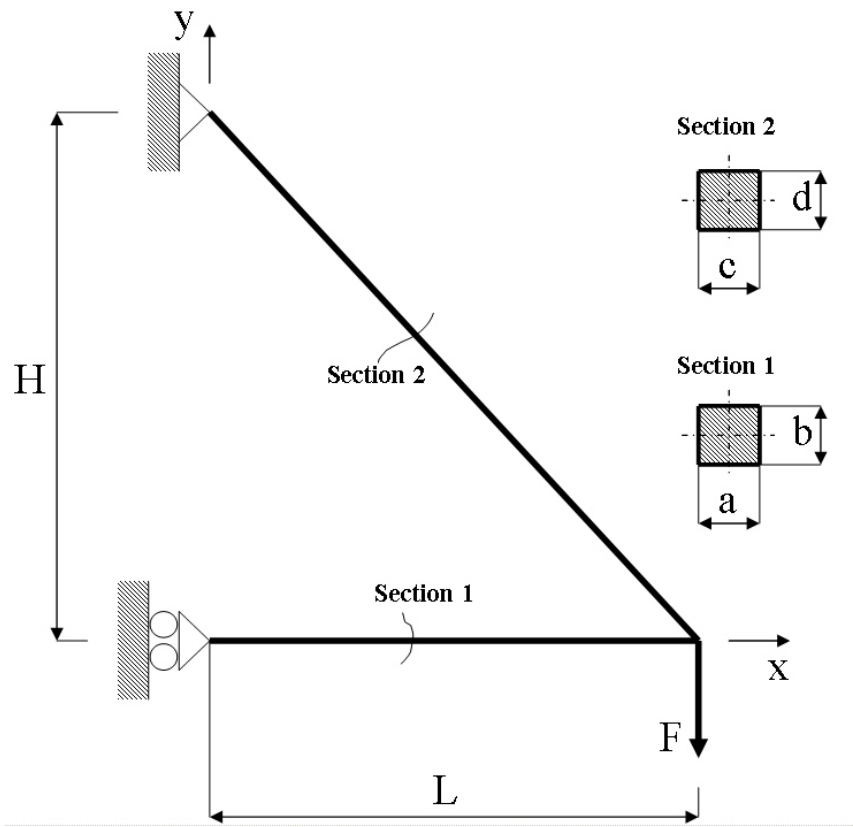
Course in ANSYS

Example0150

Example – Truss 2D



Objective:
Compute the maximum deflection
Tasks:
Display the deflection figure?
Topics:
Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, saving/restoring

$E = 210000 N/mm^2$
$\nu = 0.3$
$L = 100mm$
$H = 120mm$
$a = b = 20mm$
$c = d = 10mm$
$F = 100N$

Modeling considerations

- As you begin your model generation, you will (consciously or unconsciously) make a number of decisions that determine how you will mathematically simulate the physical system:
  - What are the objectives of your analysis?
  - Will you model all, or just a portion, of the physical system?
  - How much detail will you include in your model?
  - What kinds of elements will you use? How dense should your finite element mesh be?
- In general, you will attempt to balance computational expense (CPU time, etc.) against precision of results as you answer these questions.
- The decisions you make in the planning stage of your analysis will largely govern the success or failure of your analysis efforts.

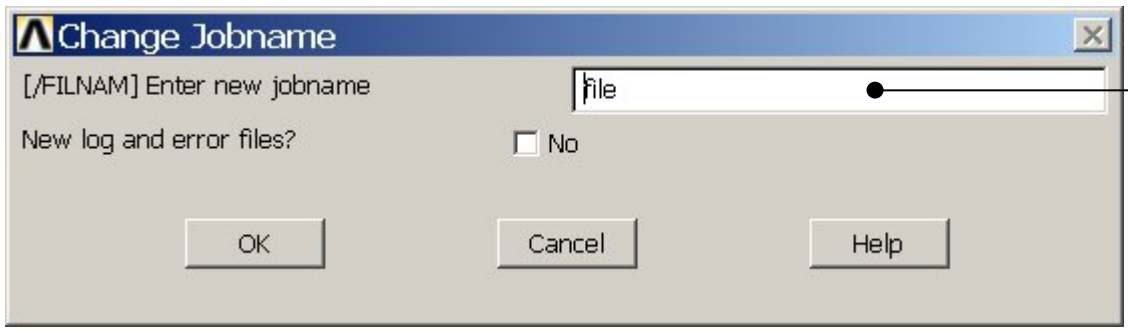
Modeling considerations

- Linear or Higher Order Elements
- Take Advantage of Symmetry
  - The axis of symmetry *must* coincide with the global Cartesian Y-axis.
  - Negative nodal X-coordinates are not permitted.
  - The global Cartesian Y-direction represents the axial direction, the global Cartesian X-direction represents the radial direction, and the global Cartesian Z-direction corresponds to the circumferential direction.
  - Your model should be assembled using appropriate element types:
    - For axisymmetric models, use applicable 2-D solids with KEYOPT(3) = 1, and/or axisymmetric shells. In addition, various link, contact, combination, and surface elements can be included in a model that also contains axisymmetric solids or shells. (The program will not realize that these "other" elements are axisymmetric unless axisymmetric solids or shells are present.)
- How Much Detail to Include
- Appropriate Mesh Density

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0150 — Command line entry

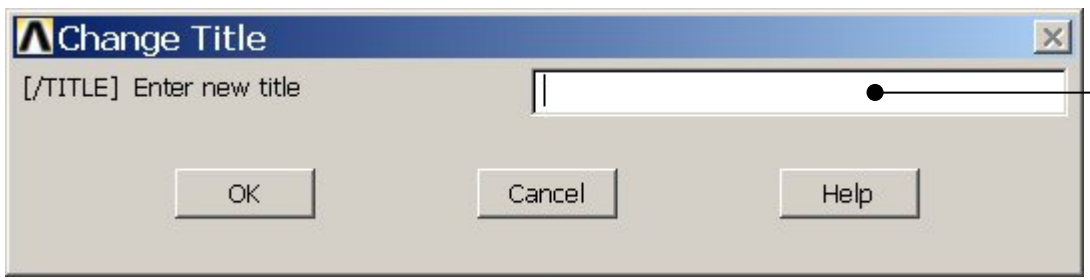


Enter: Example0150

Utility Menu > File > Change Title

/title, Truss 2D

Enter: Truss 2D



Example - Keypoints

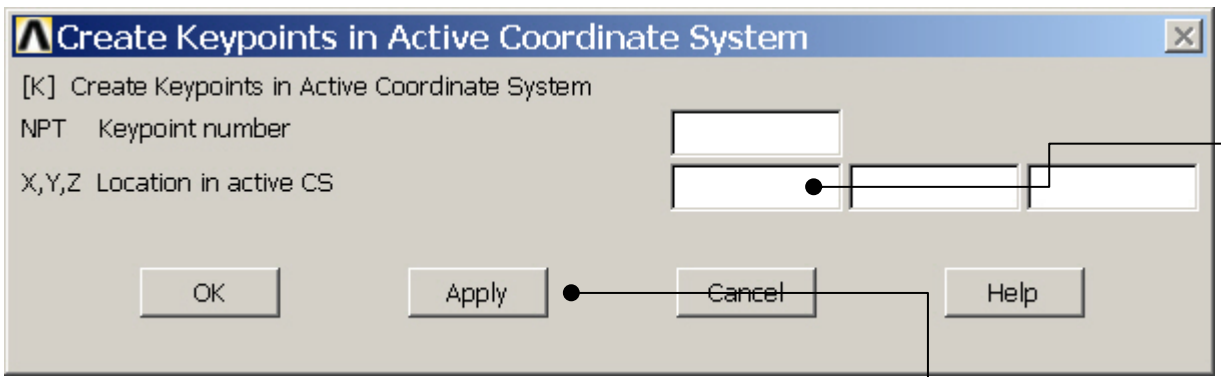
Note: An empty # result in automatic numbering.

Preprocessor > Modeling > Create > Keypoints > In Active CS

/PREP7
K,,,,
K,,100,,
K,,,120,

General format:
K,#,X,Y,Z

Keypoint number
X Keypoint x-coordinate
Y Keypoint y-coordinate
Z Keypoint z-coordinate

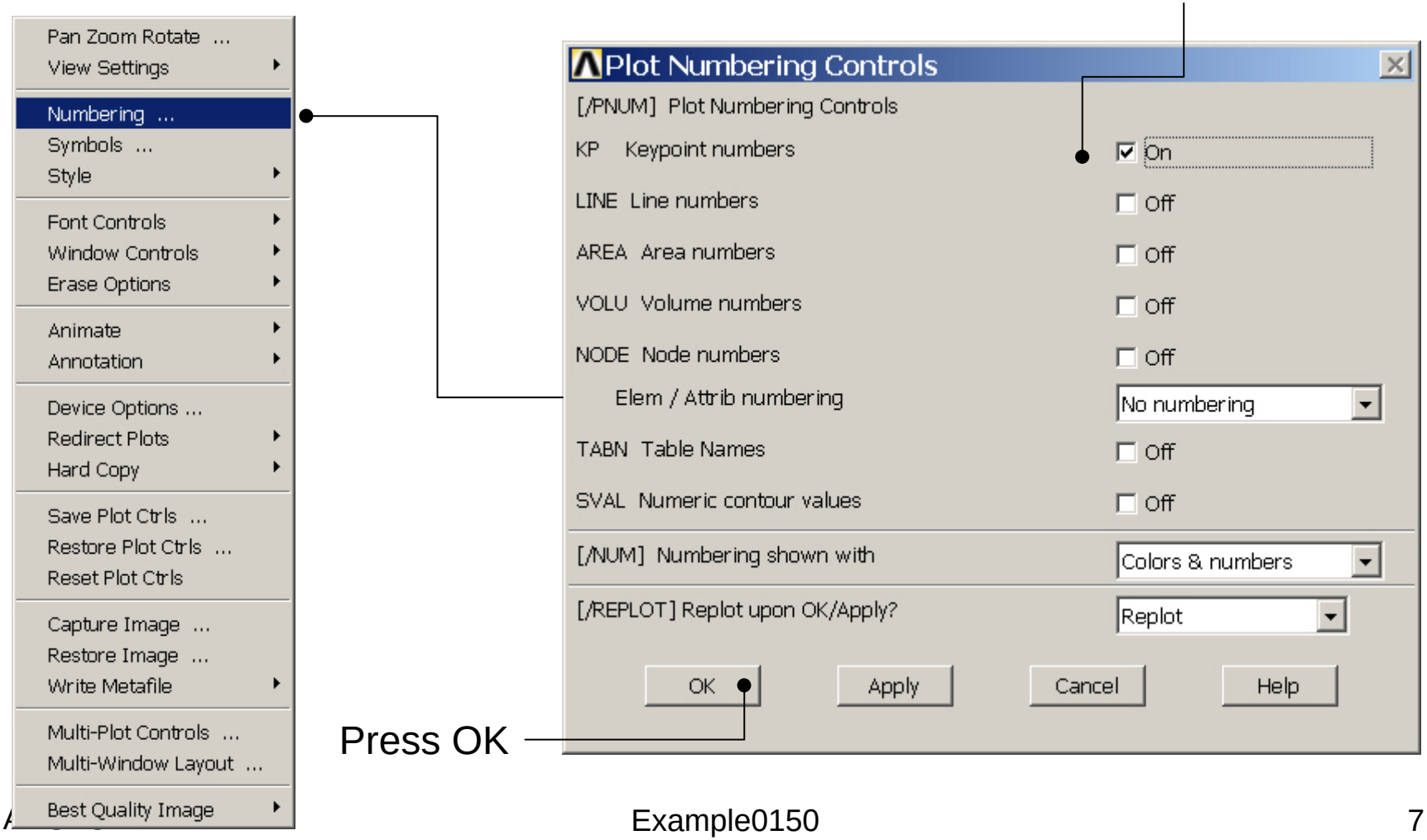


Press **Apply** for KP1
Enter 100 in the first field and
Press **Apply** for KP2
Enter 120 in the second field and
Press **Apply** for KP3

