CivilFEM®

INTRO ESSENTIAL

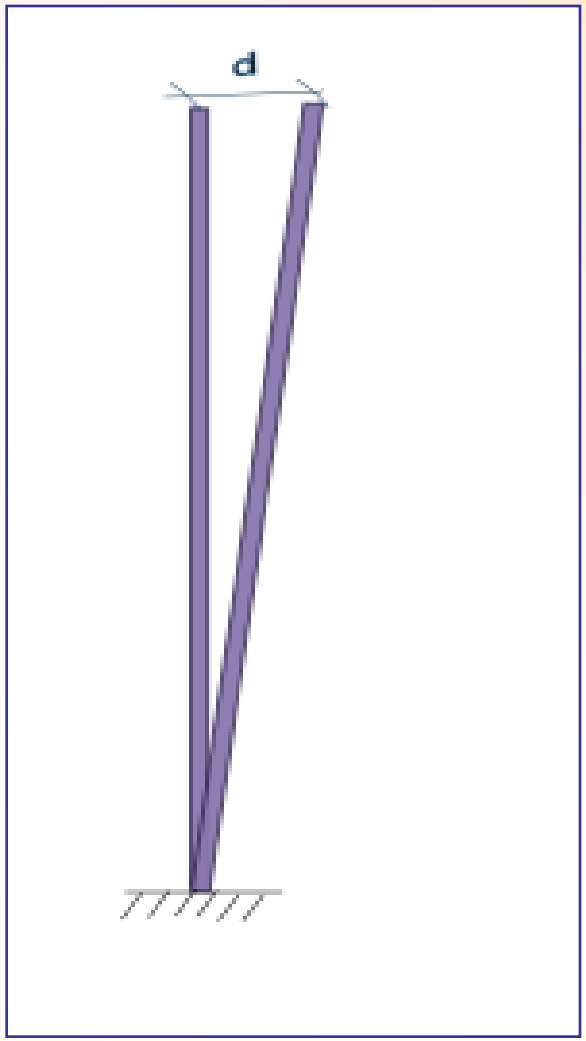
Practice 19 – Push-Over Analysis of a Cantilever Beam

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:

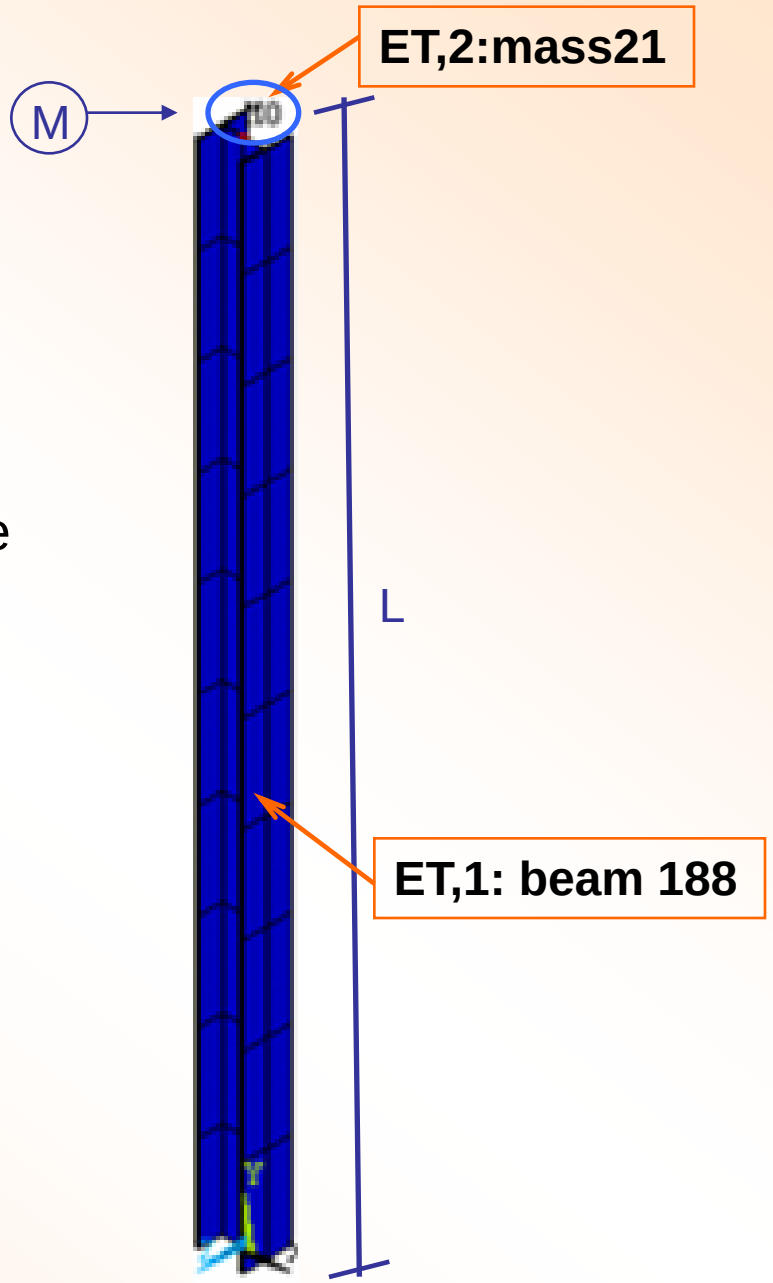
- BEAM188
- MASS21 for a mass located at the end of the beam (element 11)

Geometric Properties:

- L = 4 m
- M = 1500kg
- Section: HE A 100

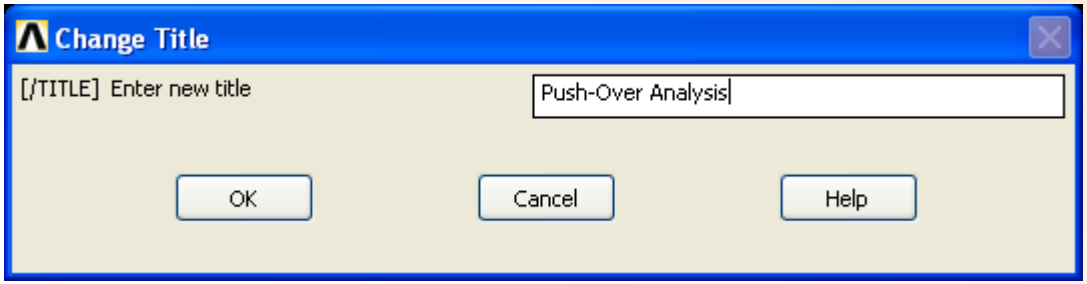
Imposed displacement:

- d = 0.1 m



1. Enter the title: Push-Over Analysis

   Utility Menu> File> Change Title…



Or issue:

```
/TITLE,Push-Over Analysis
```

2. CivilFEM setup: Code & Units

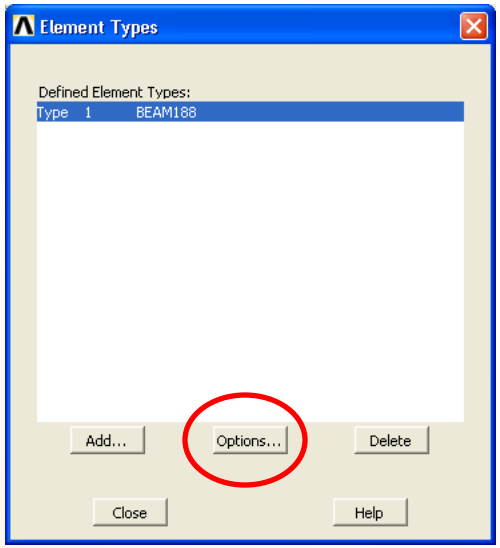
   - Main Menu> Civil Setup
     - Select International System units.
     - Select codes: EA (steel), EHE (concrete, not used) and NCSE02 (seismic)

