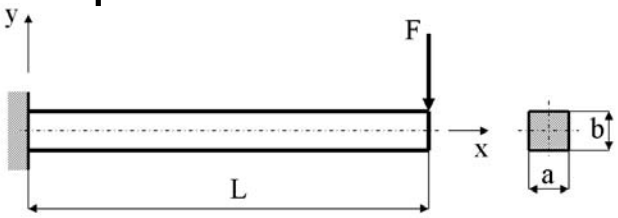
Course in ANSYS

Example0300

Example – Cantilever beam

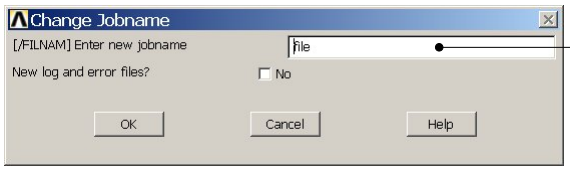


Objective:
Compute the maximum deflection and locate point of maximum deflection
Tasks:
How should this be modelled?
Compare results with results obtained from beam theory?
Topics:
Element type, Real constants, modeling, Plot results, output graphics

$E = 210000 N/mm^2$
$\nu = 0.3$
L = 100mm
a = 10mm
b = 10mm
F = 100N

Example - title

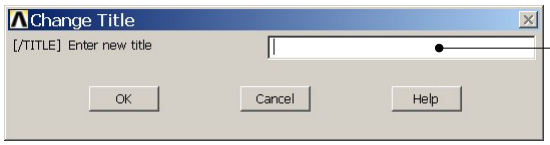
Utility Menu > File > Change Jobname — GUI
/jobname, Example0300 — Command line entry



Enter: Example0300

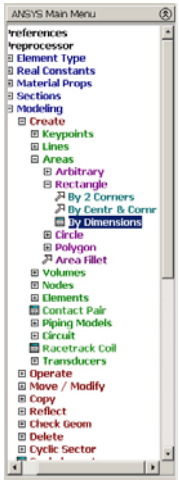
Utility Menu > File > Change Title
/title, Cantilever beam

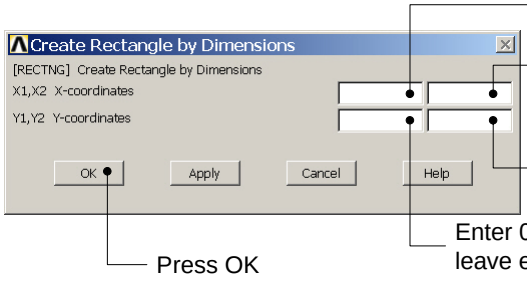
Enter: Cantilever beam



Example - Areas

Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions
Create an area given by X=(0,100) and Y=(0,10)