Example - Numbering

Utility Menu > PlotCtrls > Numbering

Switch on Keypoint numbers



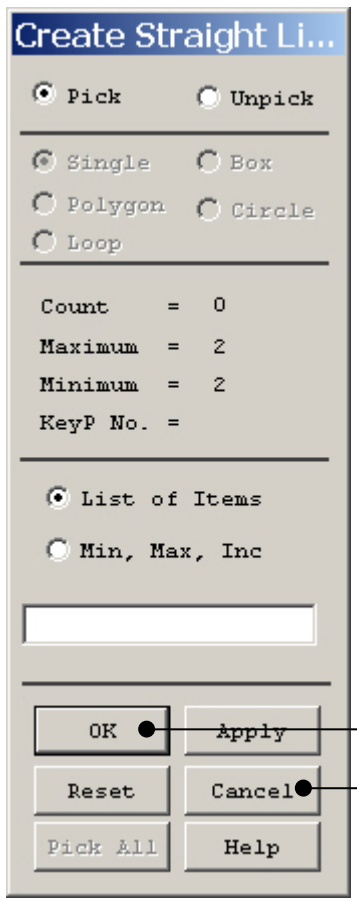
Press OK

Example - Lines

Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Create a line between Keypoint 1 and Keypoint 2 and
Create a line between Keypoint 2 and Keypoint 3.

L,1,2
L,2,3



HINT: By clicking with the right-hand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

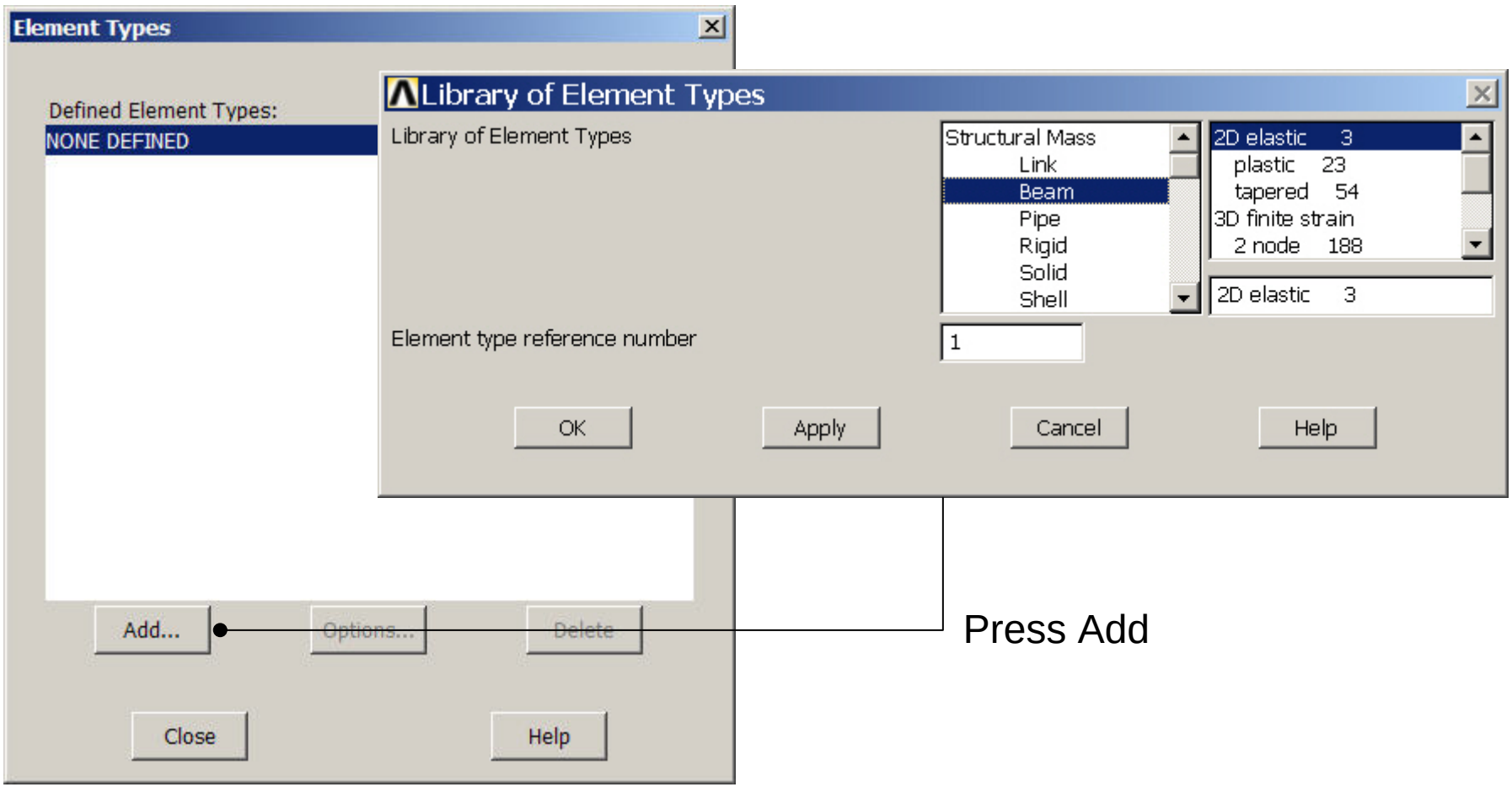
Pick: upward arrow

Unpick: downward arrow

Press OK or Cancel to finish selection

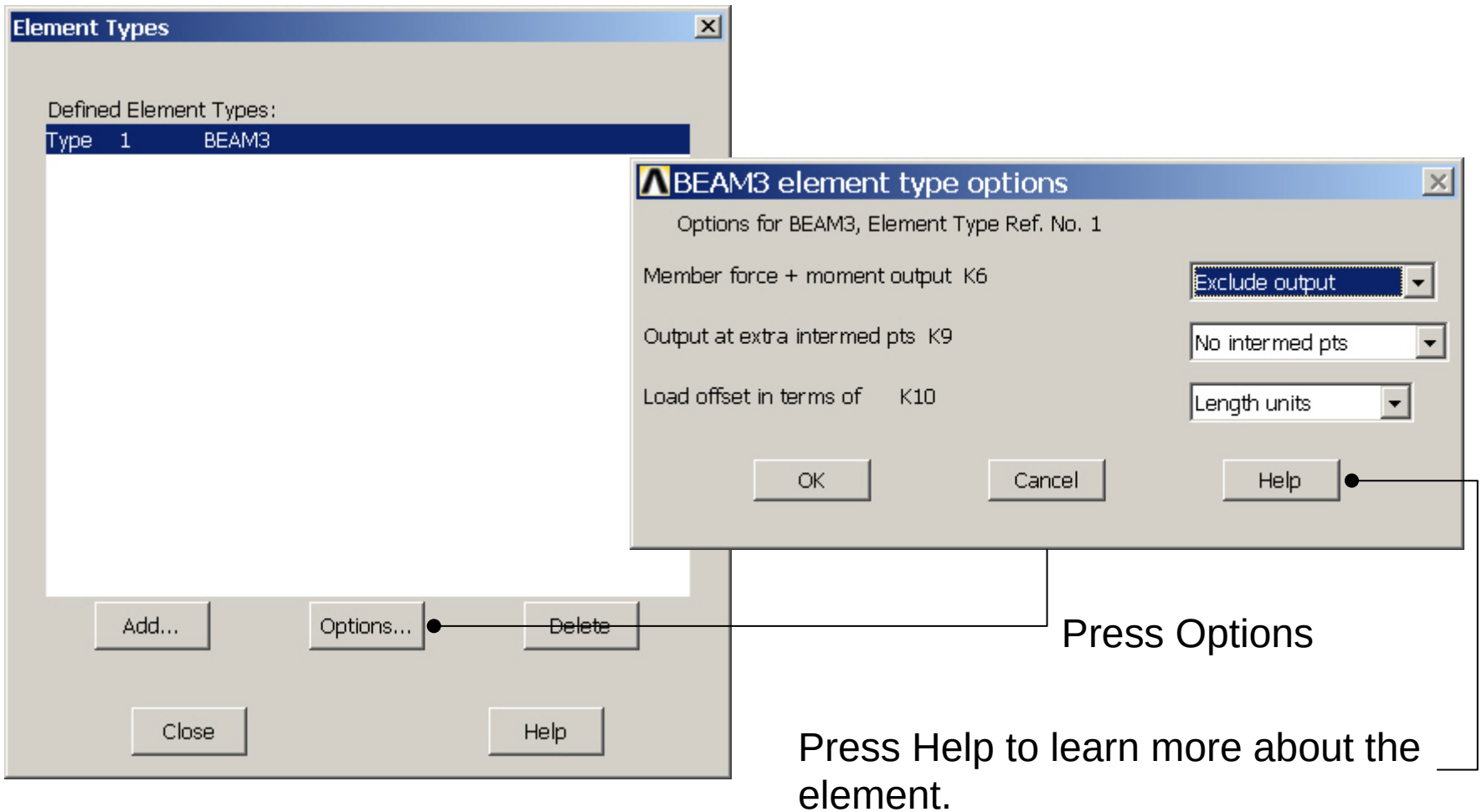
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