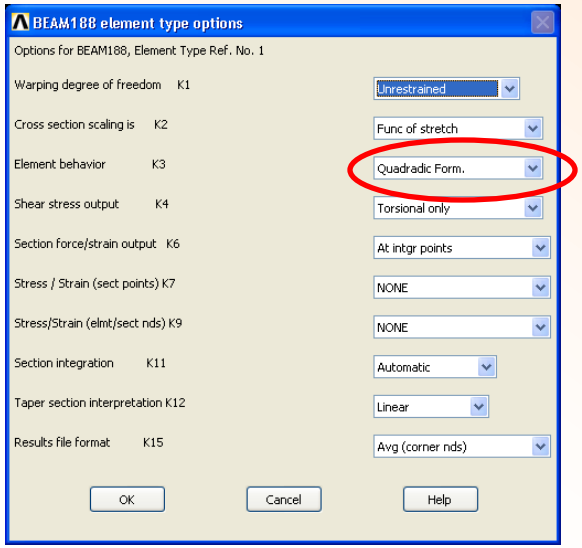
Or issue:

```
~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:





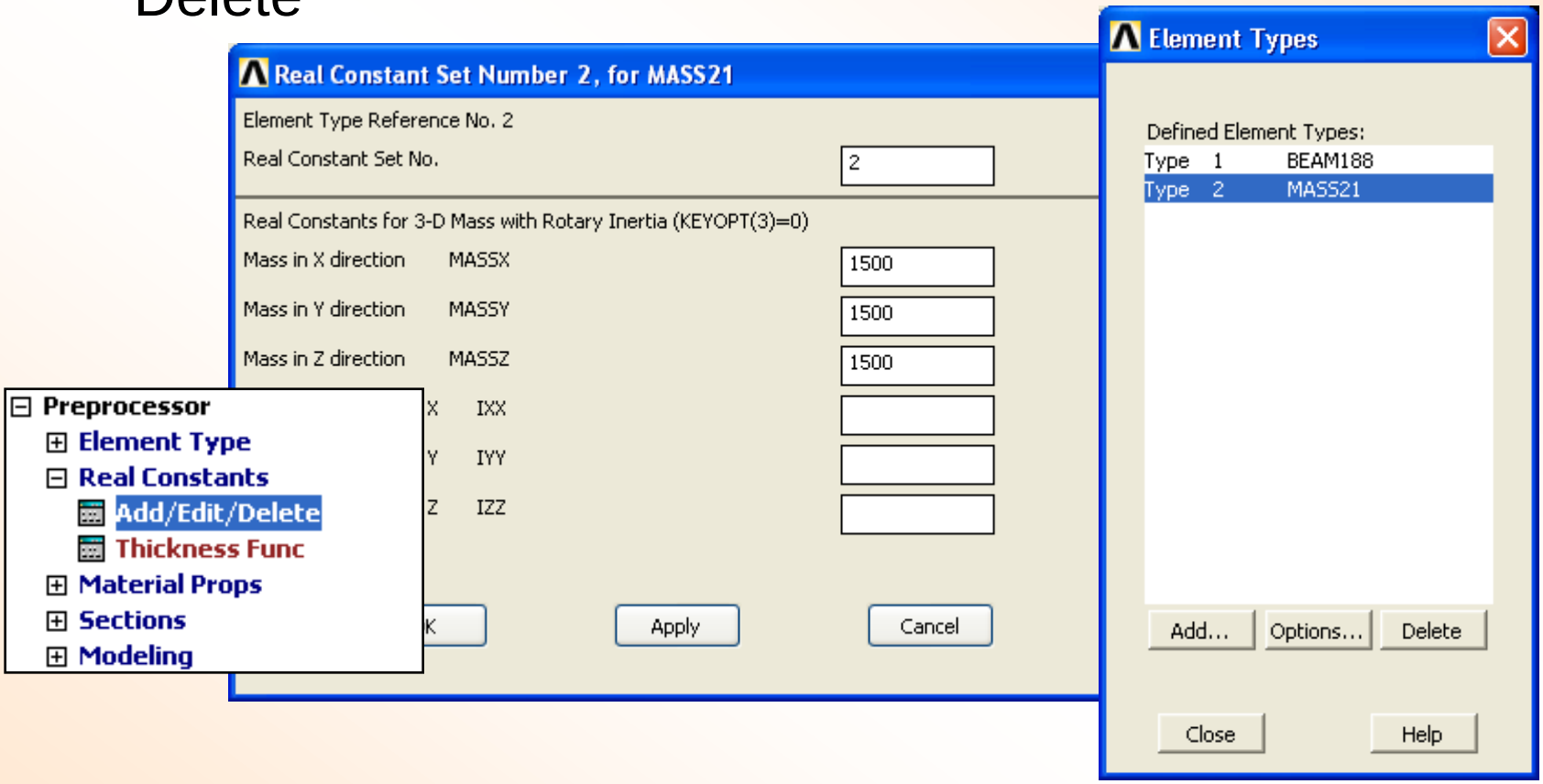
- Add MASS21 Element: Main Menu> Civil Preprocessor> Element types> Other Elements > Add/ Edit/ Delete > MASS21

Or issue:

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

4. Define real constant for the mass.

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

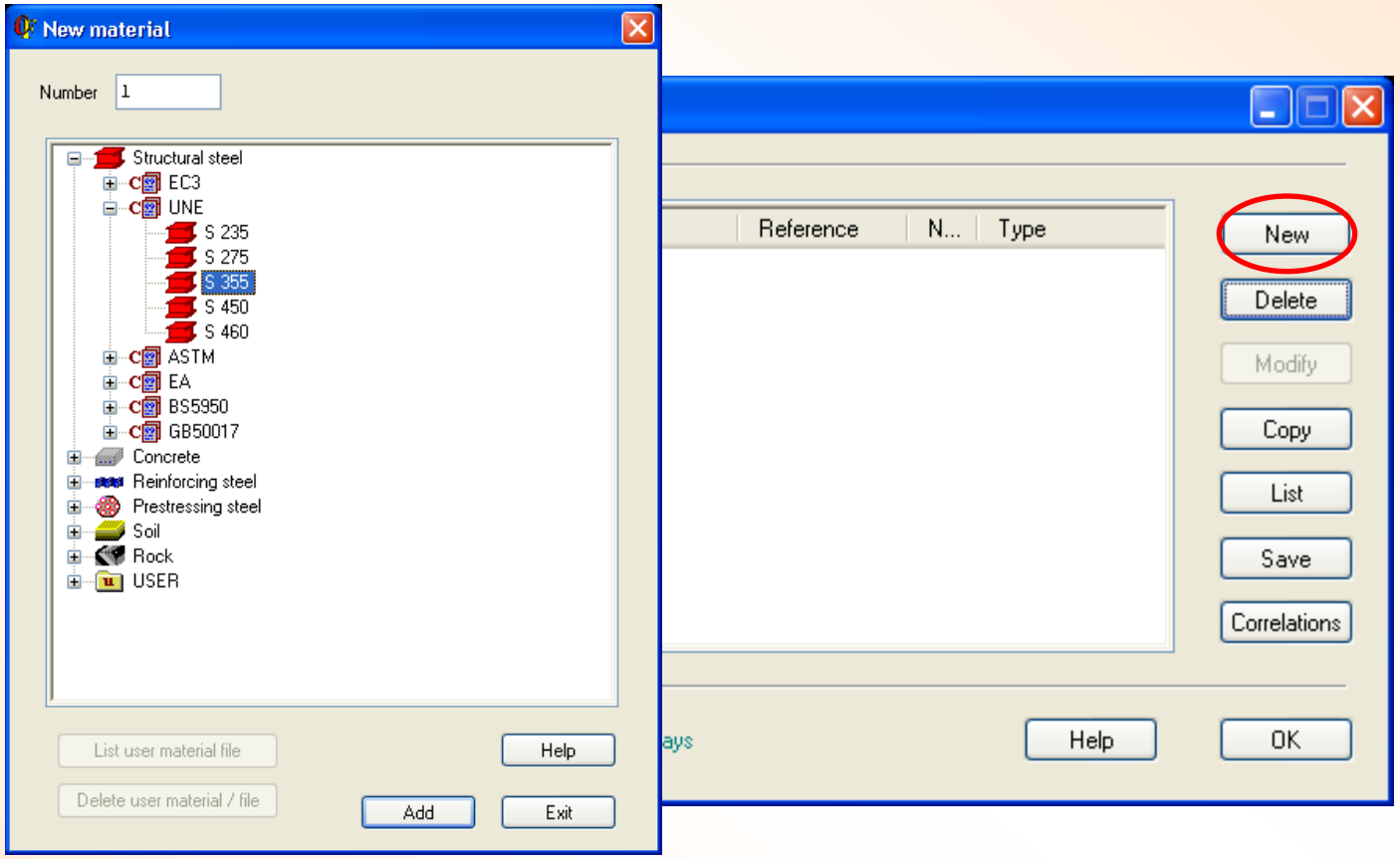


Or issue:

```
R,2,1500,1500,1500
```

5. Then define the material.

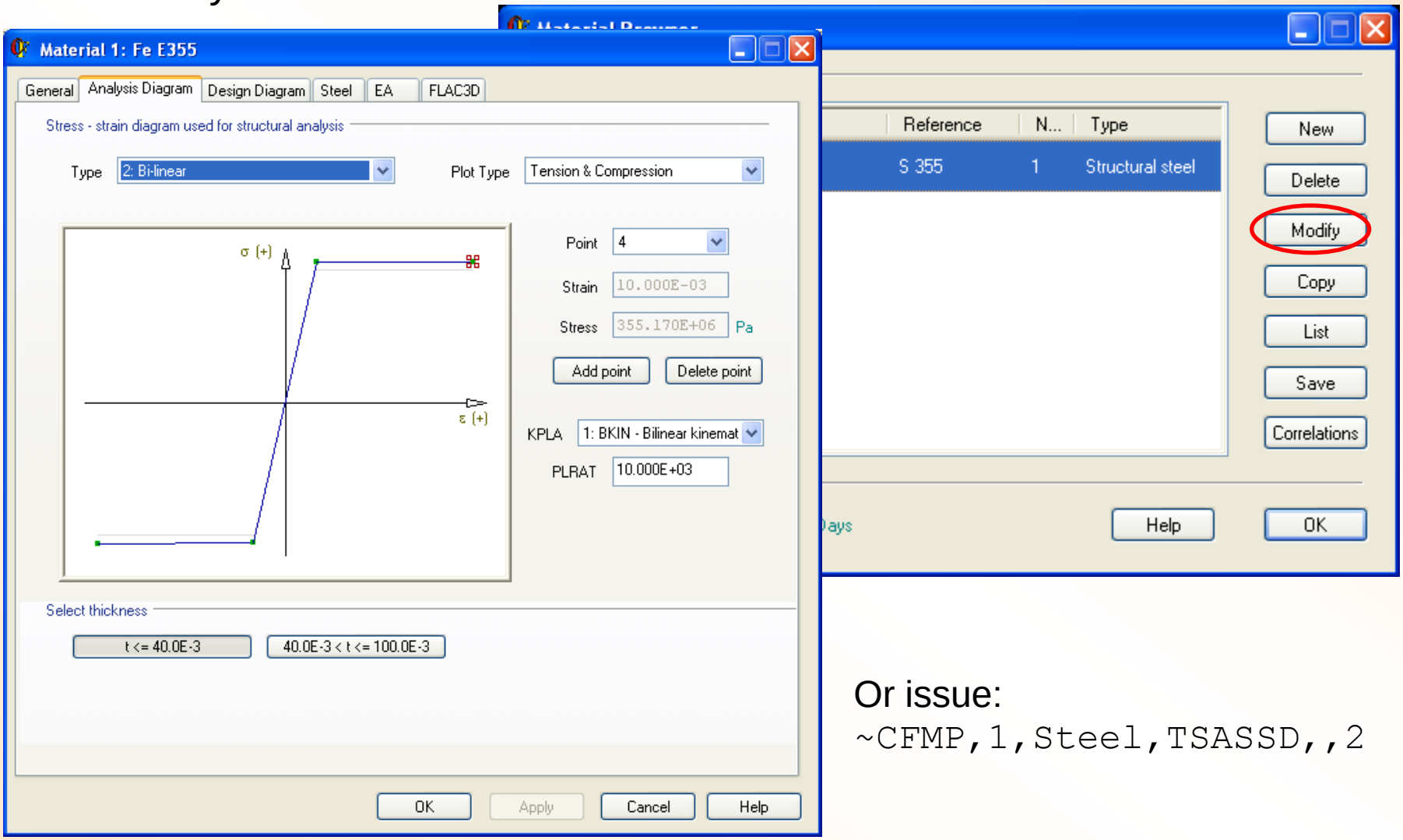
- Main Menu> Civil Preprocessor> Materials



Or issue:

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

Bilinear Analysis Diagram for Plasticity

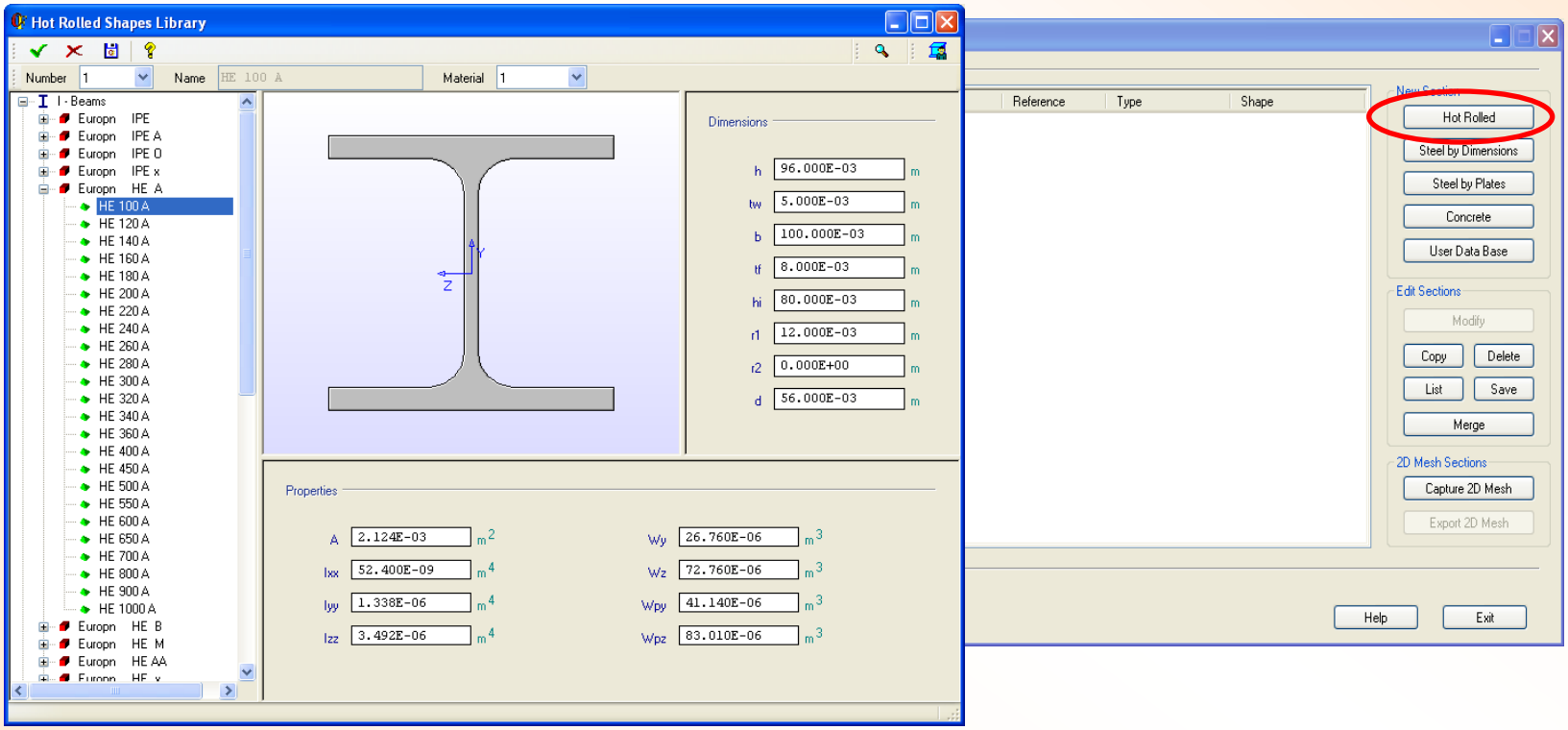


Or issue:

```
~CFMP,1,Steel,TSASSD,,2
```

6. Select the cross section of the beam.

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

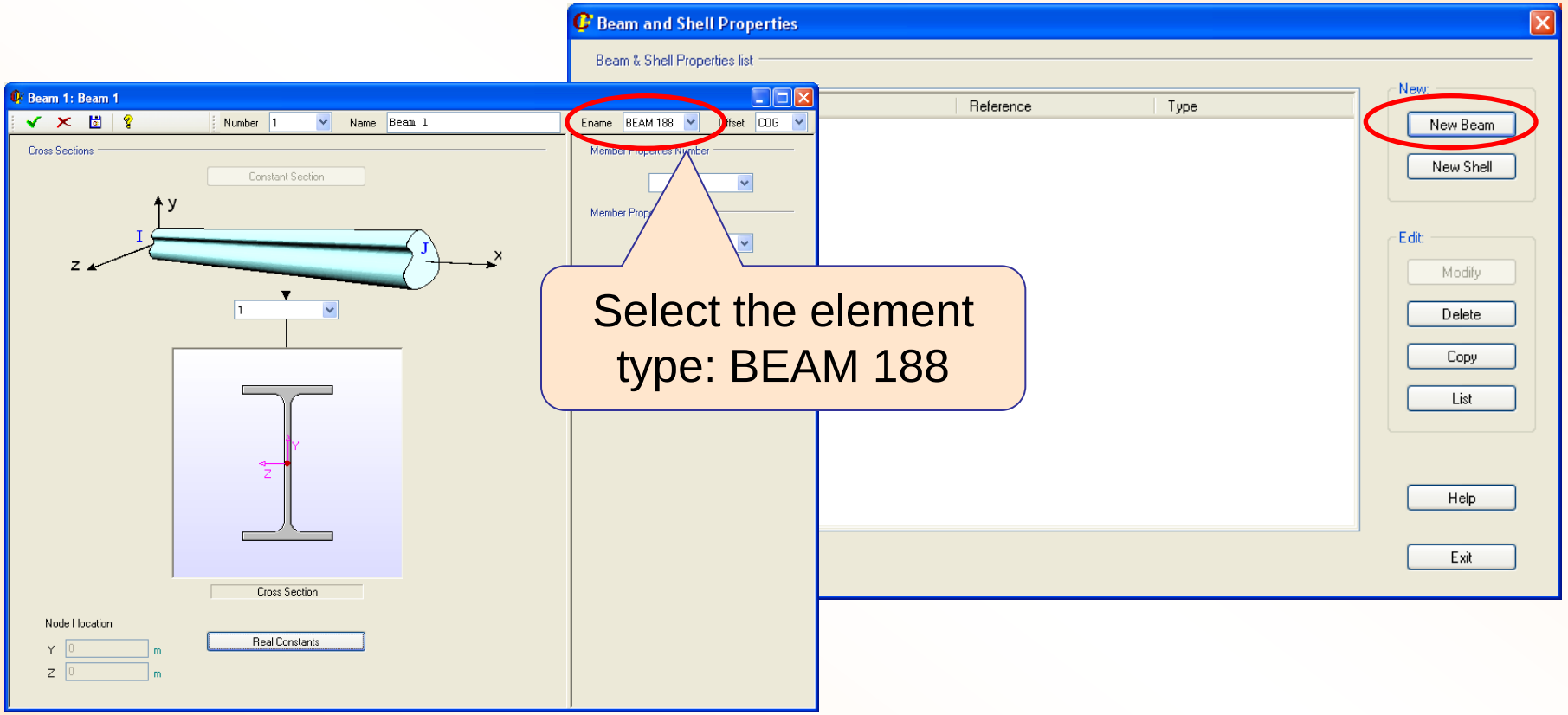


Or issue:

```
~SSECLIB,1,1,5,1
```

7. Define the Beam & Shell properties.

- Main Menu> Civil Preprocessor> Beam & Shell Properties



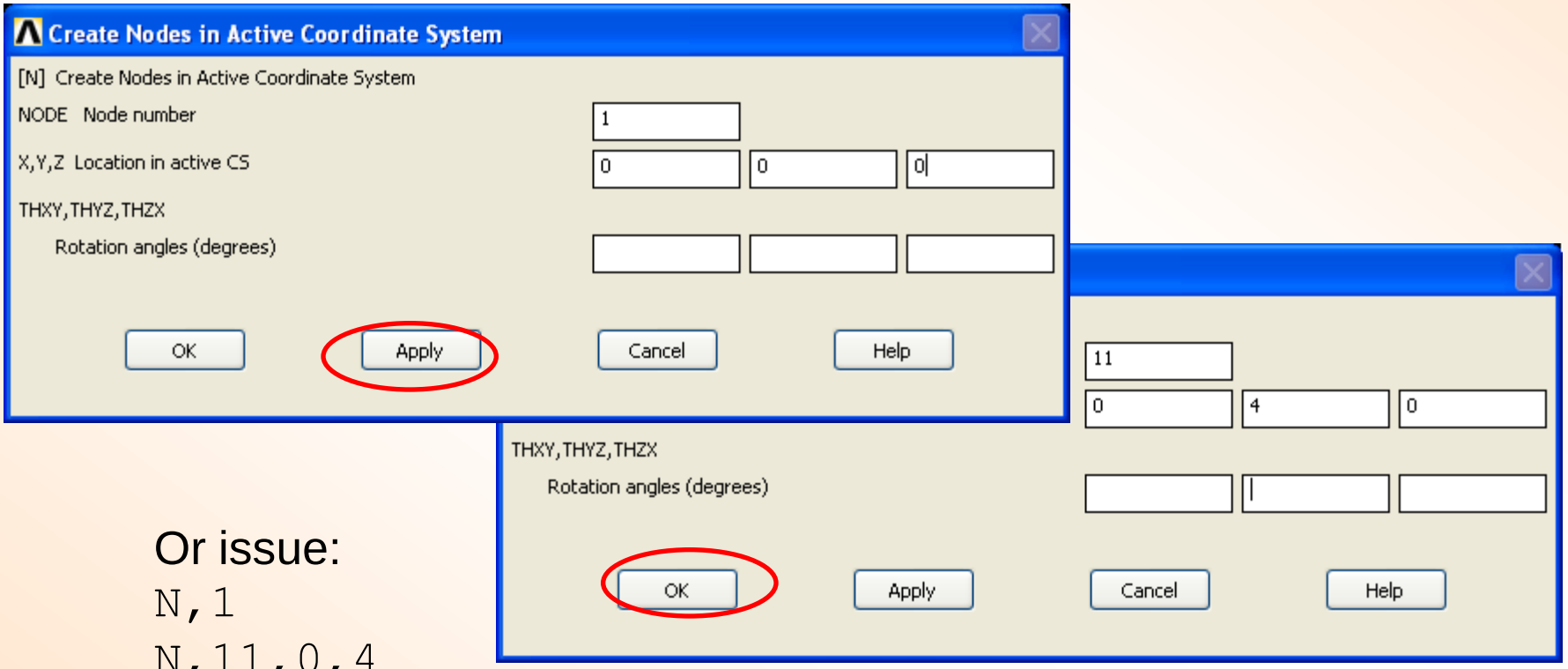
Or issue:

```
~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

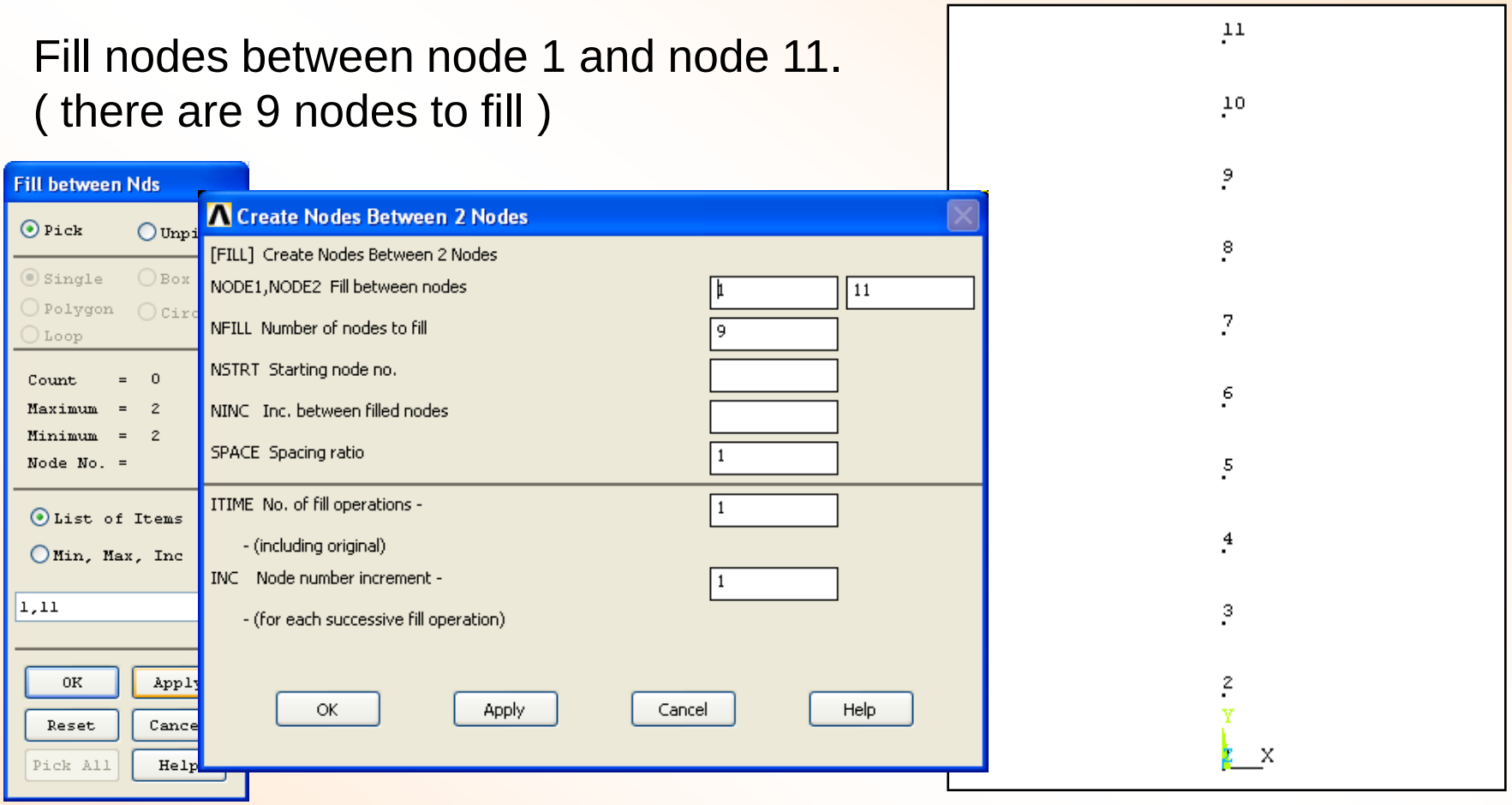


Or issue:

```
N,1
N,11,0,4
```

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

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



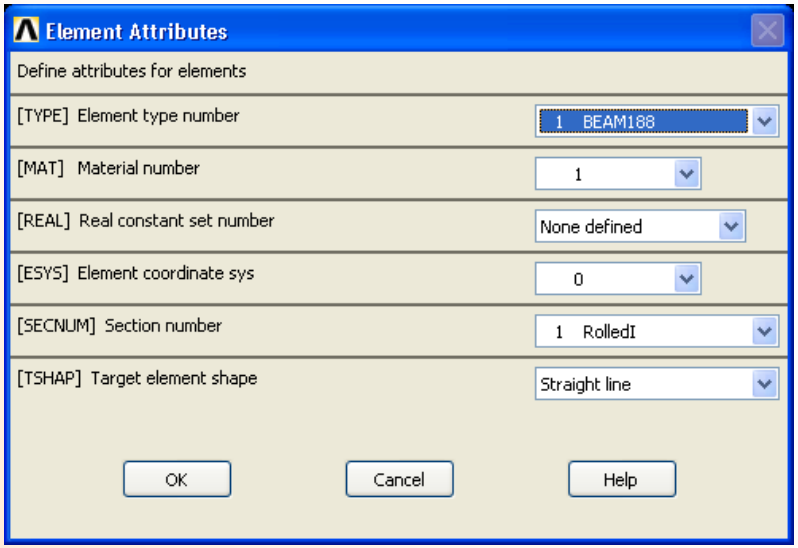
Or issue:

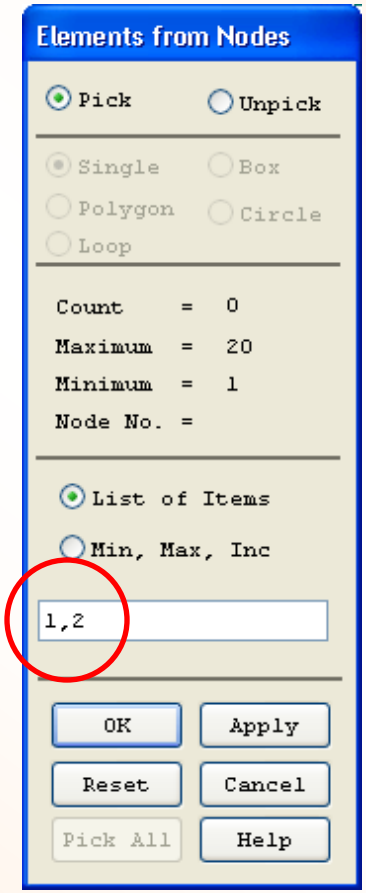
```
FILL,1,11
```

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:

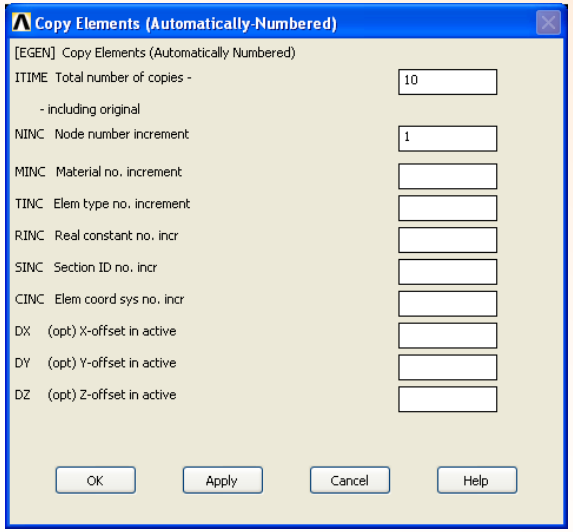
```
TYPE,1
MAT,1
SECNUM,1
E,1,2
```

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

Copy elements:

Or issue:

```
EGEN,10,1,ALL
```



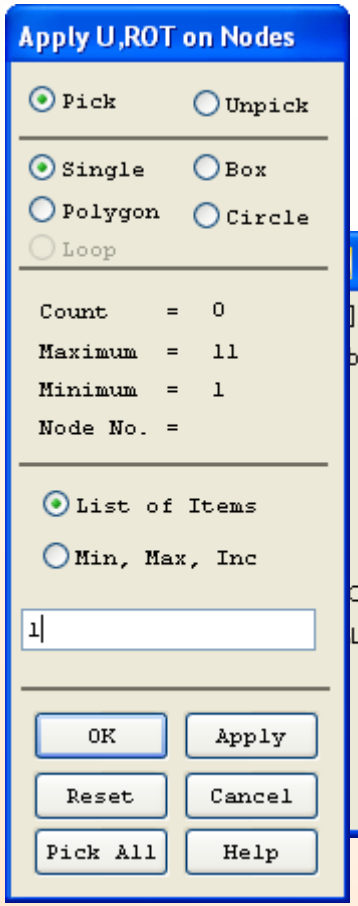
- Define mass element:

Issue:

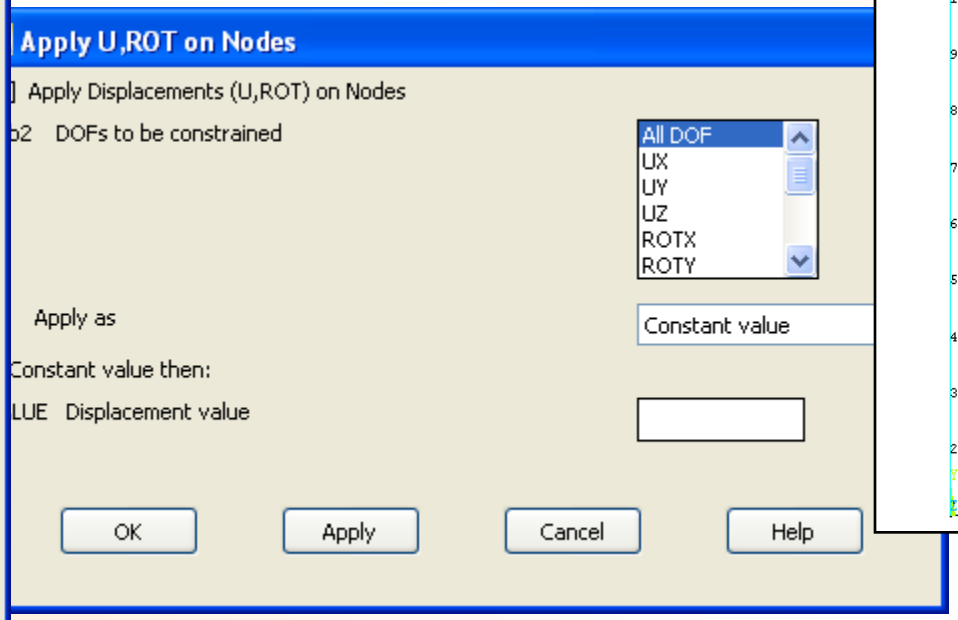
```
TYPE,2  $  REAL,2  $  E,11
```

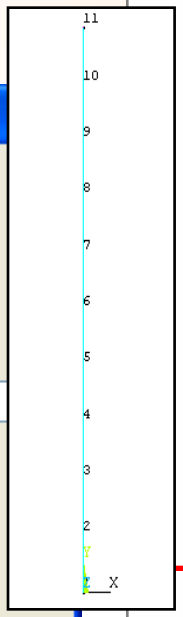
10. Boundary Conditions and Loads

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



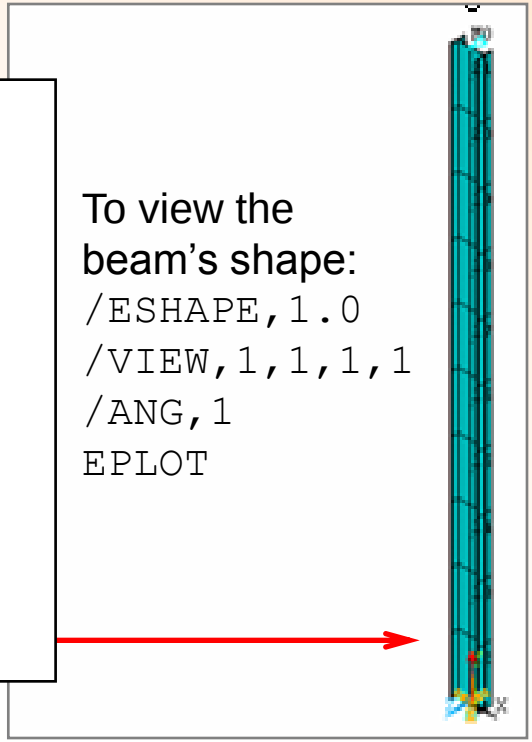
Constrain all displacements at Node 1.