Enter 0 or leave empty

Enter 100

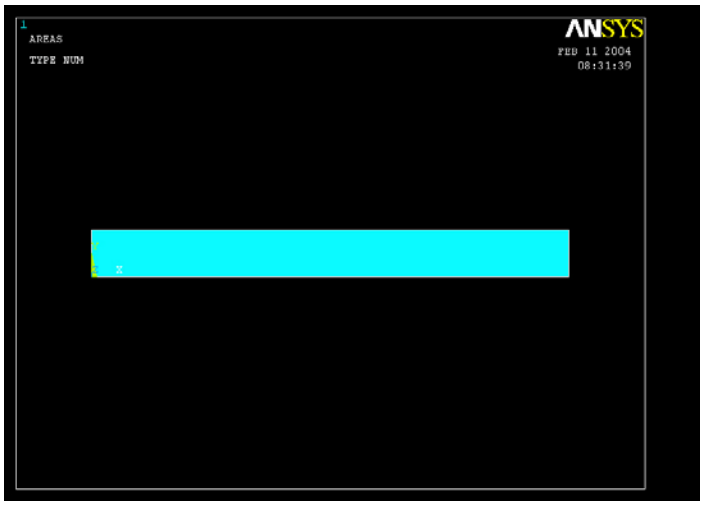
Enter 0 or leave empty

Enter 10

Press OK

Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

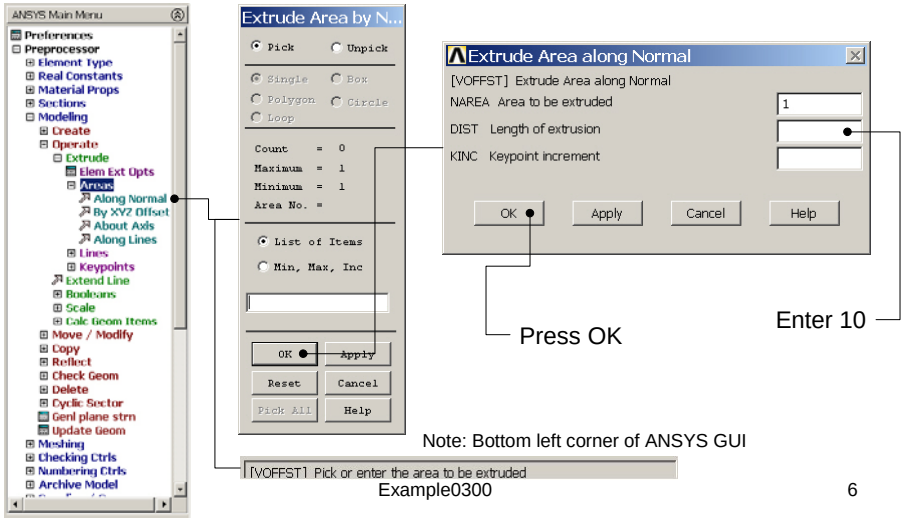
Example - Area



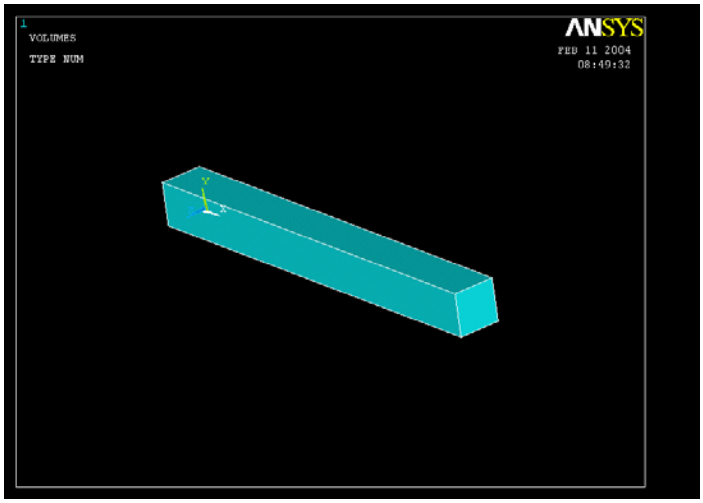
Example - Operate

Preprocessor > Modeling > Operate > Extrude > Areas > Along Normal

Create a volume by extruding the area 10 along its surface normal vector