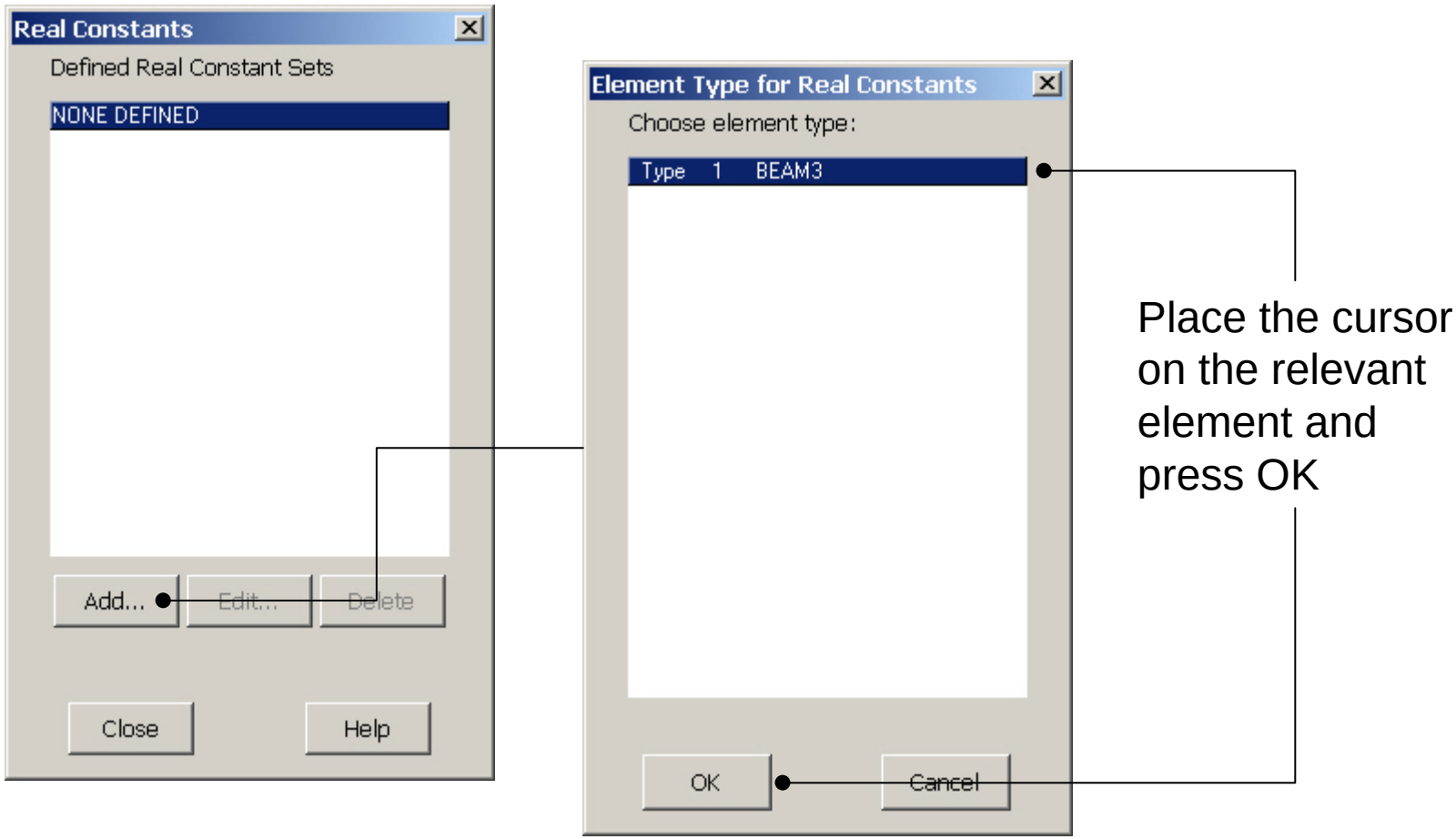
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



