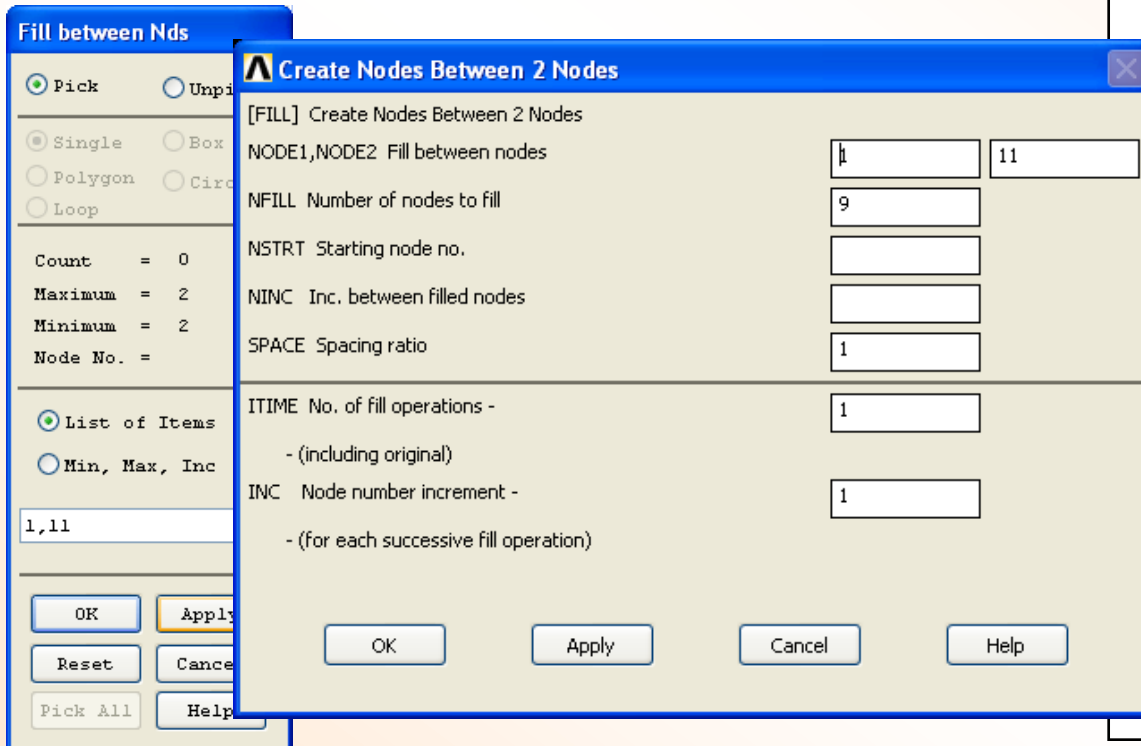
N, 1

N, 11, 0, 4

THXY,THYZ,THZX
Rotation angles (degrees)

- Main Menu> Preprocessor> Modeling> Create> Nodes> Fill between Nds

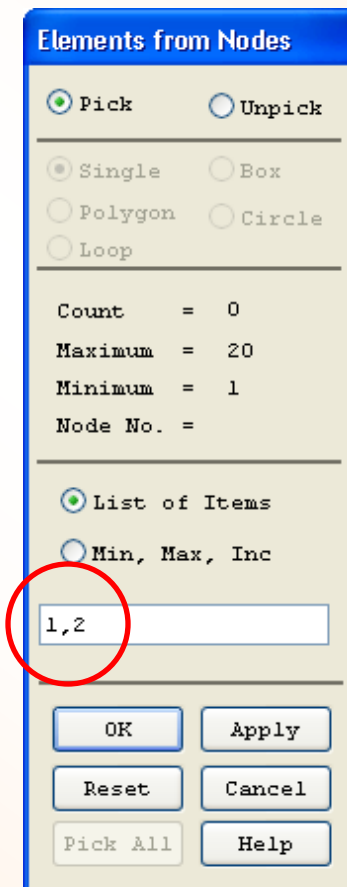
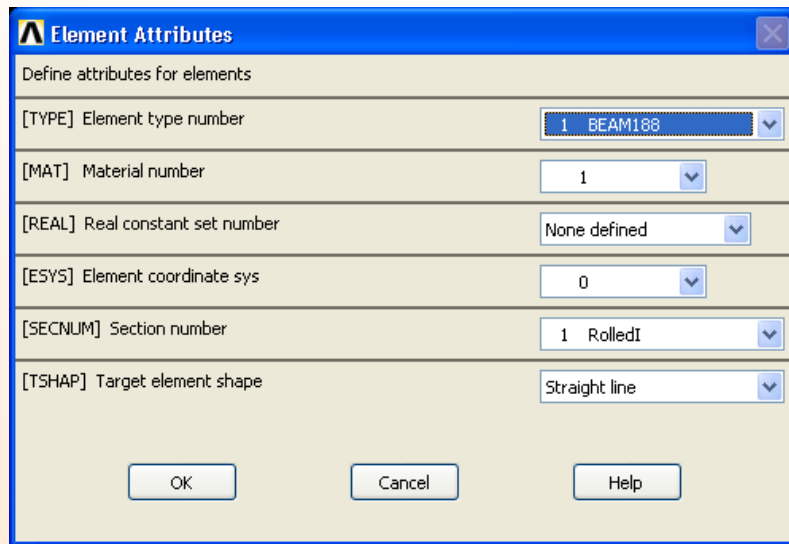
Fill nodes between node 1 and node 11.
(there are 9 nodes to fill)