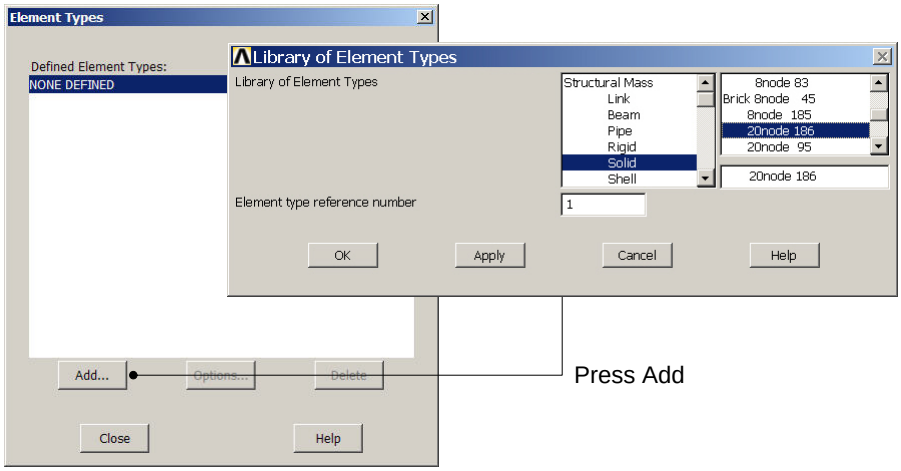
Example – Mouse rotate



Rotate by holding the Ctrl key down while using the right hand mouse button

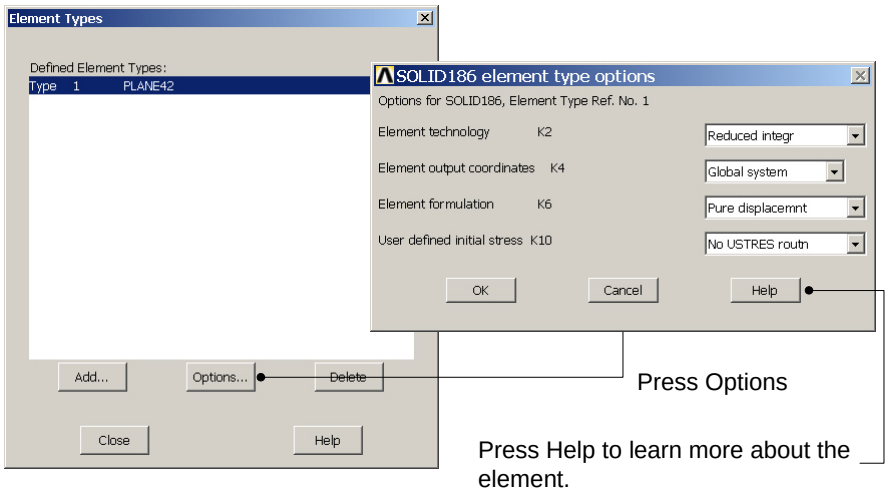
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete

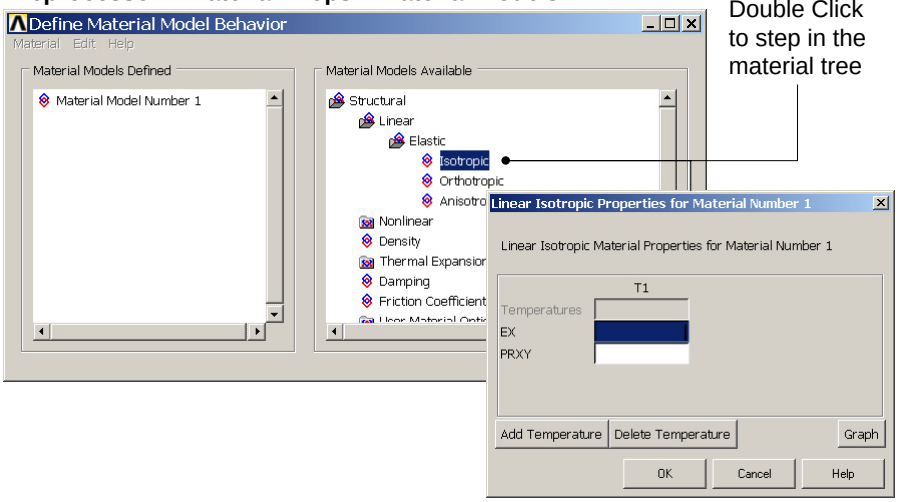


Example – Real Constants

No Real Constants are necessary for pure 3D solid models!