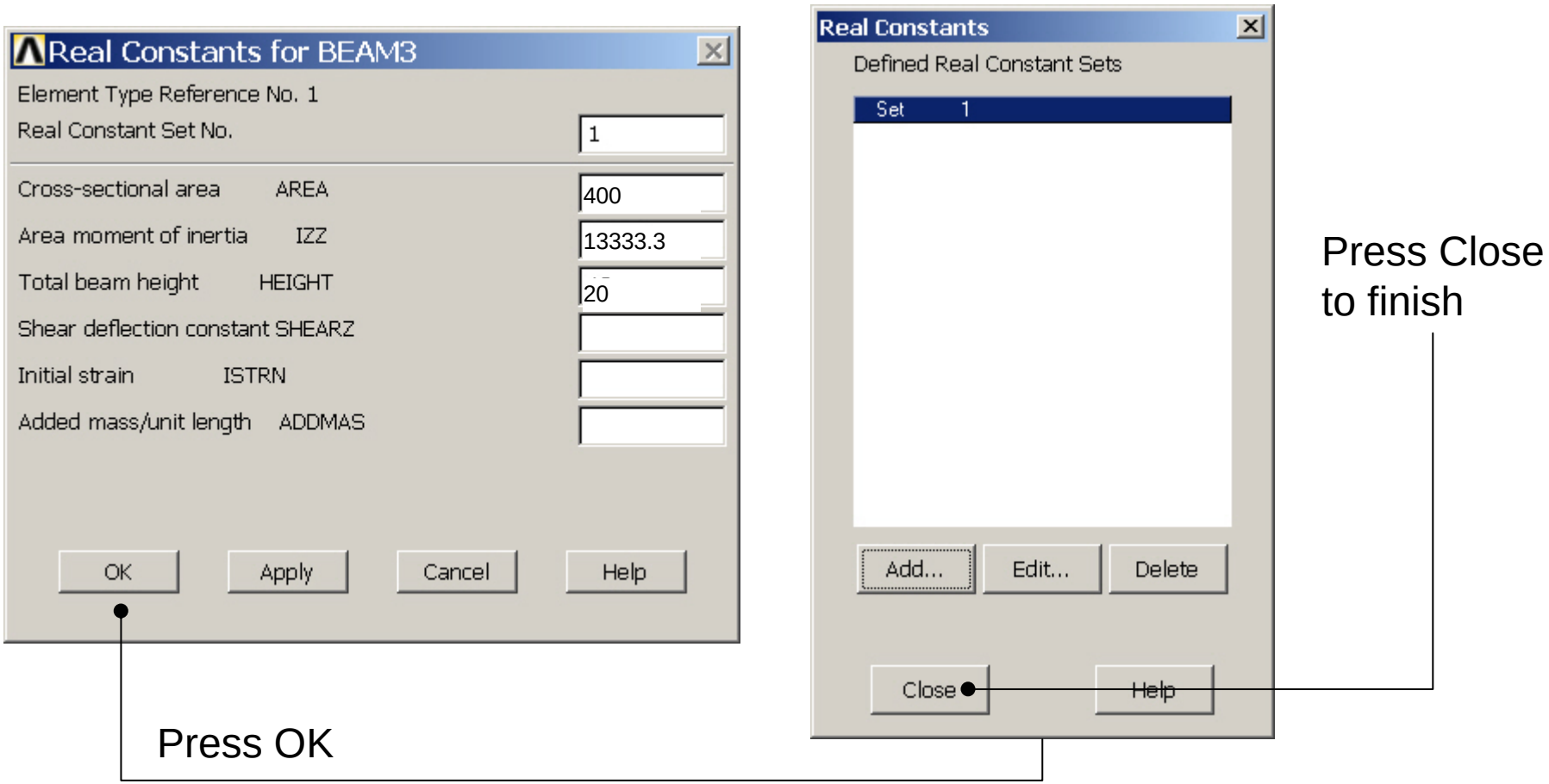
Example – Real Constants

Preprocessor > Real Constants > Add



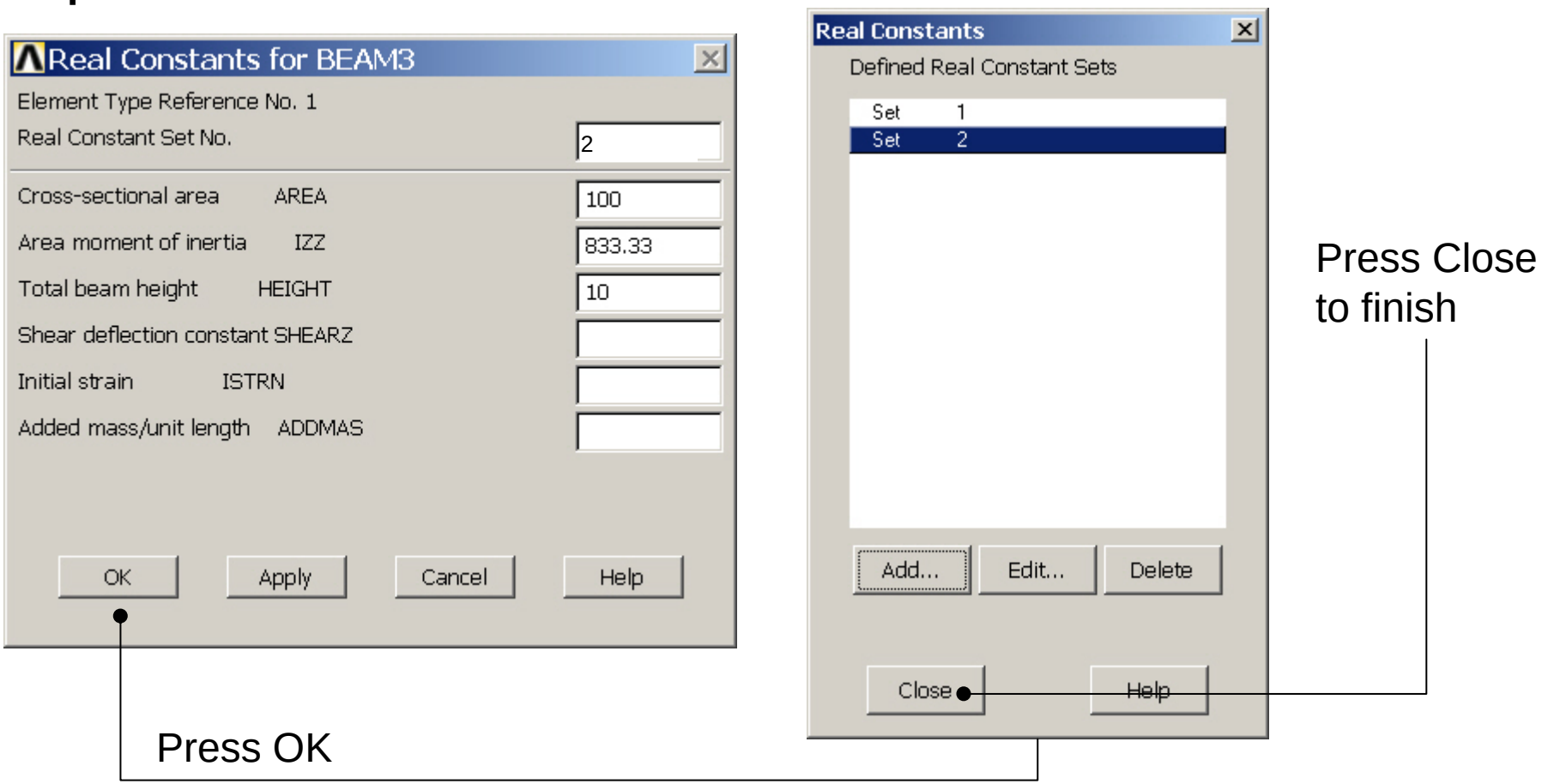
Example - Real Constants

Preprocessor > Real Constants > Add



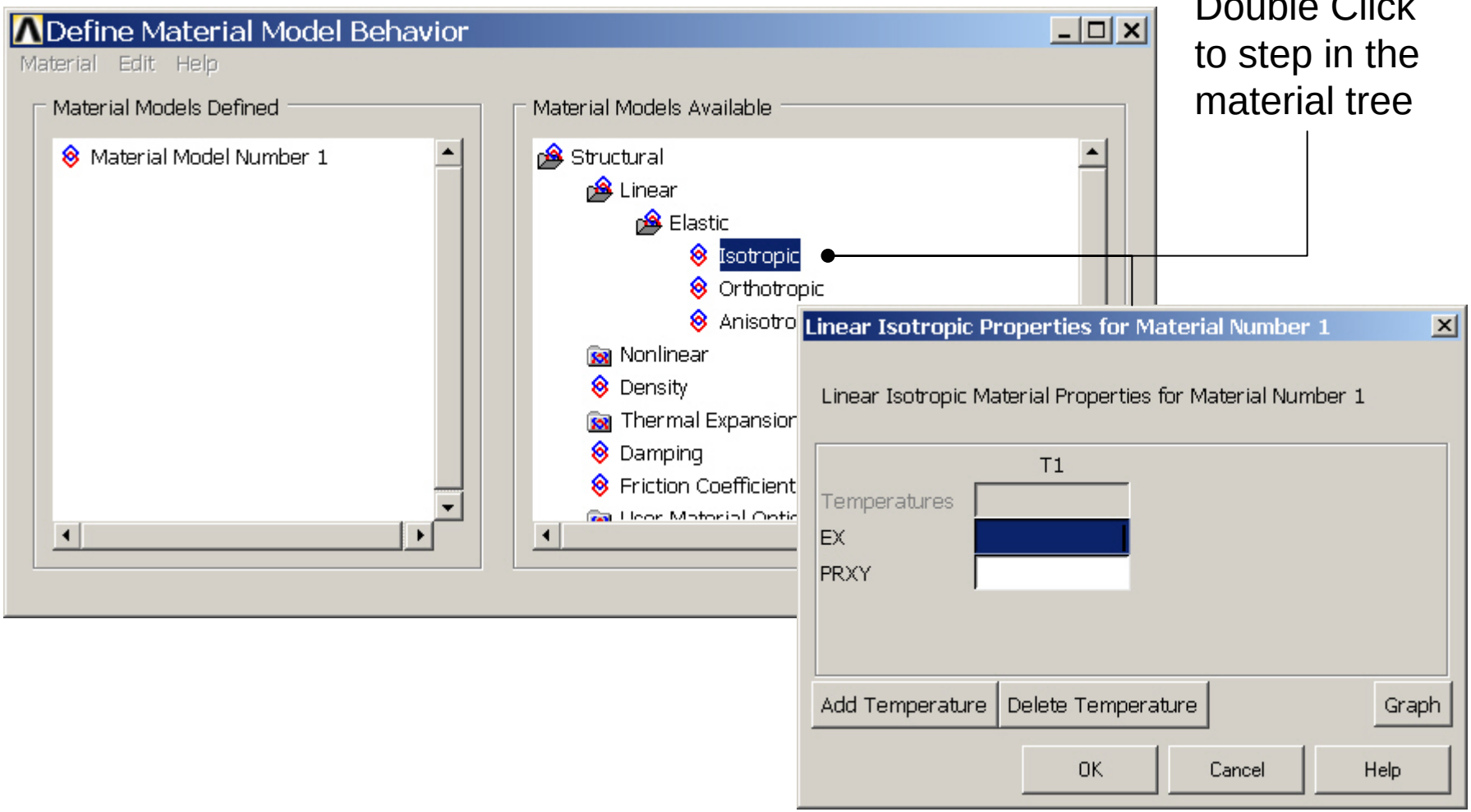
Example - Real Constants

Preprocessor > Real Constants > Add



Example - Material Properties

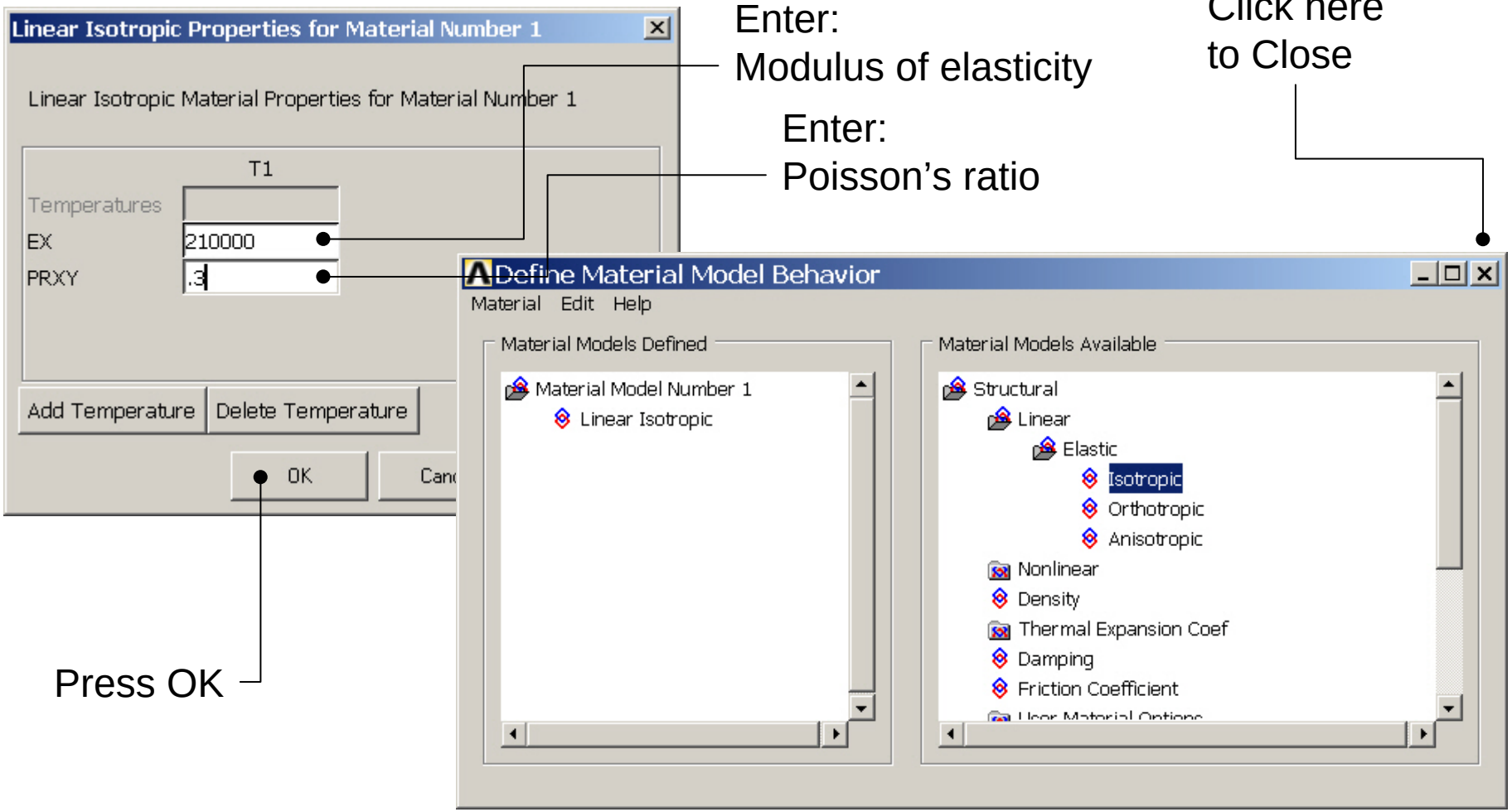
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