Or issue:
FILL, 1, 11

9. Mesh the model.

- Main Menu> Preprocessor> Create> Elements > Elem Attributes (choose BEAM188, material 1, section 1)
- Main Menu> Preprocessor> Create> Elements> Auto Numbered> Thru Nodes :



Or issue:

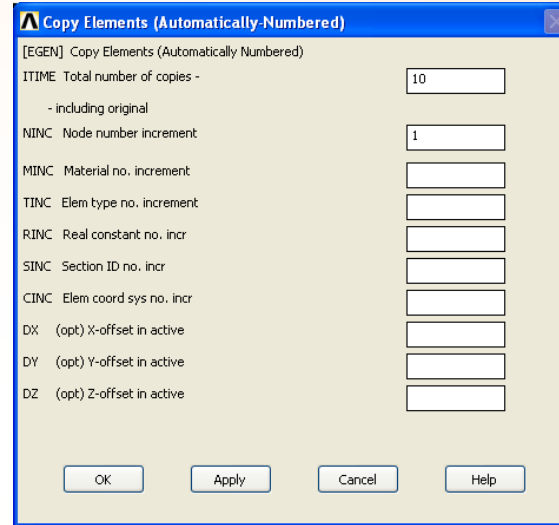
```
TYPE,1  
MAT,1  
SECNUM,1  
E,1,2
```

- Main Menu > Preprocessor > Copy > Elements > Auto Numbered

Copy elements:

Or issue:

`EGEN, 10, 1, ALL`



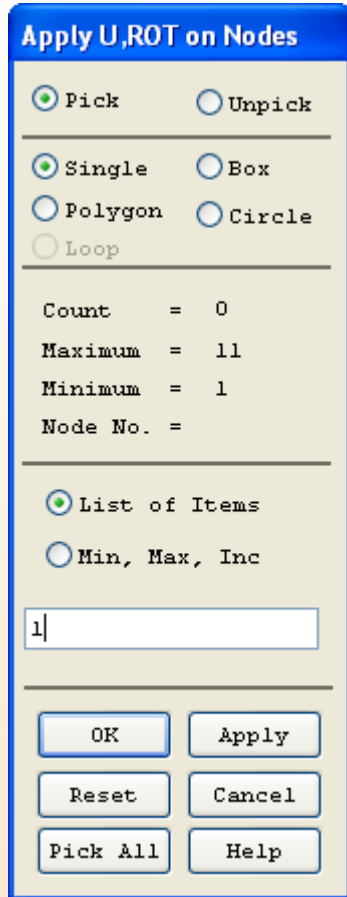
- Define mass element:

Issue:

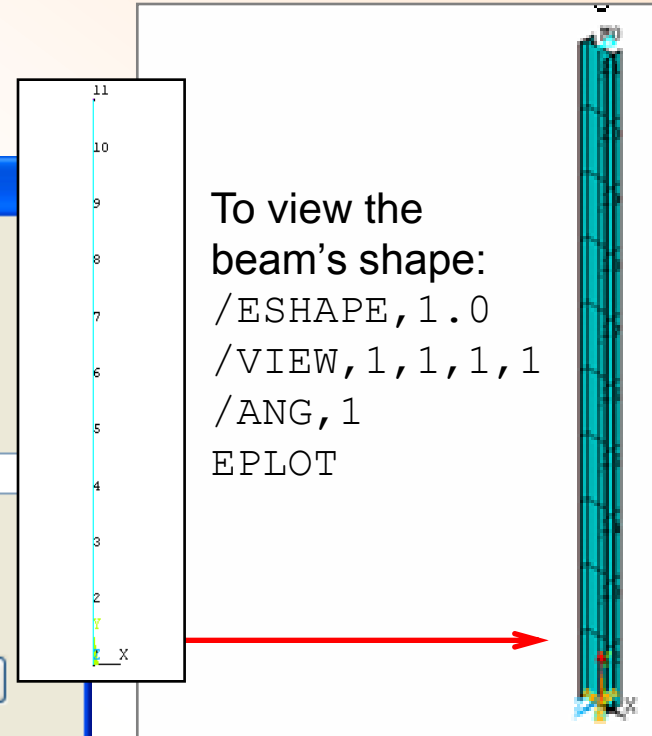
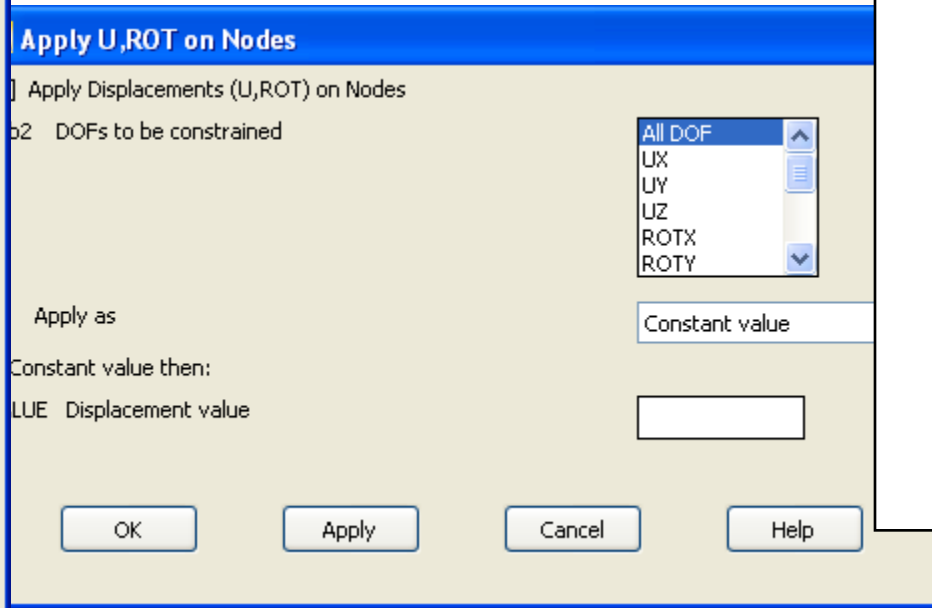
`TYPE, 2 $ REAL, 2 $ E, 11`

10. Boundary Conditions and Loads

- Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On nodes



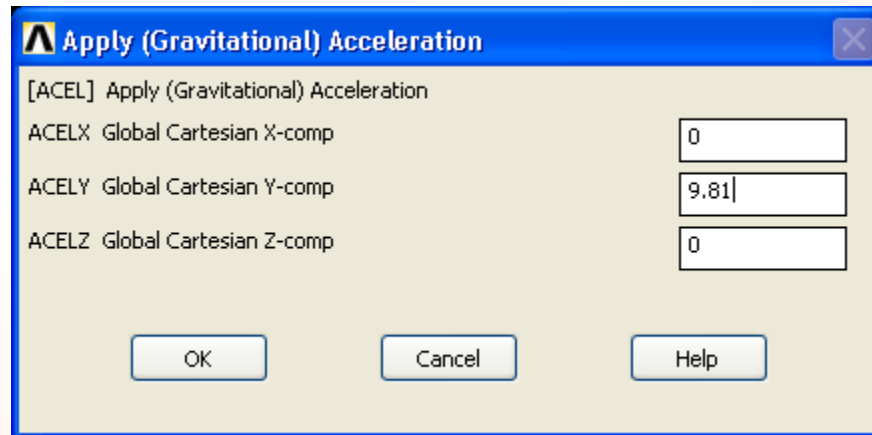
Constrain all displacements at Node 1.