Example - Material Properties

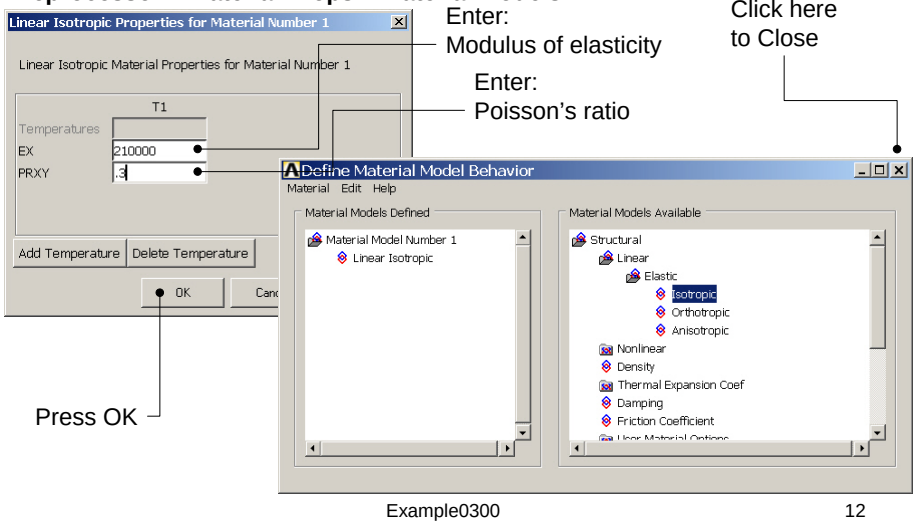
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

Example - Material Properties

Preprocessor > Material Props > Material Models



Enter: Modulus of elasticity

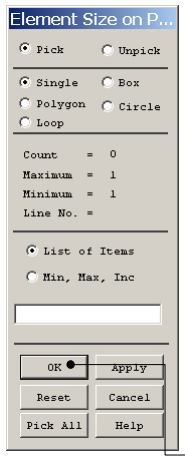
Enter: Poisson's ratio

Click here to Close

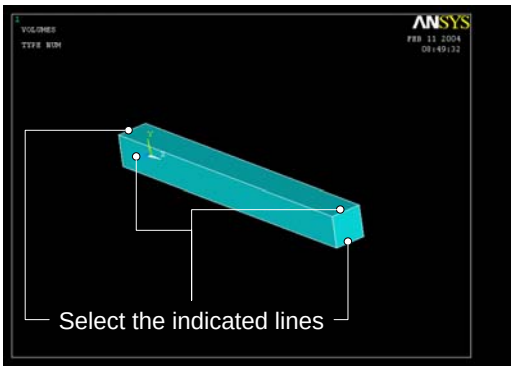
Press OK

Example - Meshing

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



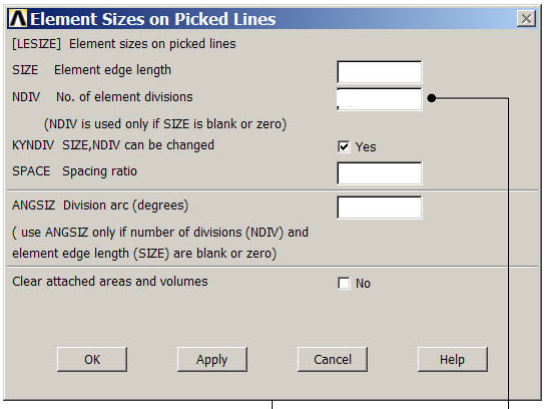
Select/Pick Lines to specify mesh size for