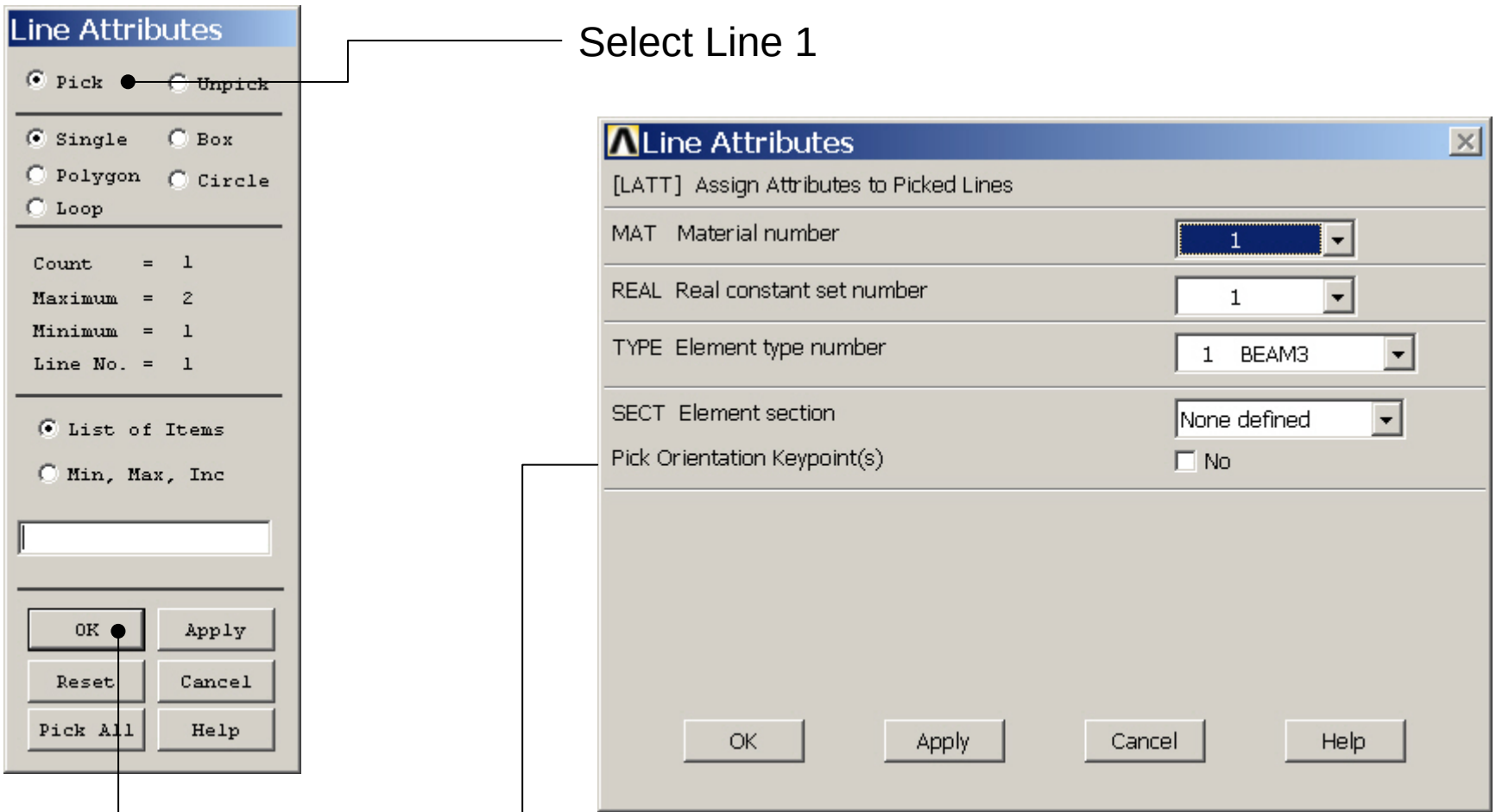
Example - Material Properties

Preprocessor > Material Props > Material Models



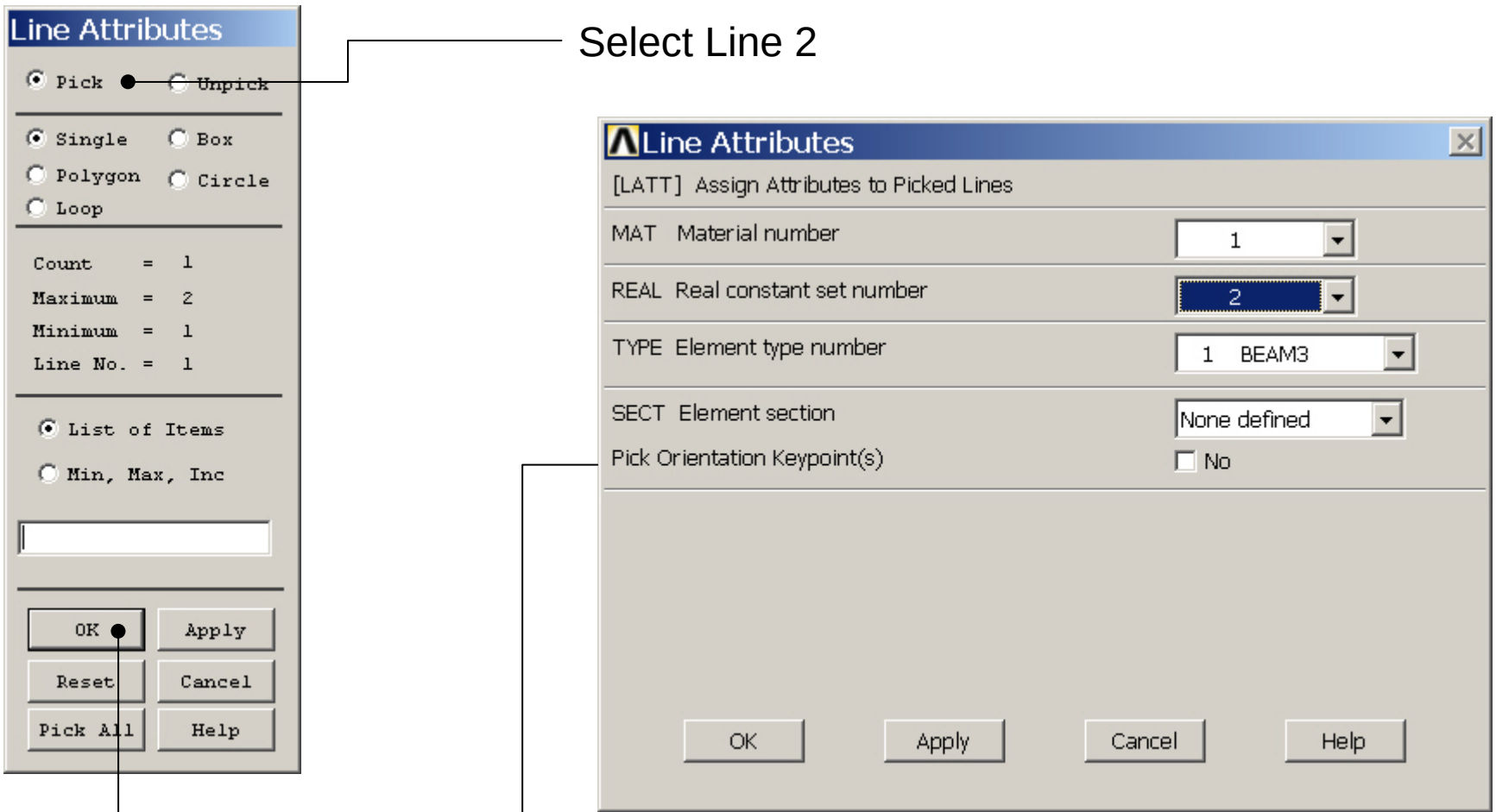
Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines



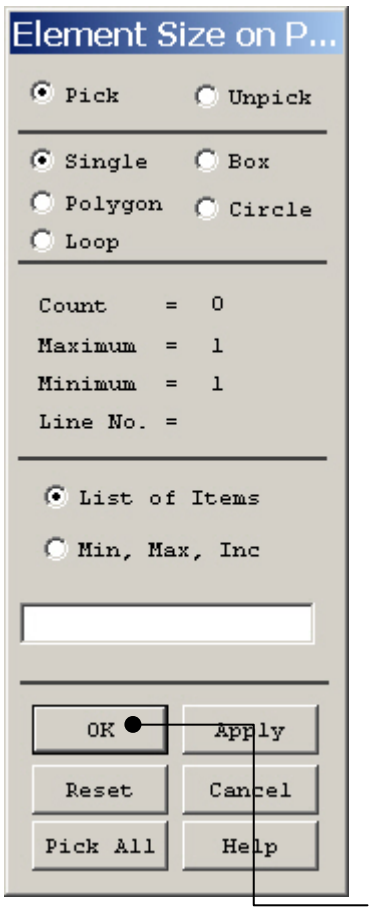
Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines

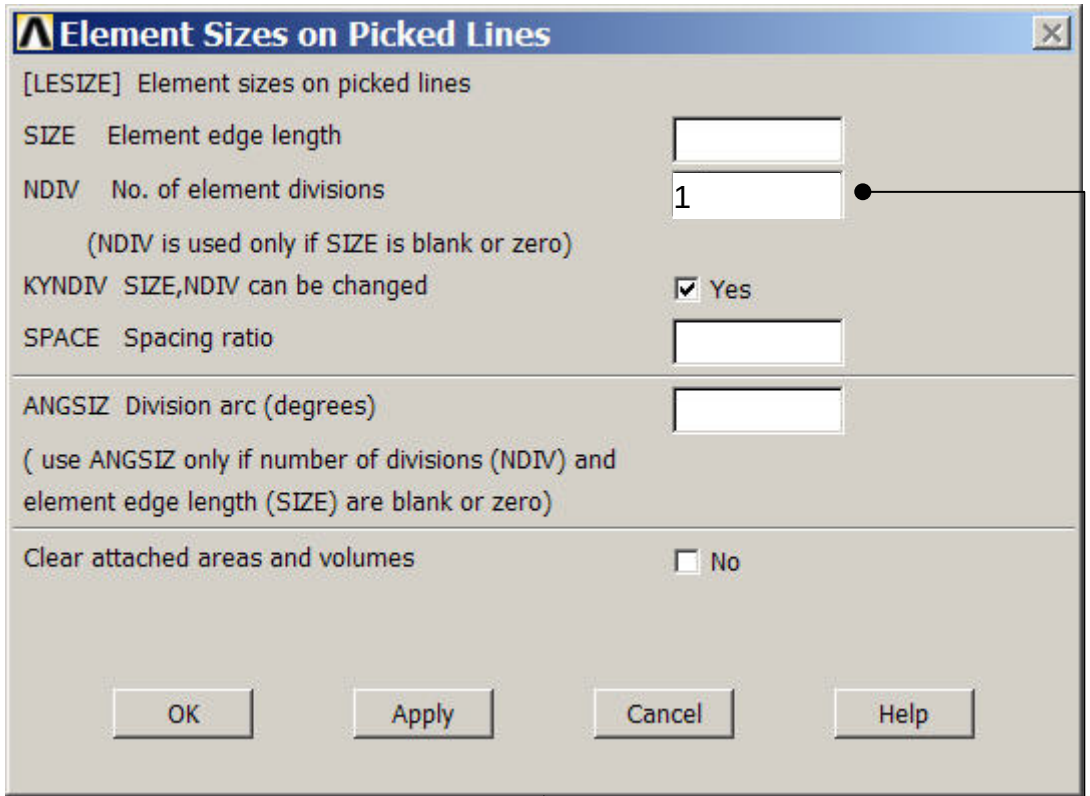


Example - Meshing

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines



Select/Pick Lines to specify mesh size for

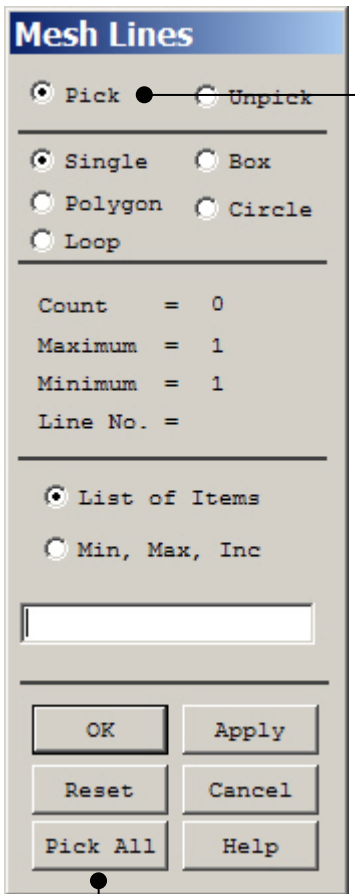


Press OK when finish with selection

Enter 1

Example - Meshing

Preprocessor > Meshing > Mesh > Lines



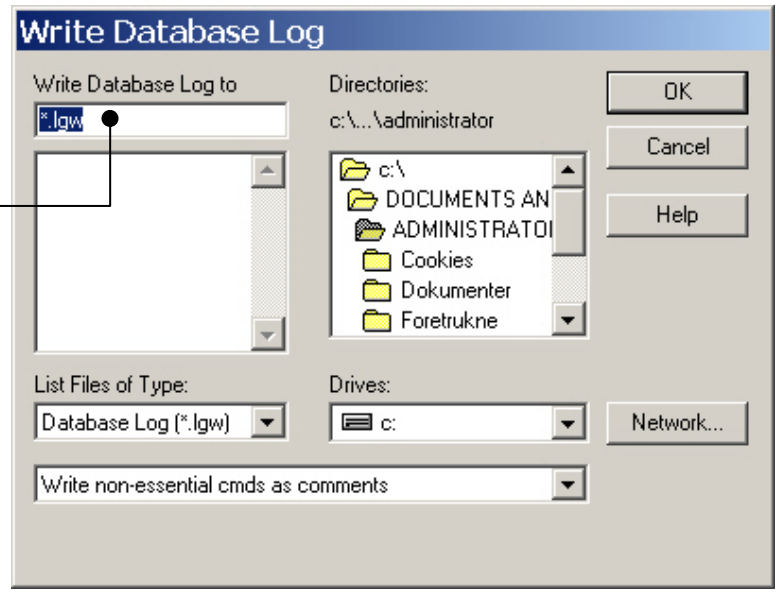
Select individual lines to be meshed by Picking

NB: It is often necessary to “Clear” the model for example if Element Type is to be changed