Or issue:
/SOLU
D,1,ALL

- Main Menu> Solution> Define Loads> Apply> Structural> Inertia> Gravity >Global

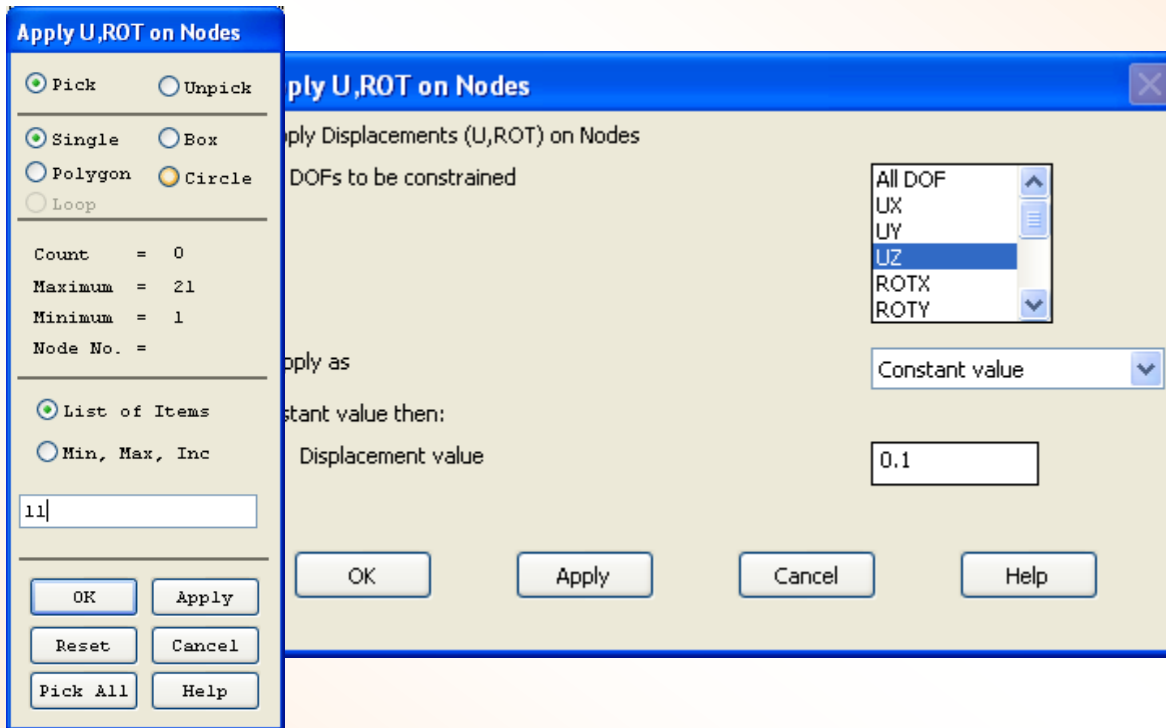
Define the Gravity (it must be input as positive).



Or issue:

ACEL, , 9.81

- Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

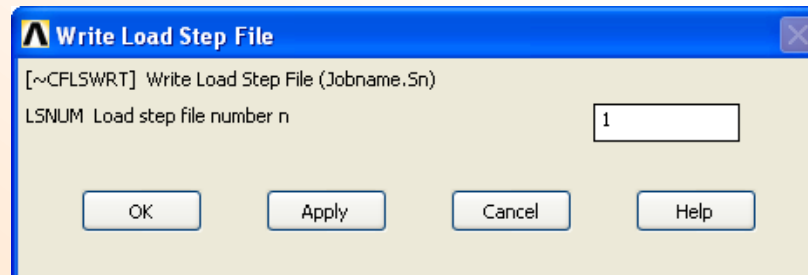


Displacement in Z
for Push-Over

Or issue:
D, 11, UZ, 0.1

Save in a
Load State File

- Main Menu > Civil Solution > Load State > Write LS File



Or issue:
~CFLSWRT, 1

11. Define Spectrum

- Main Menu > Seismic Design > Define Spectra > By code

The screenshot shows the 'Define spectrum' dialog box with the following parameters and callouts:

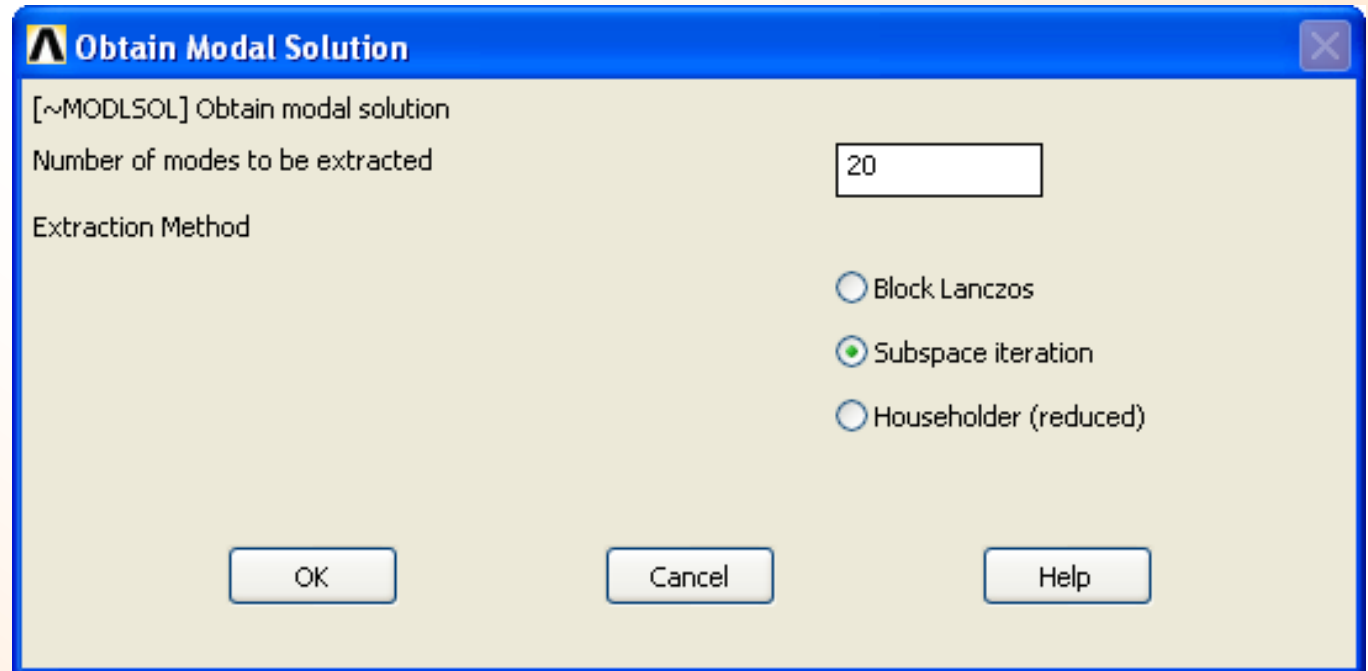
- Basic acceleration [ab/g] (Art. 2.1, Fig 2.1 and Ap 1)**: 0.4 (Callout: Basic acceleration)
- Spectrum type**: SPTYPE: ELASTIC (Callout: Spectrum Type)
- Risk non-dimensional coefficient (Art. 2.2)**: RD: 1 (Callout: Risk coefficient)
- Soil type (Art 2.3.1)**: C: TYPE I (Callout: Subsoil class)
- Contribution coefficient [K] (Art. 2.3 and Ap1)**: [K]: K: 1 (Callout: Contribution factor)
- Structure type (Art. 3.7.3.1 and Table 3.1)**: OMEGA: Hot rolled steel secti, Open, 4 (Callout: Damping factor in %)
- Ductility (Art. 3.7.3.1 Table 3.1)**: MU: Non ductile (Mu=1) (Callout: Ductility)

Or issue:

```
~DEFSPEC, ALL, 0.4, I, ELASTIC, 1, 1, 4, 1
```

12.Modal Solution

- Main Menu> Seismic Design> Modal Solution
(Choose Subspace iteration and 20 for number of modes)