Press OK when finish with selection

Example - Meshing

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

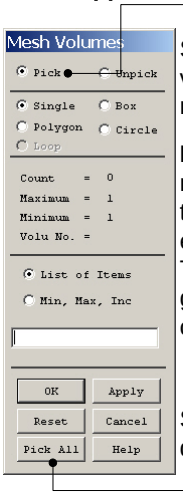


Enter 2

Press OK when finish with selection

Example - Meshing

Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 or 6 sided

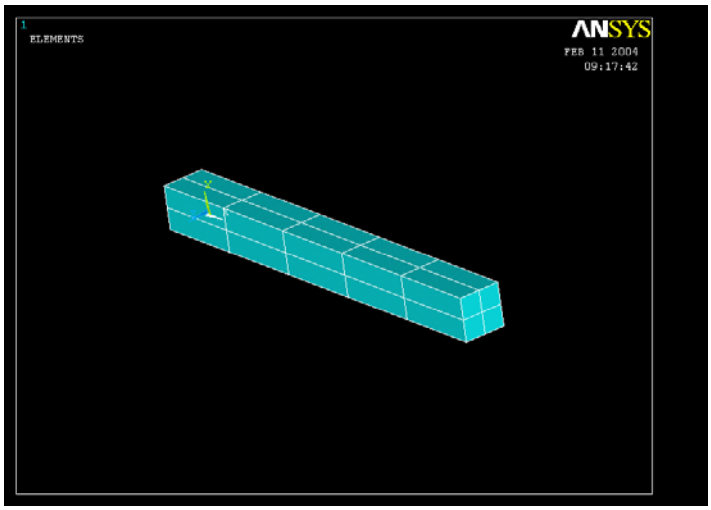


Select individual volumes to be meshed

NB: It is often necessary to "Clear" the model for example if Element Type or model geometry is to be changed

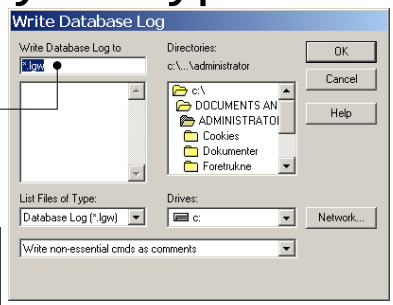
Select all volumes defined to be meshed

Example – 3D Mesh

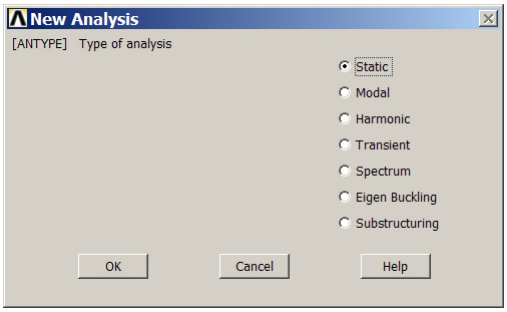


Example – Analysis Type

File > Write DB log file
Enter “example0300.lgw”