To view the beam's shape:

```
/ESHAPE,1.0
/VIEW,1,1,1,1
/ANG,1
EPLOT
```

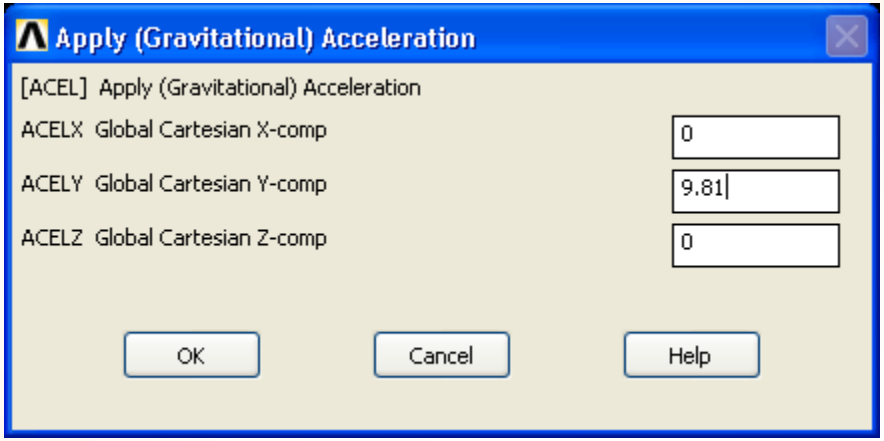


Or issue:

```
/SOLU
D,1,ALL
```

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

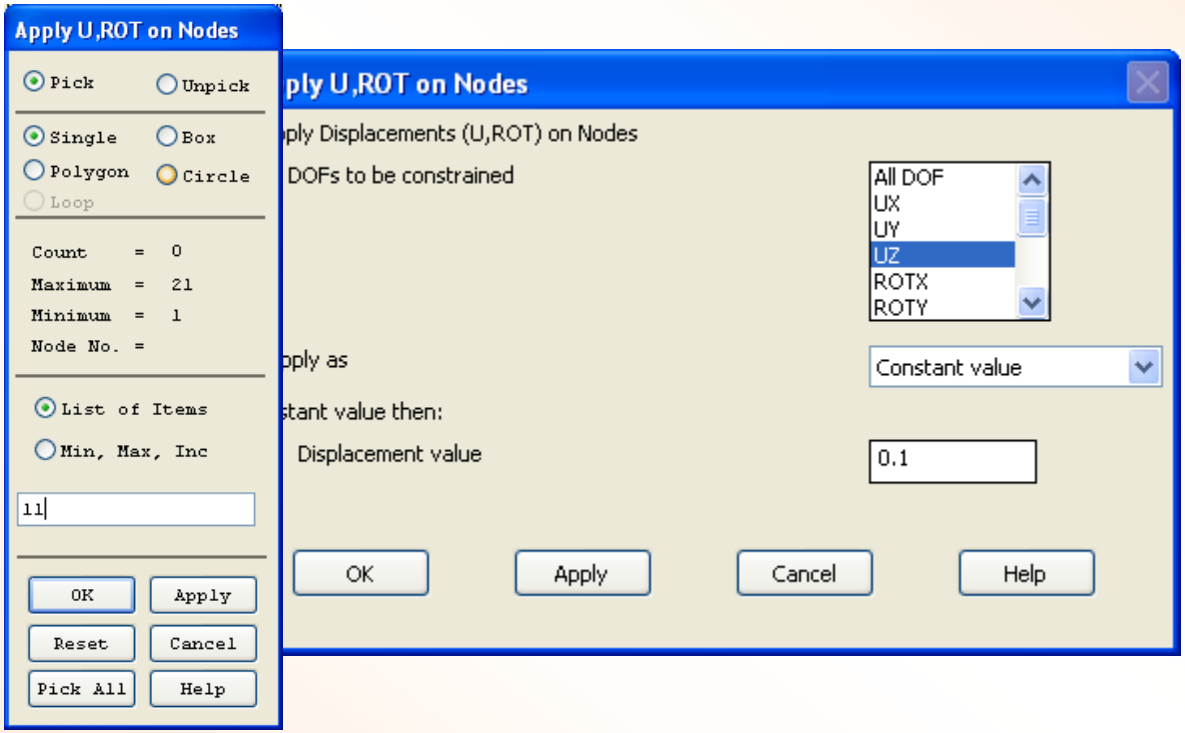
Define the Gravity (it must be input as positive).



Or issue:

```
ACEL,,9.81
```

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



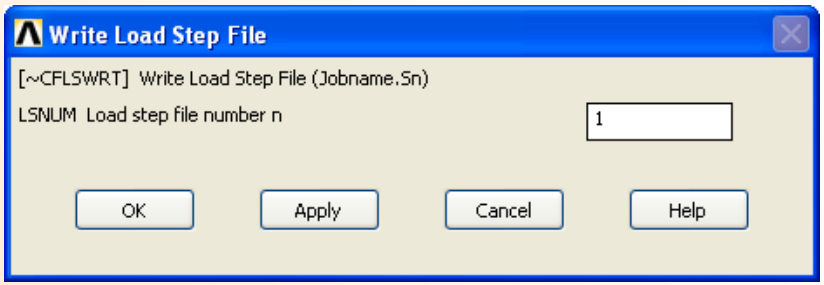
Displacement in Z for Push-Over

Or issue:

```
D,11,UZ,0.1
```

Save in a Load State File

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

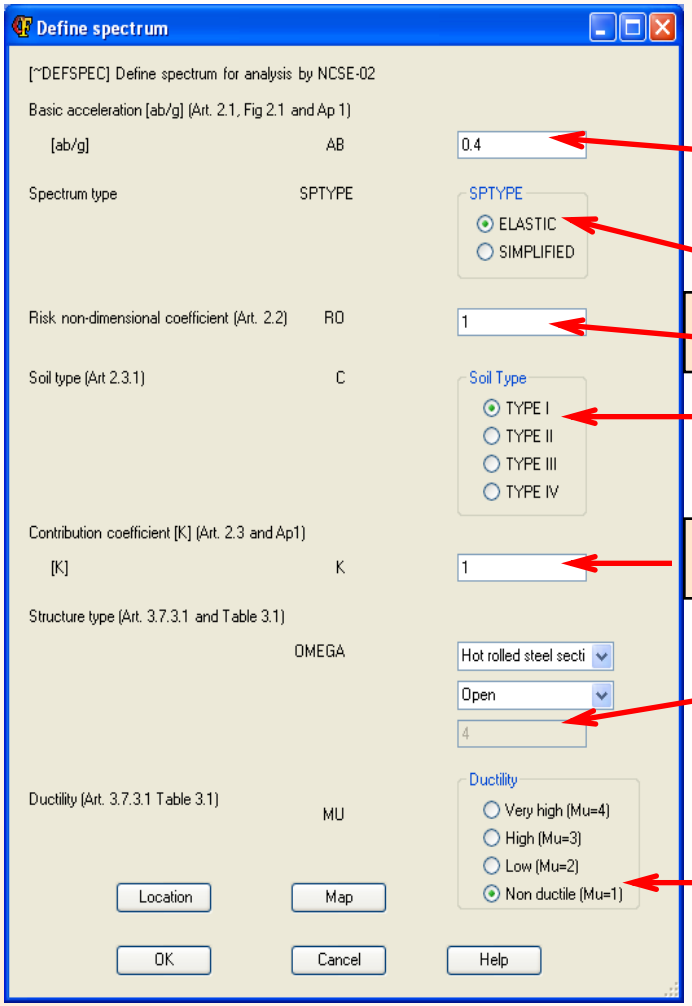


Or issue:

```
~CFLSWRT,1
```

11.Define Spectrum

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



Basic acceleration

Spectrum Type

Risk coefficient

Subsoil class

Contribution factor

Damping factor in %

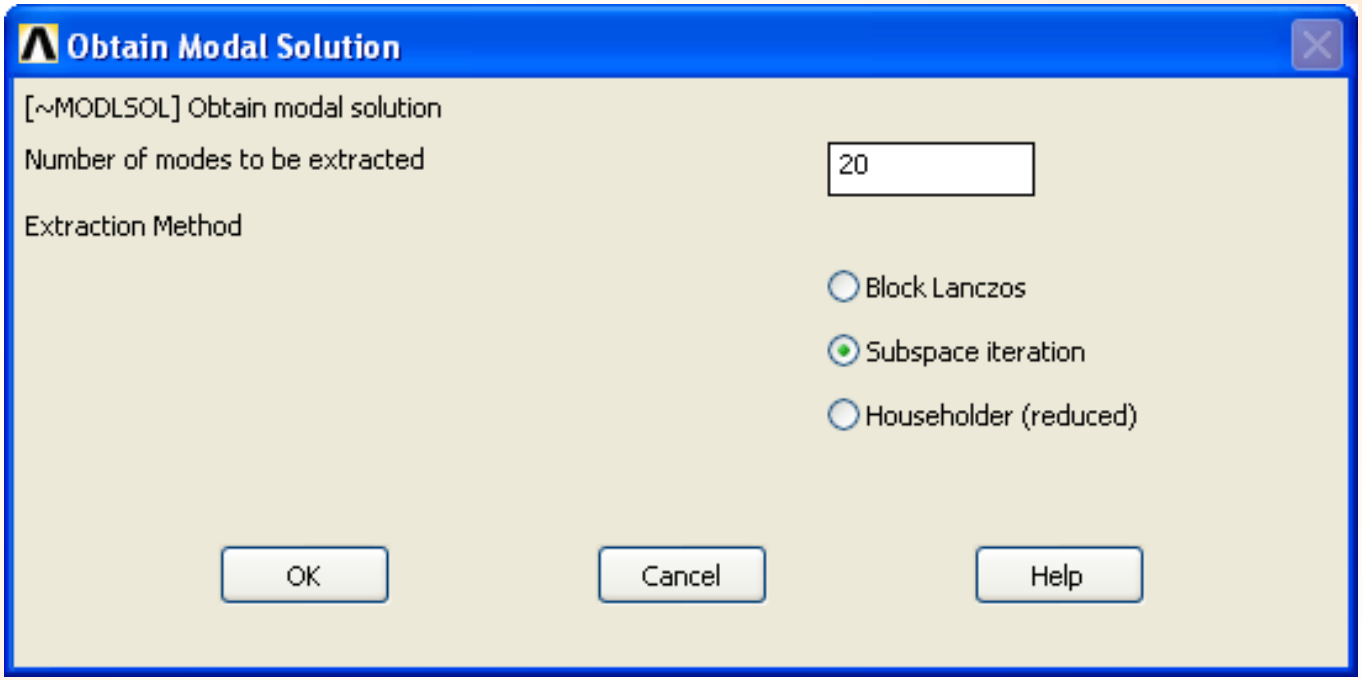
Ductility

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)