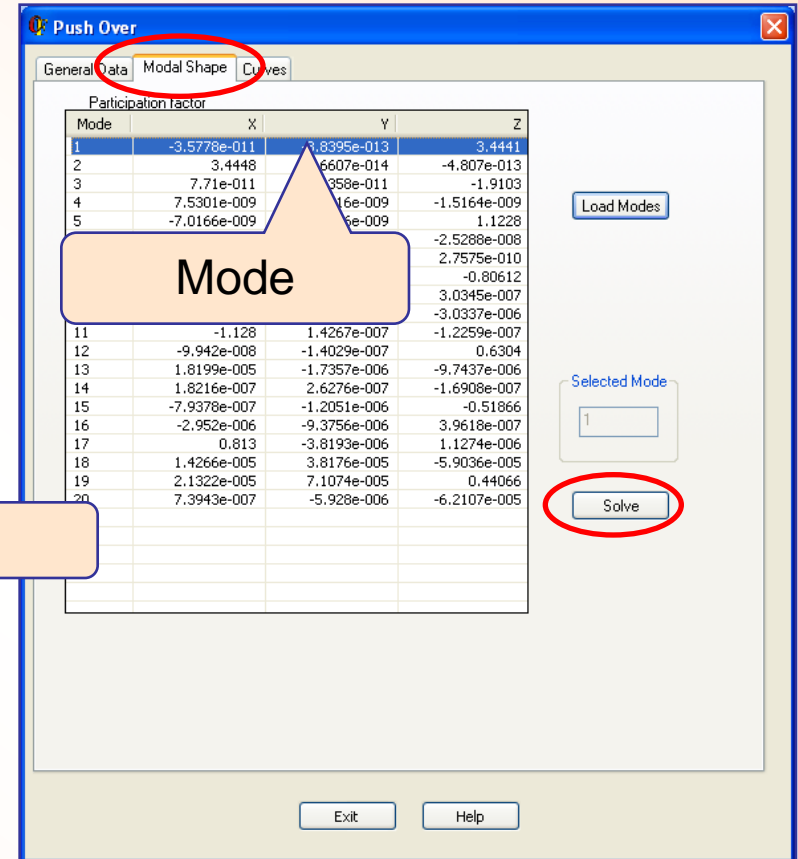
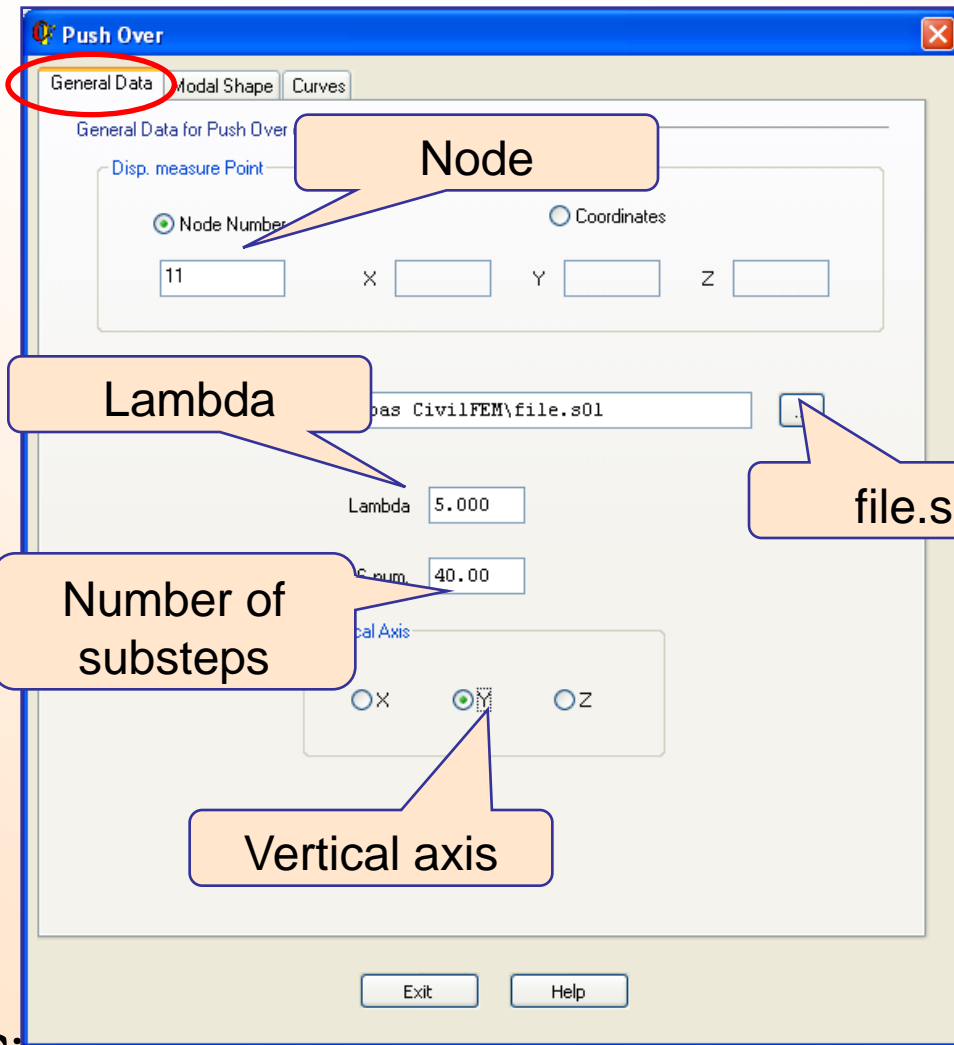


Or issue:

~MODLSOL, 20, SUBSP

13. Push-Over

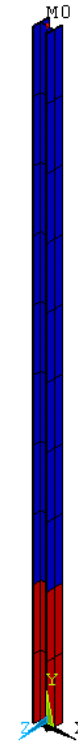
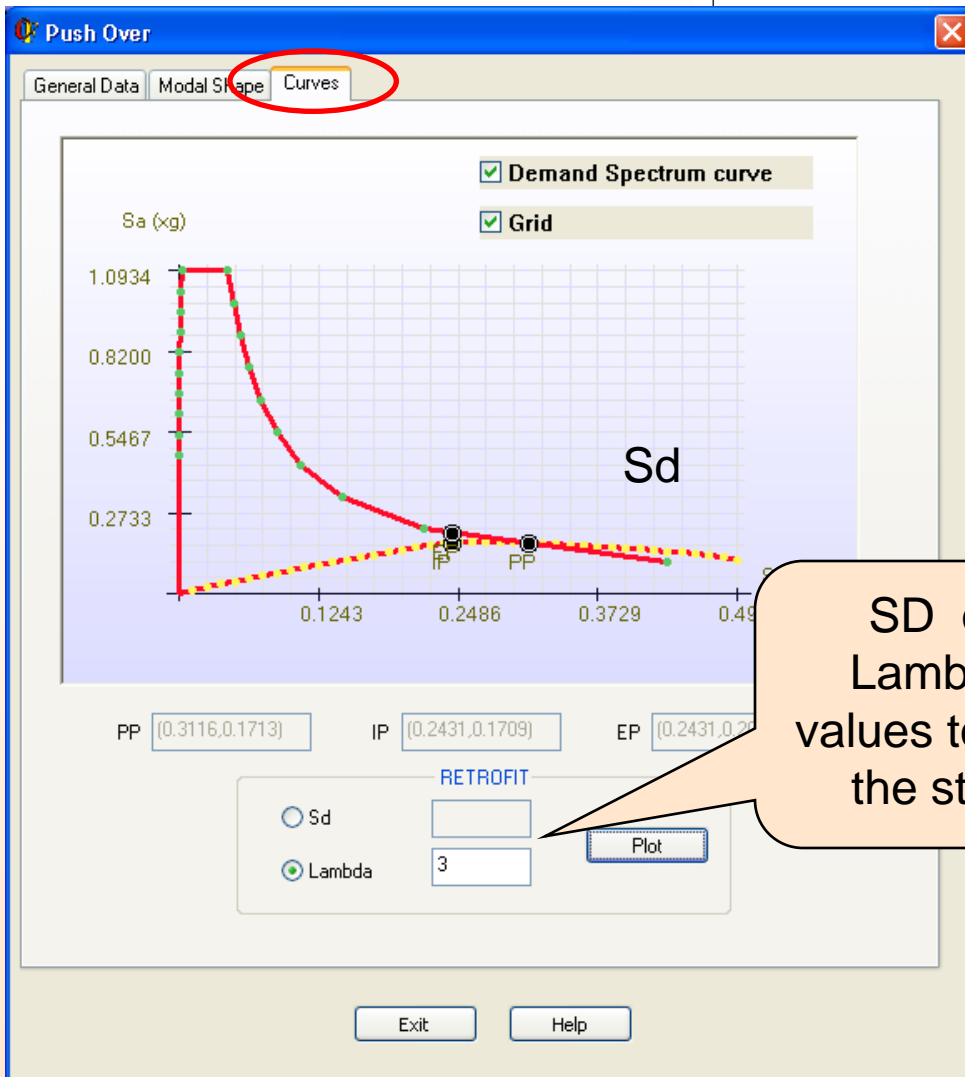
- Main Menu> Seismic Design> Push Over Analysis
(Choose: node 11, Lambda = 5, 40 substeps, Y for vertical axis, and mode 1)