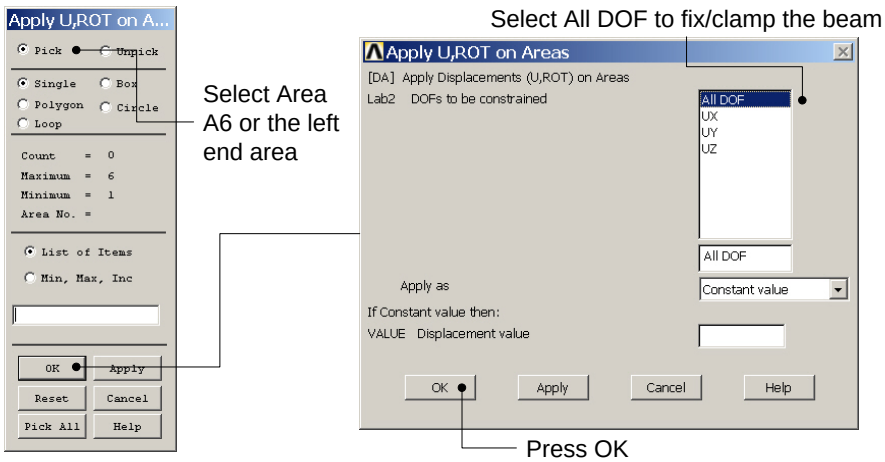
Solution > Analysis Type > New Analysis



Example – Define Loads

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

Select All DOF to fix/clamp the beam



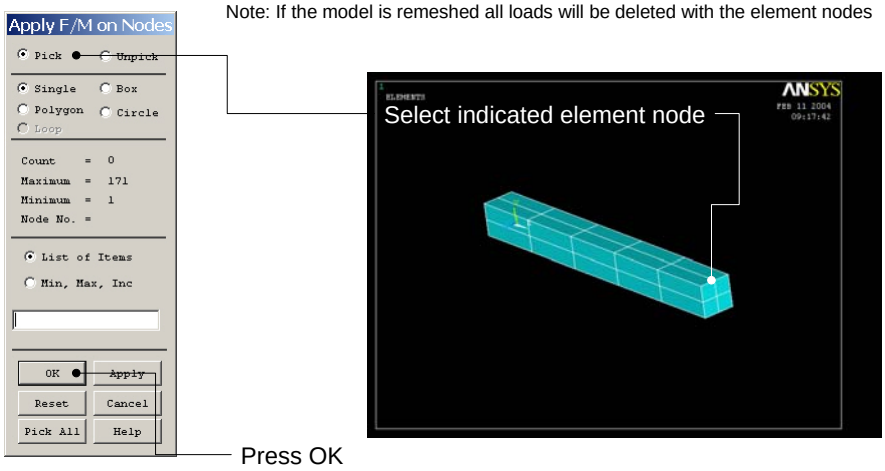
Select Area A6 or the left end area

Press OK

Example – Define Loads

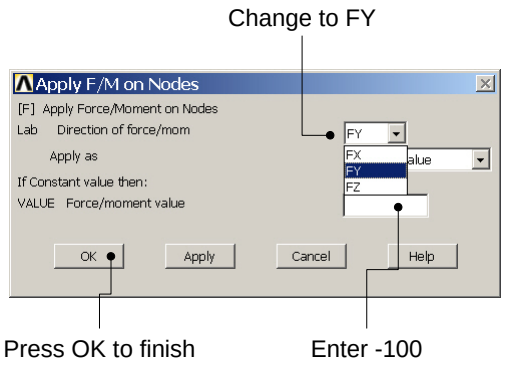
Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

Note: If the model is remeshed all loads will be deleted with the element nodes

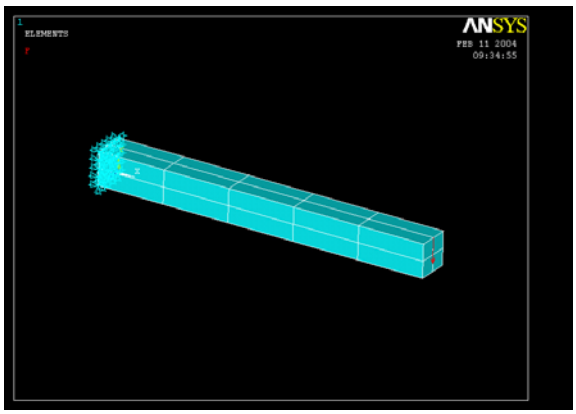


Example – Define Loads

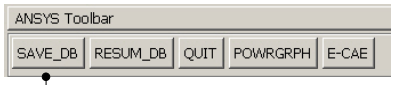
Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes



Example - Save



Display of Analysis model

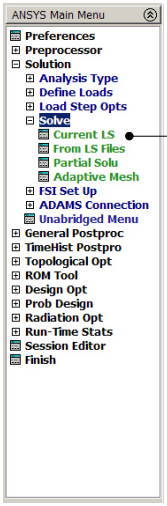


Save the model

Example0300 21

Example - Solve

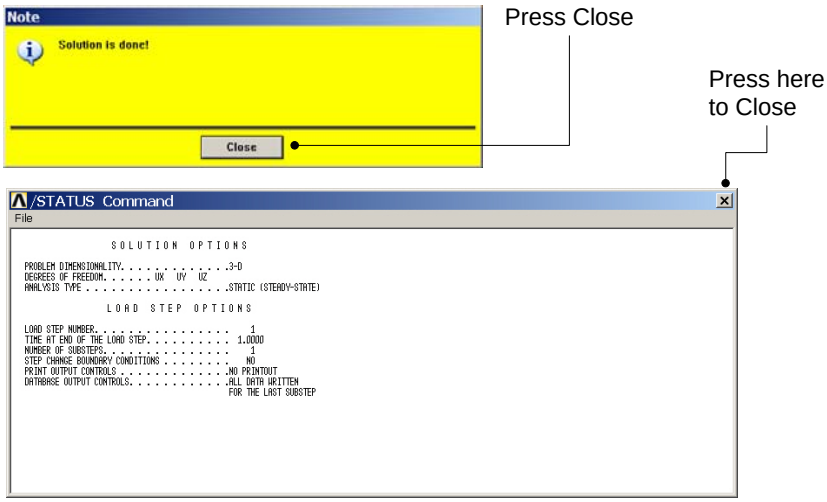
Solution > Solve > Current LS



Press OK

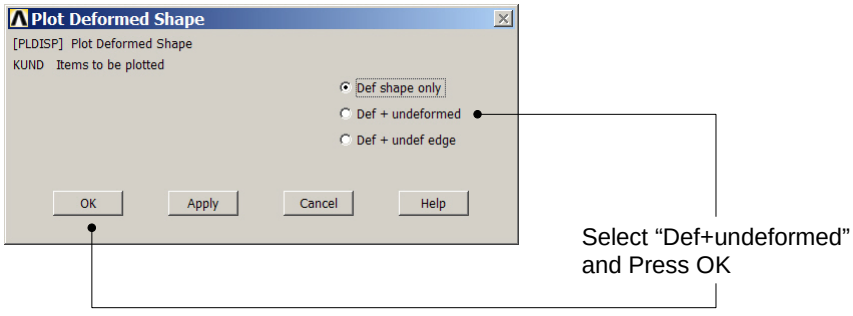
Example0300 22

Example - Solve

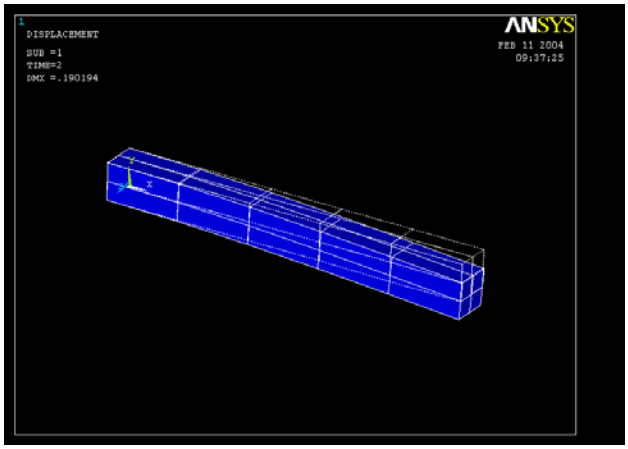


Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Example - PostProcessing

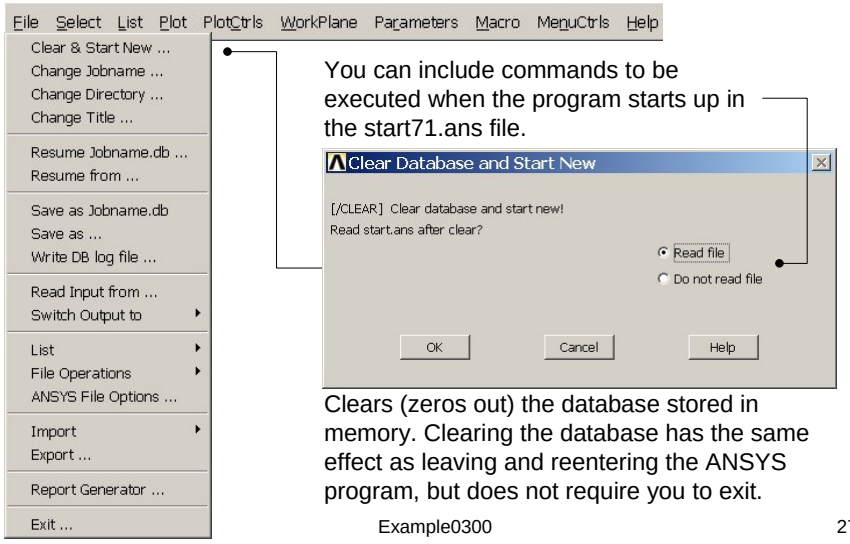


Read Maximum displacement: DMX

Example – Comments/Questions

- What did change compared to the Beam model?
- The “example0300.lgw” can be edited in “Notepad”
- What are the assumptions in this case?
- Will the shape or 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.