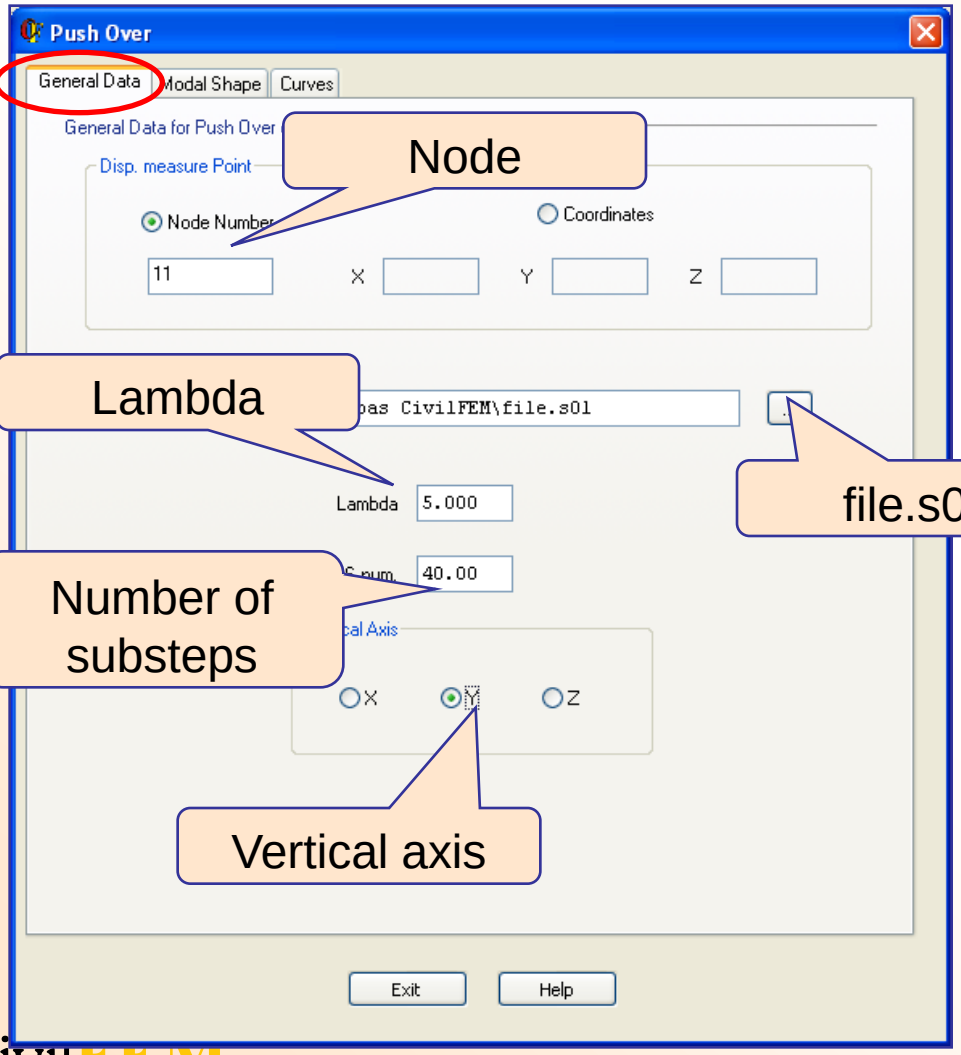
Or issue:

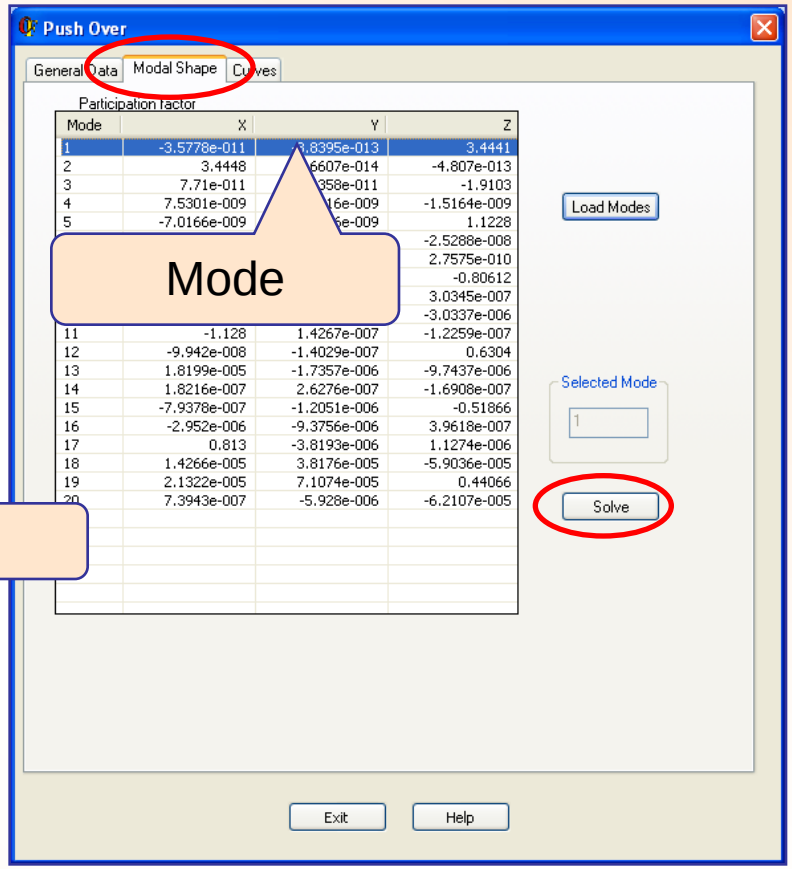
```
~MODLSOL,20,SUBSP
```

13. Push-Over

- Main Menu> Seismic Design> Push Over Analysis

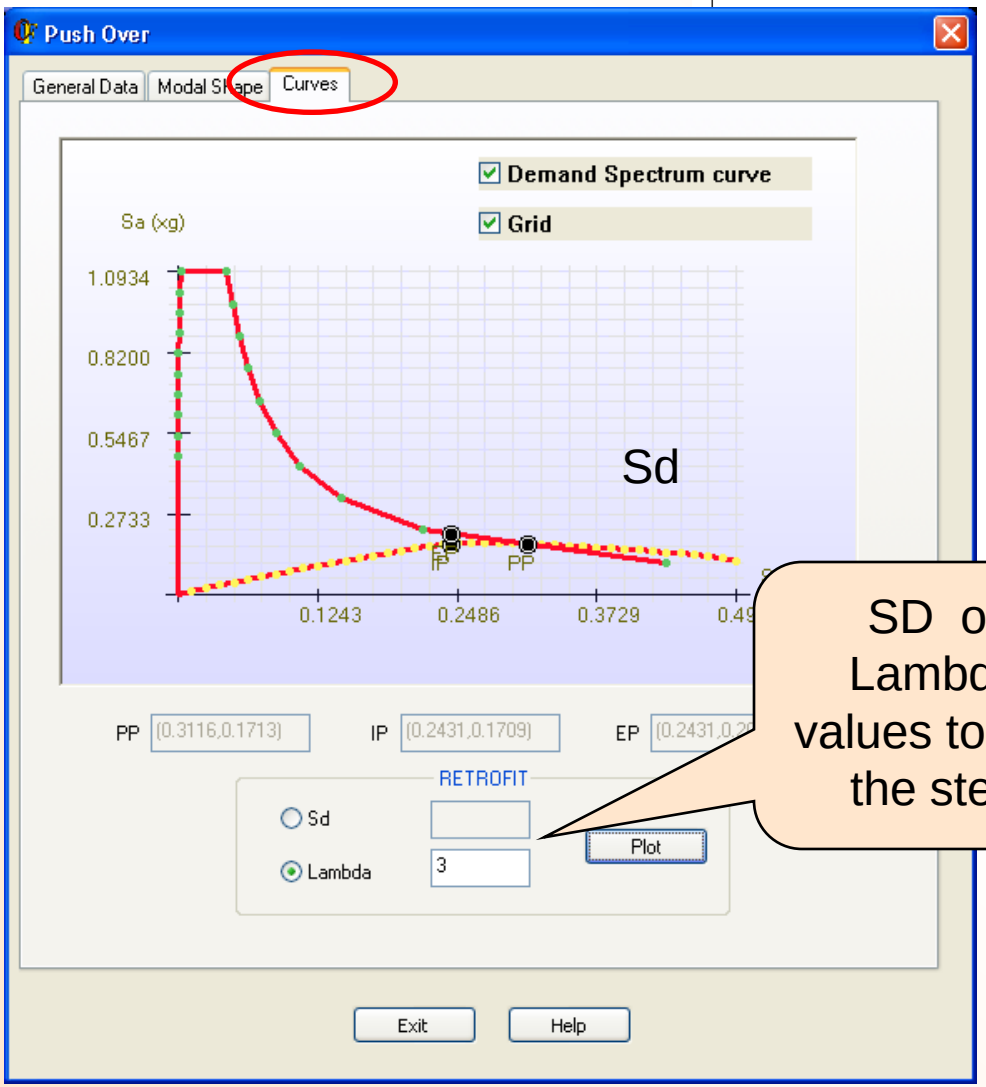
(Choose: node 11, Lambda = 5, 40 substeps, Y for vertical axis, and mode 1)

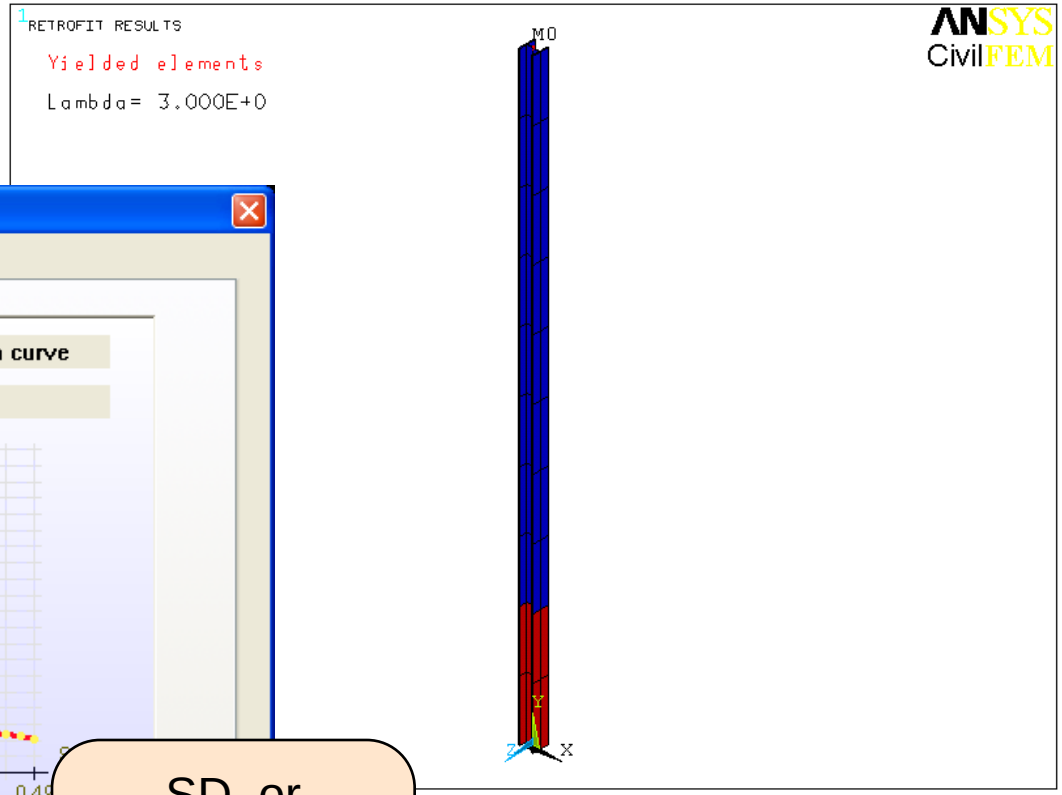




Or issue:

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





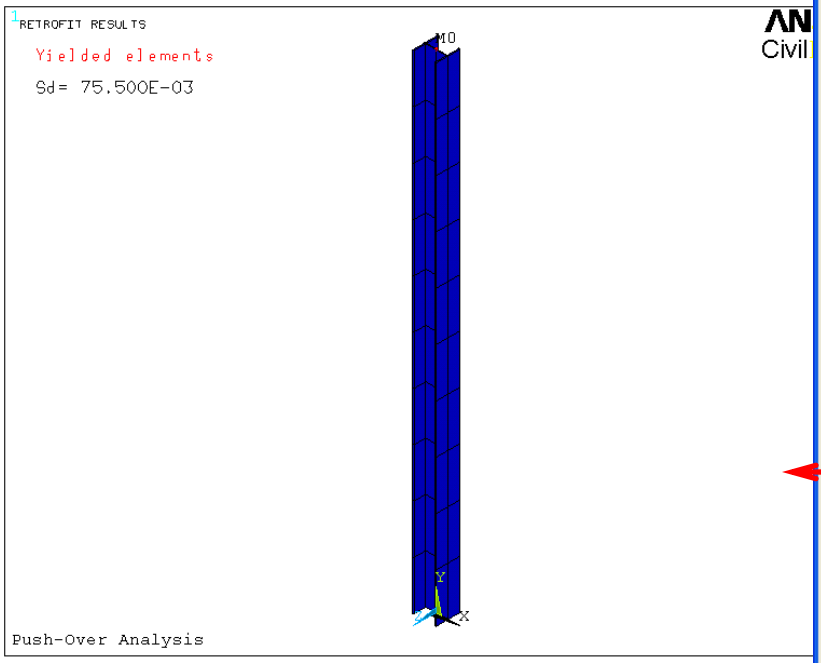
SD or Lambda values to plot the step

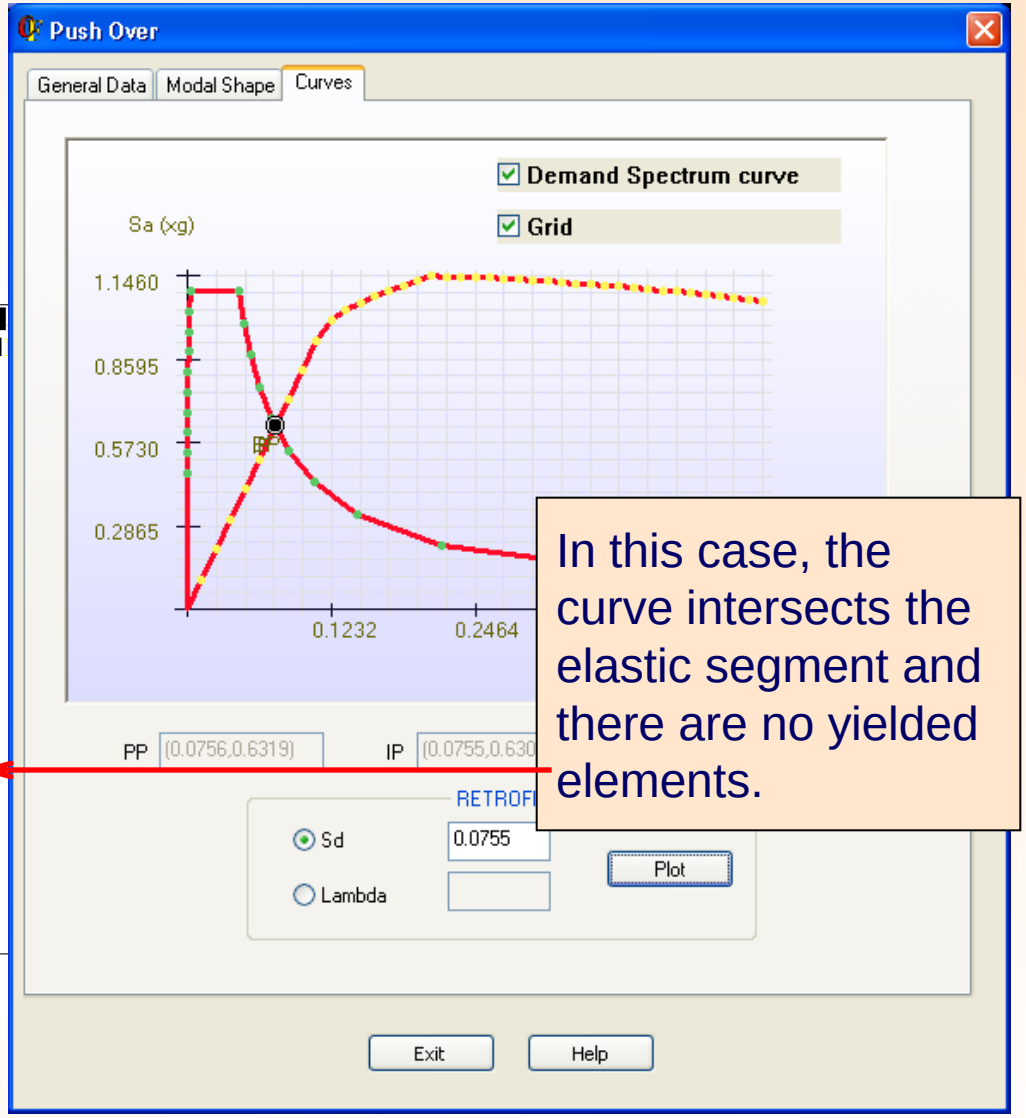
There are yielded elements; HA 100 section is too weak.

Or issue:

```
/POST1
~RETROFT,LAMBDA,3
```

The same example with HE 200 section.

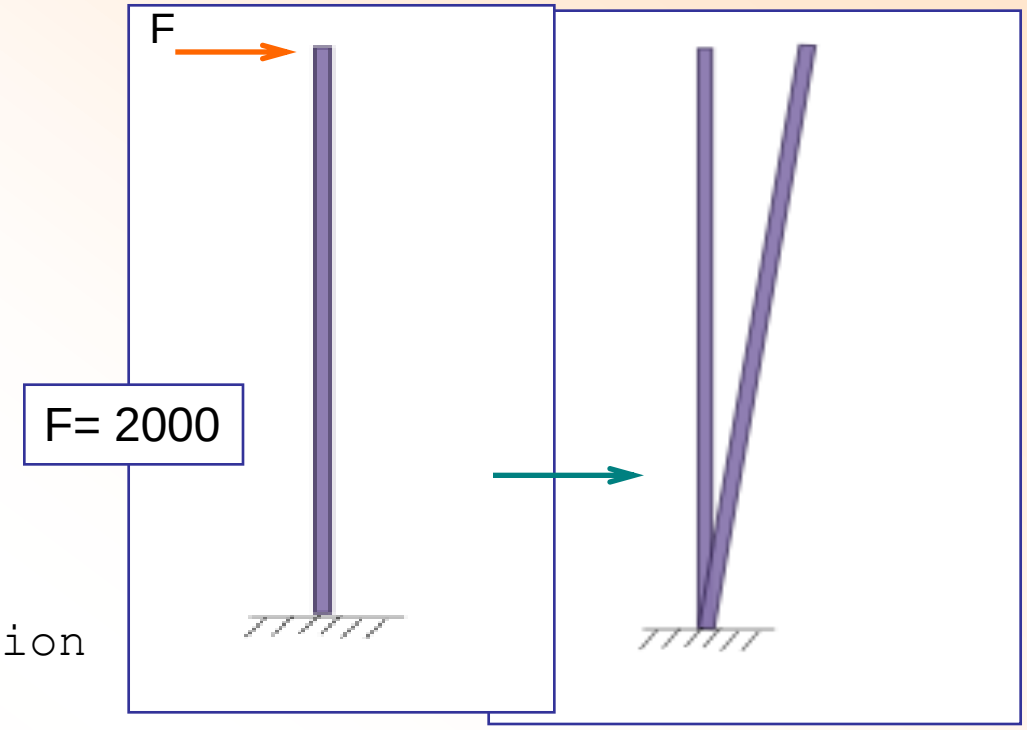




In this case, the curve intersects the elastic segment and there are no yielded elements.

The same example applying a load.

It is useful to use the ARCLEN command




```
/SOLU

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

ANTYPE,0
ARCLEN,1

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