Or issue:

```
~PUSHDEF, 1, 5, 40, Y, NODE, 11  
~PUSHMOD, 1  
~PUSHSLV
```

1 RETROFIT RESULTS
Yielded elements
Lambda = 3.000E+0



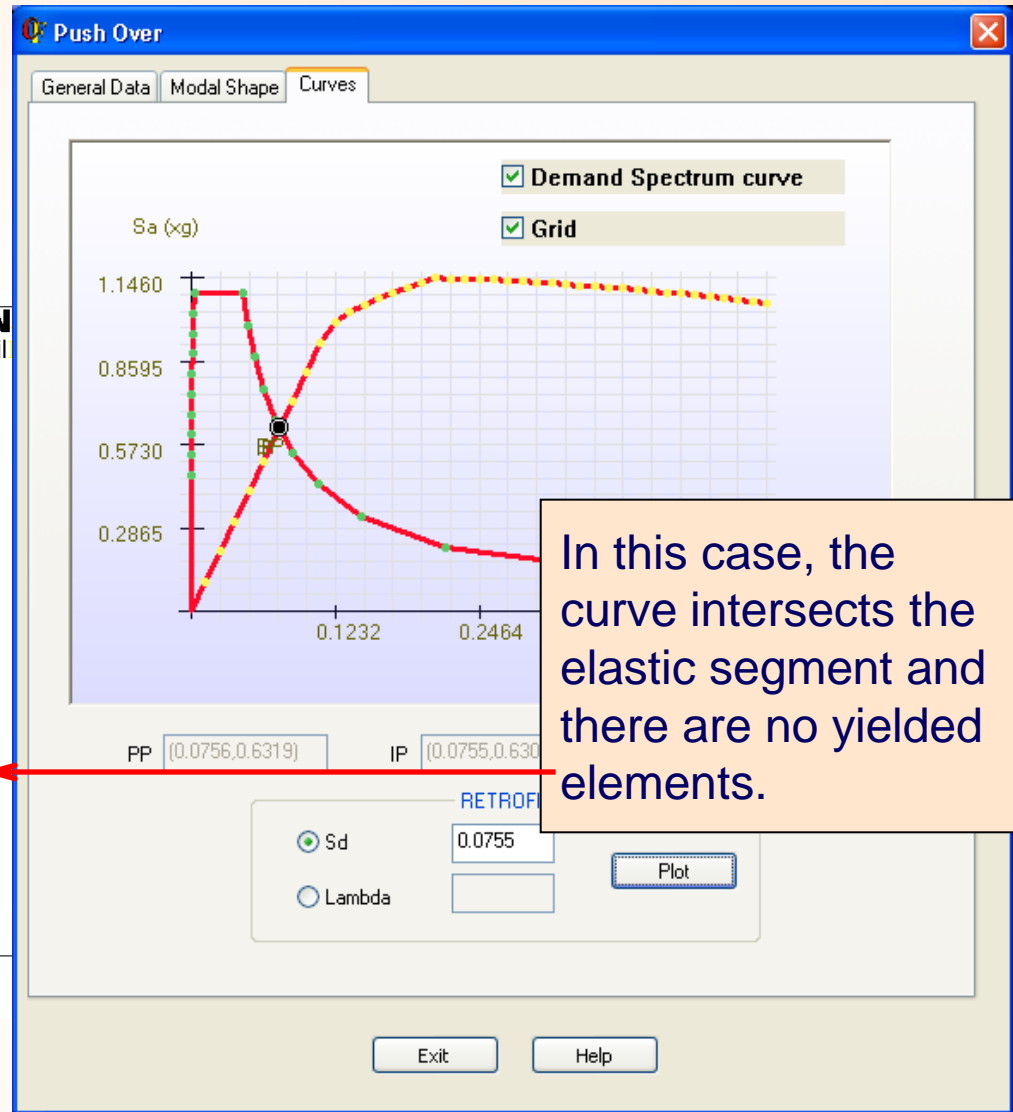
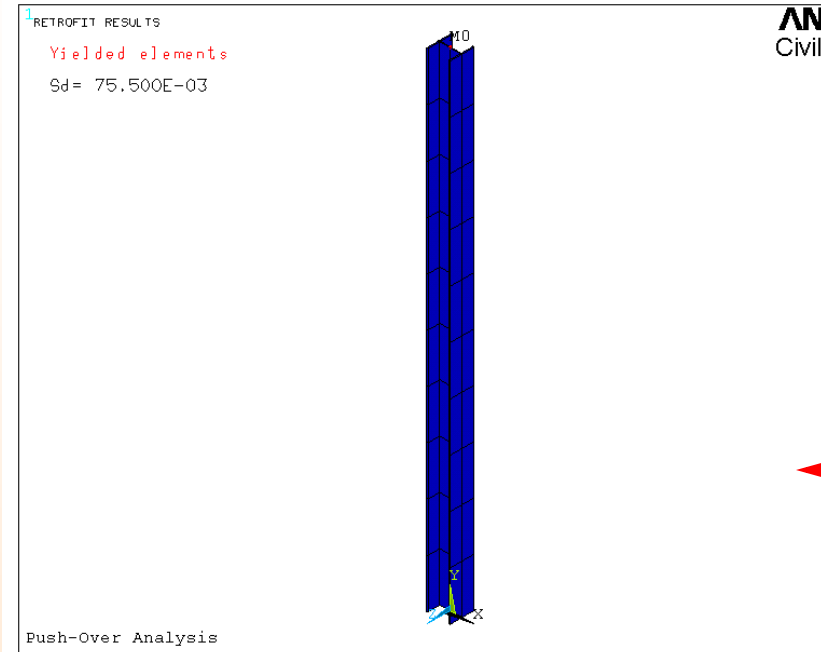
There are yielded elements;
HA 100 section is too weak.

Or issue:

/POST1

~RETROFT, LAMBDA, 3

The same example with HE 200 section.



In this case, the curve intersects the elastic segment and there are no yielded elements.

The same example
applying a load.

It is useful to use the ARCLLEN
command

```
/SOLU
```

```
F,11,FZ,2000 ! Load definition  
~CFLSWRT,1
```

```
ANTYPE,0
```

```
ARCLLEN,1
```

```
~PUSHDEF,1,1,10,Y,NODE,11
```

```
~PUSHMOD,1
```

```
~PUSHSLV
```