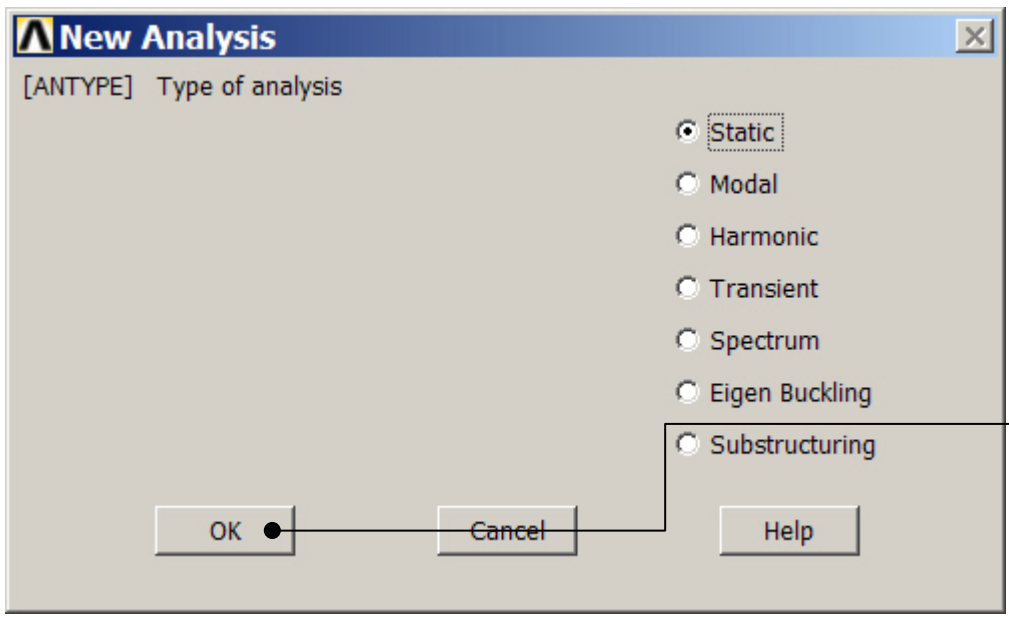
Select all lines defined to be meshed

Example – Analysis Type

File > Write DB log file
Enter “example0150.lgw”



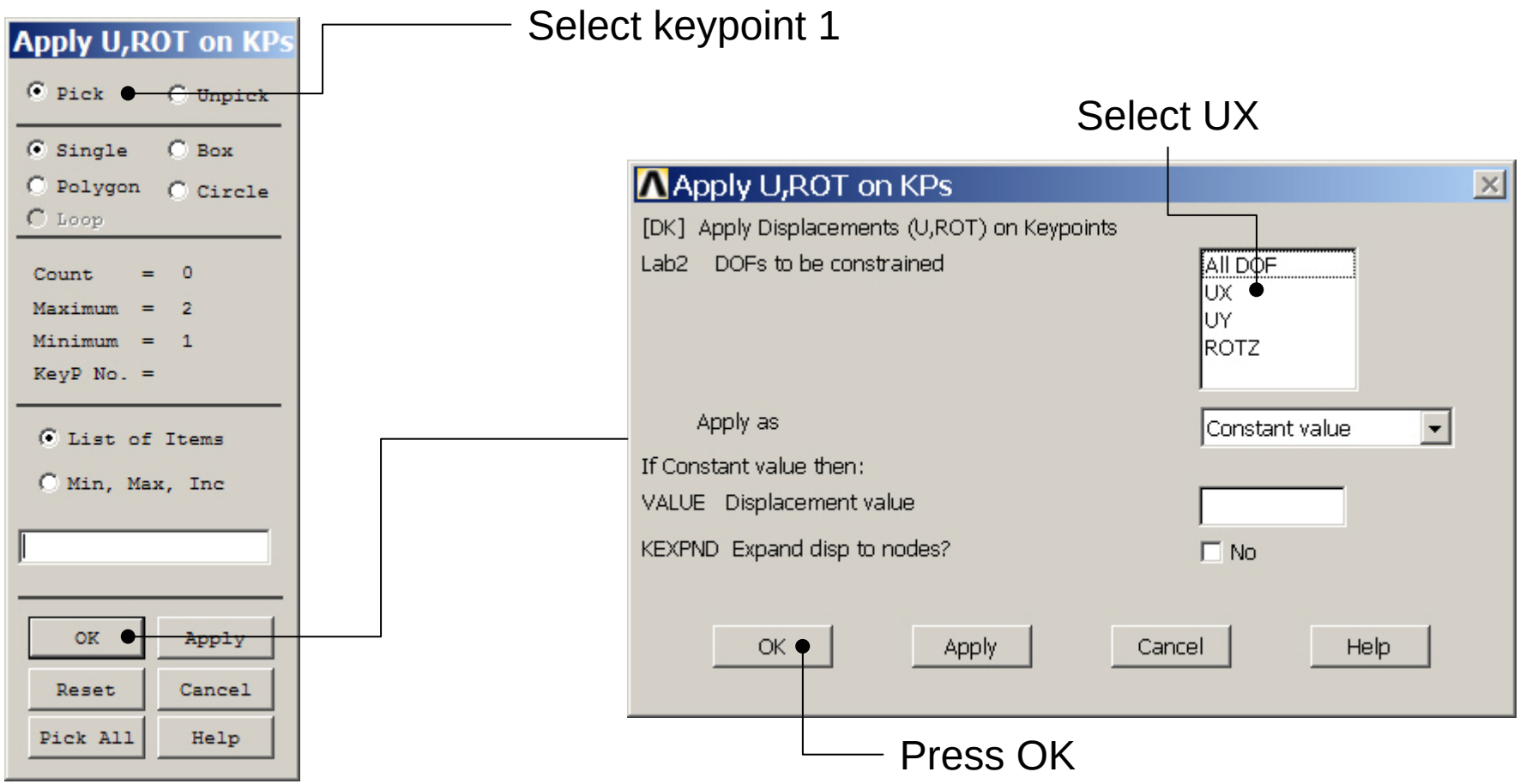
Solution > Analysis Type > New Analysis



Press OK

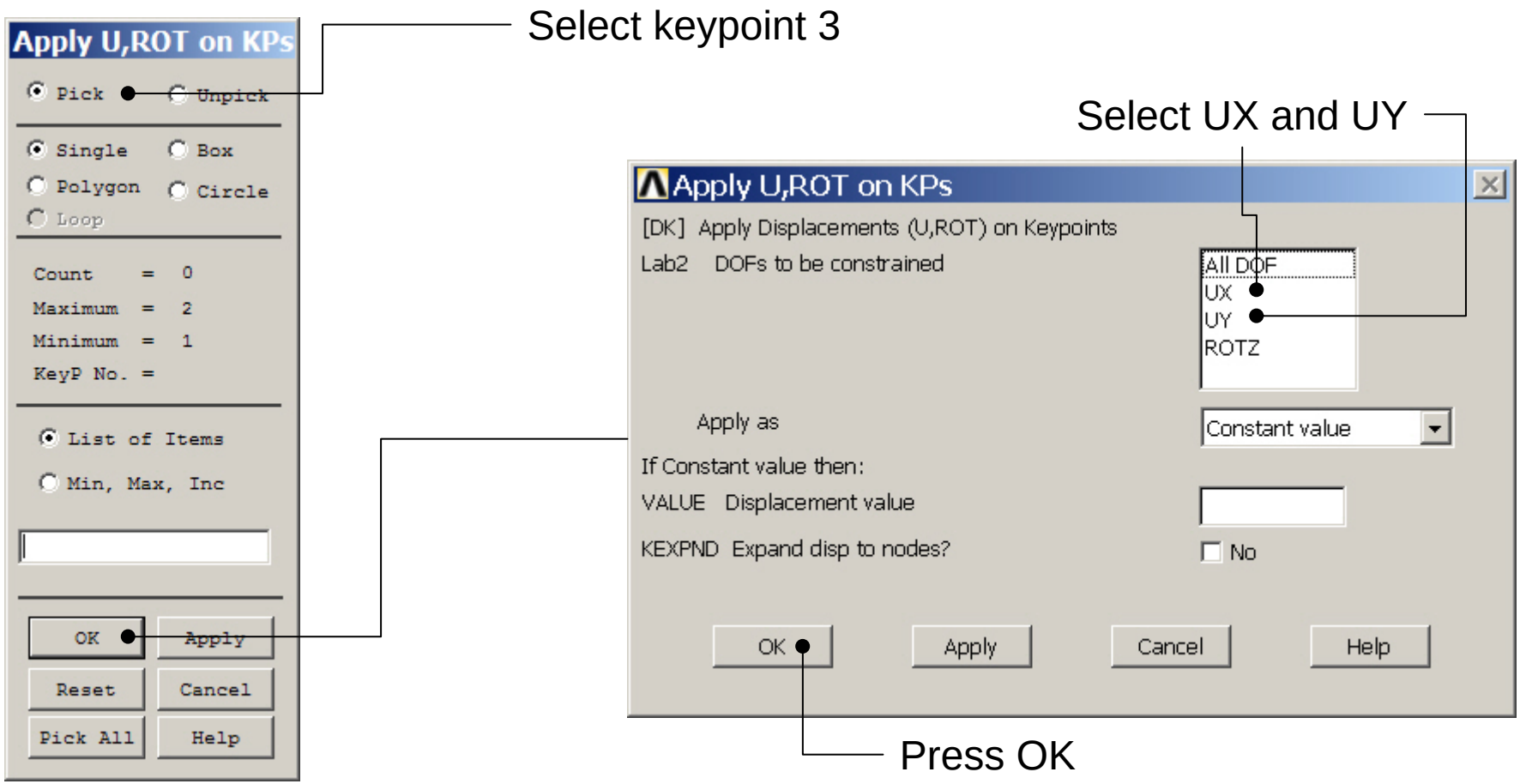
Example – Define Loads

Solution > Define Loads > Apply > Structural > Displacement > On Keypoints



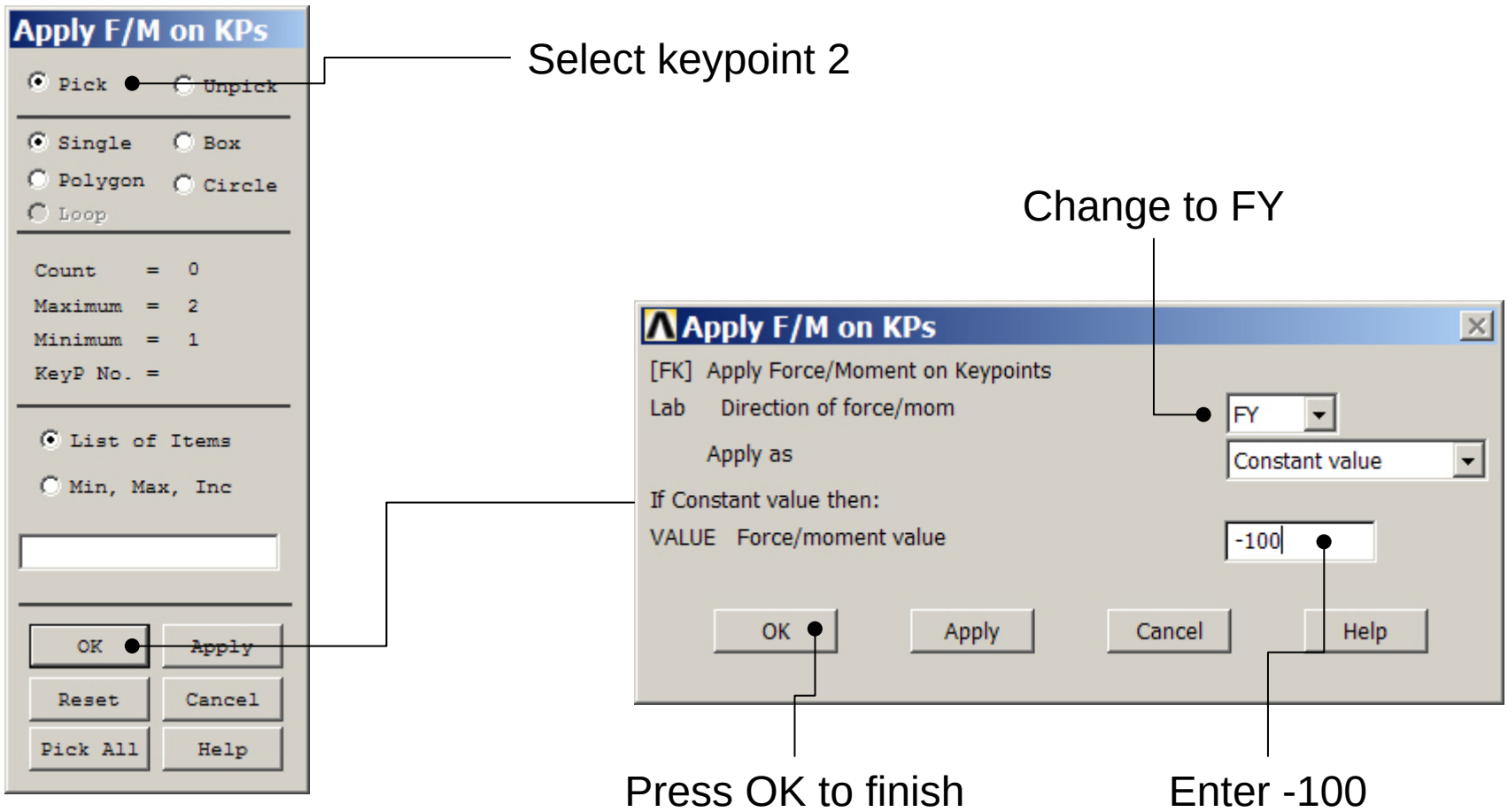
Example – Define Loads

Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

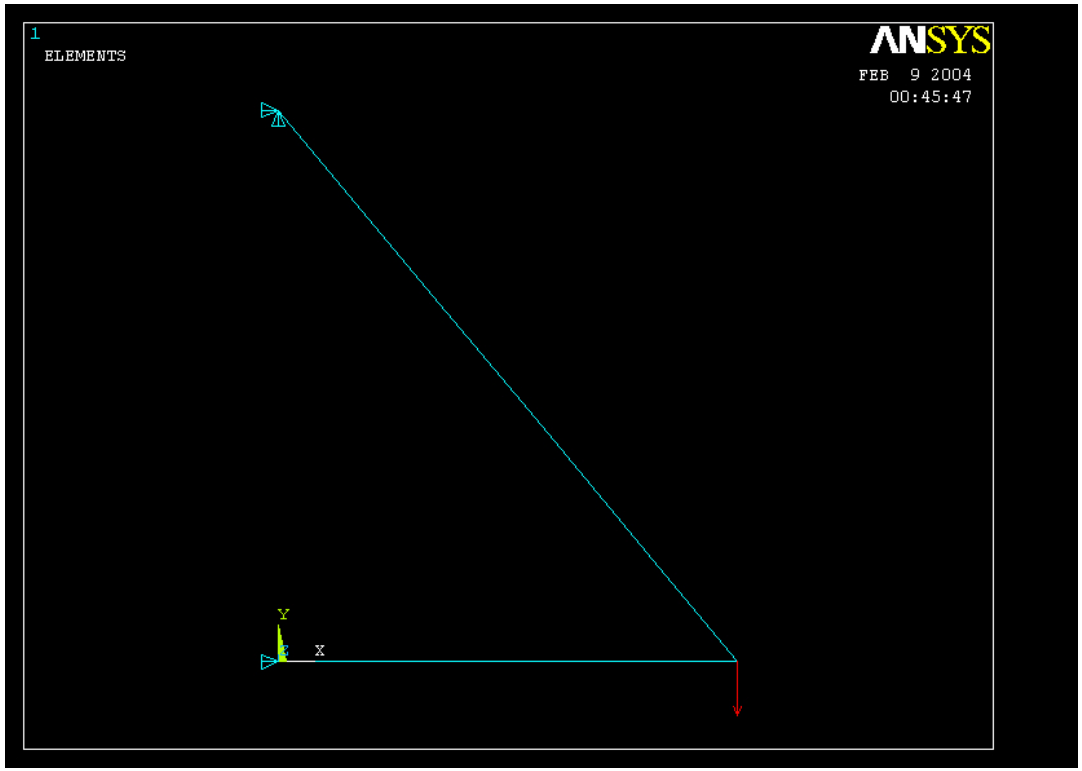


Example – Define Loads

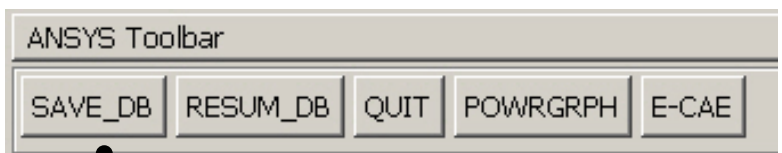
Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints



Example - Save



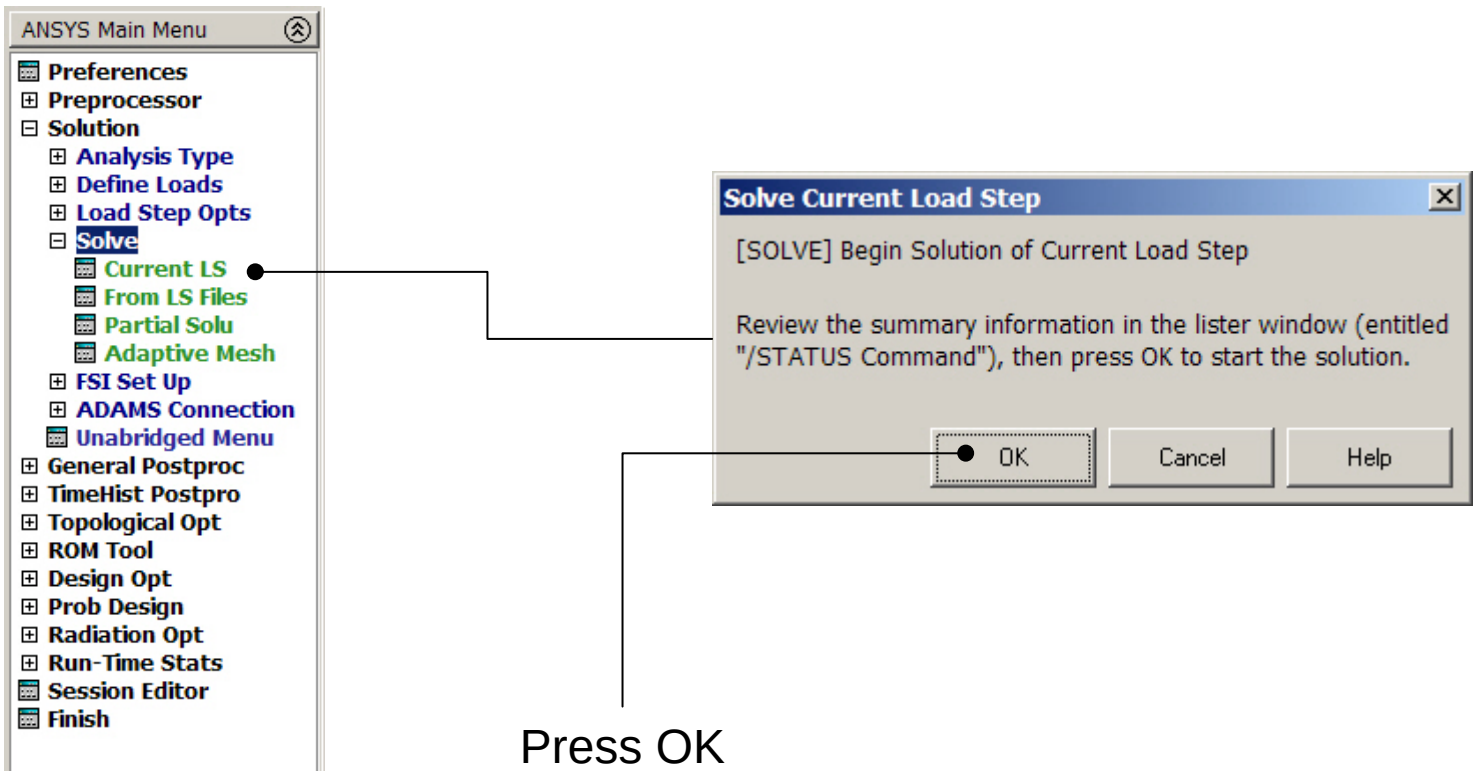
Display of Analysis model



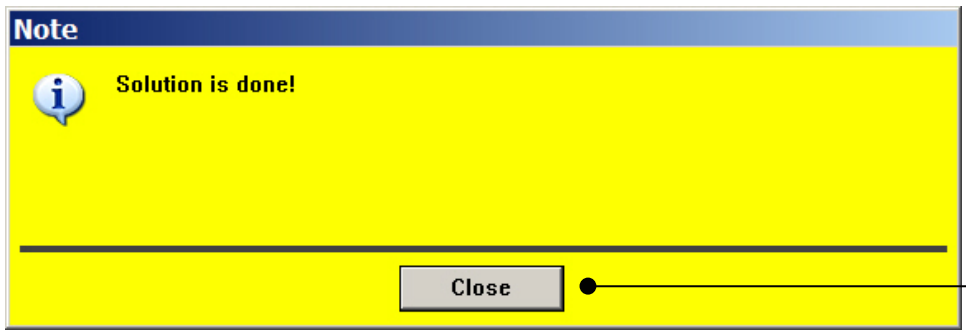
Save the model

Example - Solve

Solution > Solve > Current LS

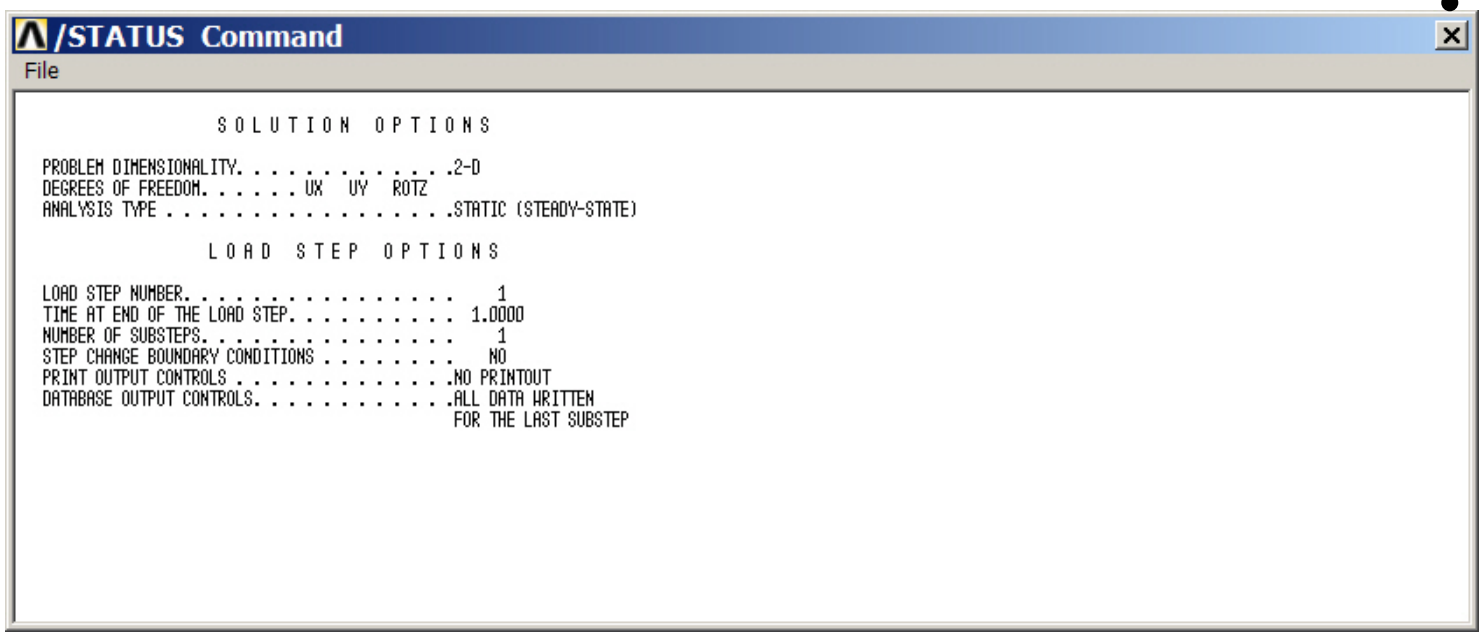


Example - Solve



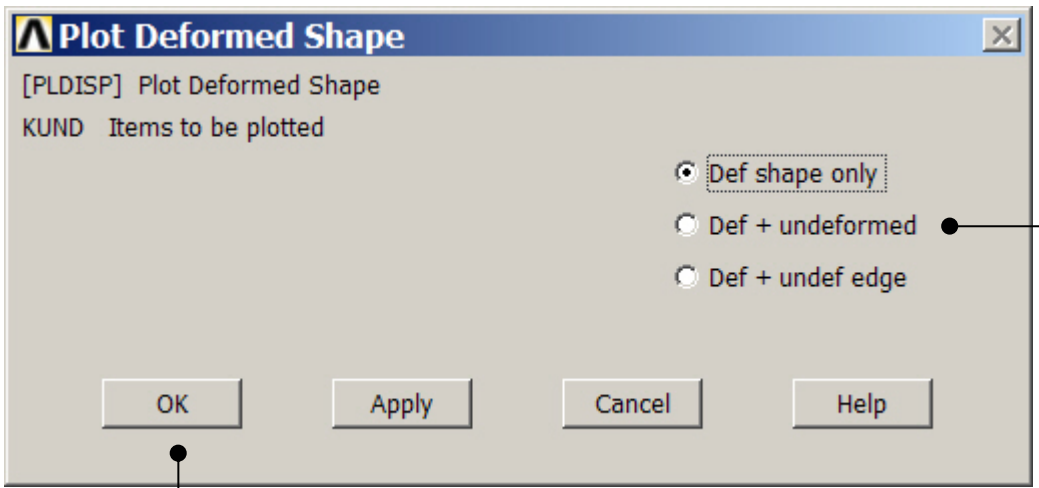
Press Close

Press here
to Close



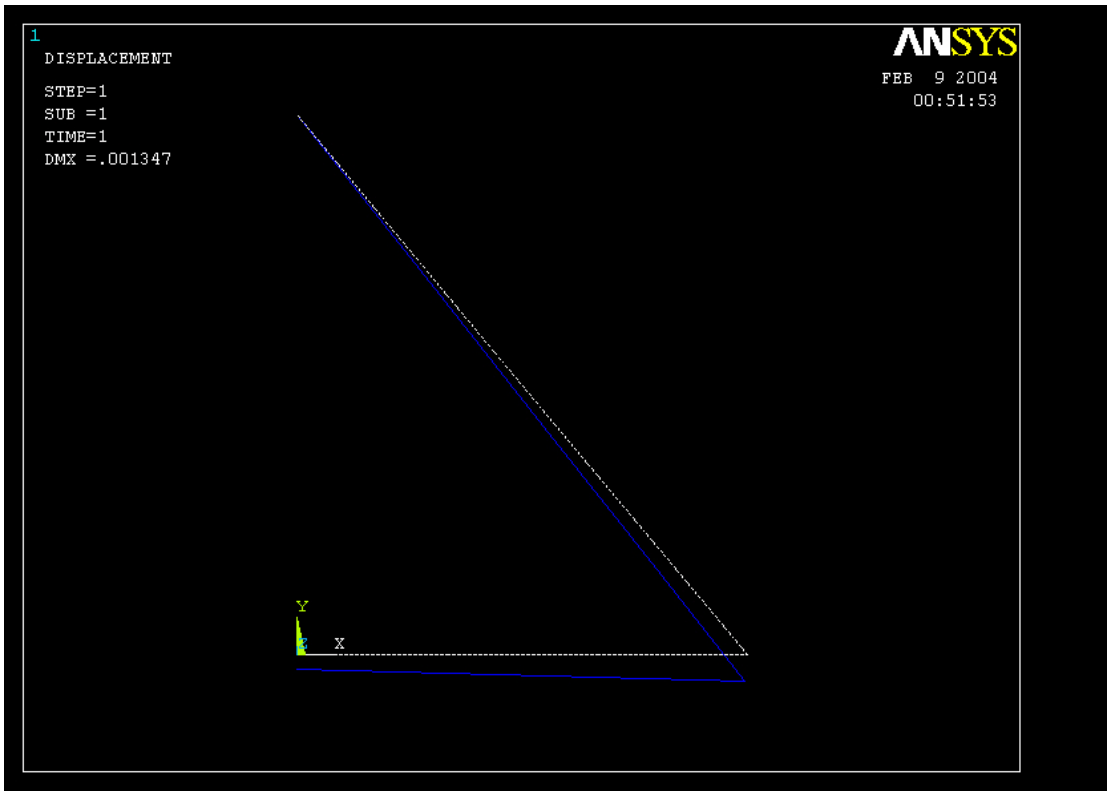
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Select “Def+undeformed” and Press OK

Example - PostProcessing

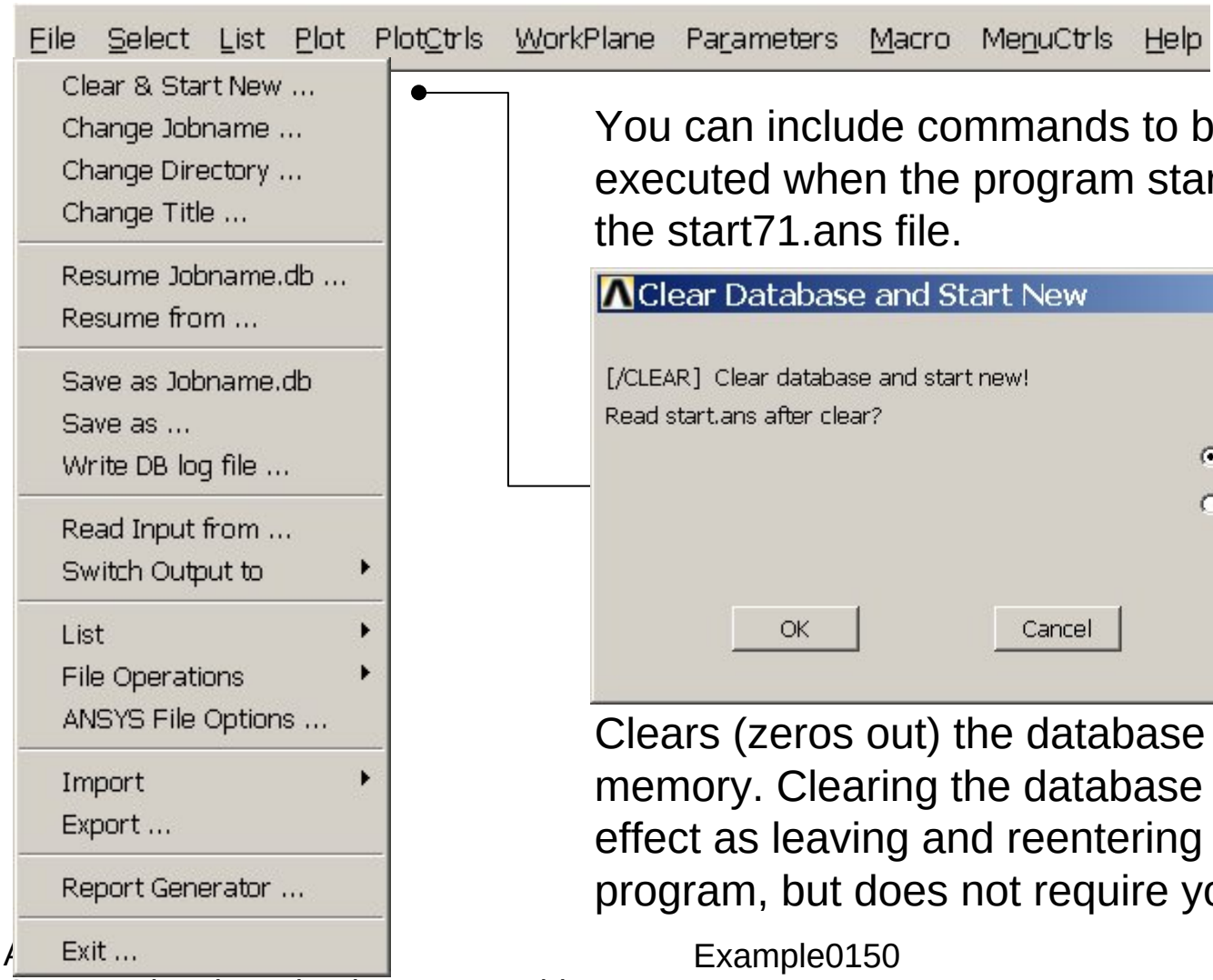


Read Maximum displacement: DMX

Example – Comments/Questions

- Try Link elements instead of beam elements?
- The “example0150.lgw” can be edited in “Notepad”
- Will the number of elements affect the solution?

File menu



You can include commands to be executed when the program starts up in the start71.ans file.

Clears (zeros out) the database stored in memory. Clearing the database has the same effect as leaving and reentering the ANSYS program, but does not require you to exit.