Design, Modelling and Manufacture 3

H7082

4

ANSYS Files

- frame.db
  Database file (binary). This file stores the geometry, boundary conditions and any solutions.

- frame.dbb
  Backup of the database file (binary).

- frame.err
  Error file (text). Listing of all error and warning messages.

- frame.out
  Output of all ANSYS operations (text). This is what normally scrolls in the output window during an ANSYS session.

- frame.log
  Logfile or listing of ANSYS commands (text). Listing of all equivalent ANSYS command line commands used during the current session.

Ansys Log file example

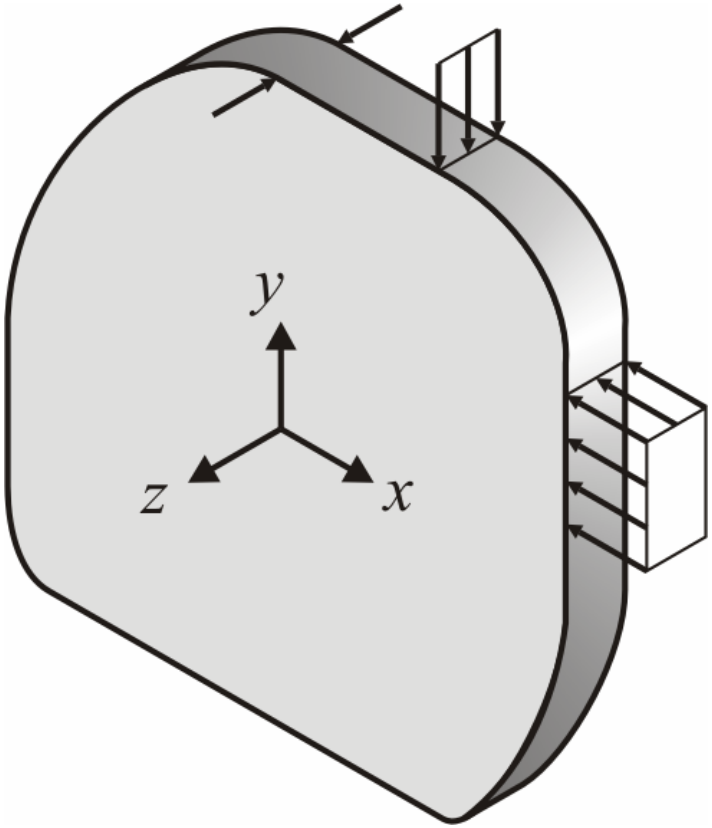
```
!/BATCH
FINISH
/CLEAR,NOSTART
/TITLE,2-D Truss Analysis
!*
KEYW,PR_SET,1
KEYW,PR_STRUC,1
KEYW,PR_THERM,0
KEYW,PR_ELMAG,0
KEYW,PR_FLUID,0
KEYW,PR_MULTI,0
KEYW,PR_CFD,0
KEYW,LSDYNA,0
/PMETH,OFF
!*
/PREP7
!*
ET,1,LINK1
!*
!*
R,1,.5, ,
!*
!*
UIMP,1,EX, , ,30e6,
UIMP,1,NUXY, , ,0.3,
!*
SAVE
N,1,6,0,,,,,
N,2,18,0,,,,,
N,3,0,8,,,,,
N,4,12,8,,,,,
N,5,24,8,,,,,
NLIST,ALL, , , ,NODE,NODE,NODE
/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,1
/PNUM,SVAL,0
/NUM,0
!*
/PNUM,ELEM,0
/REPLOT
!*
SAVE
FLST,2,2,1
FITEM,2,1
FITEM,2,2
E,P51X
FLST,2,2,1
FITEM,2,1
FITEM,2,3
E,P51X
FLST,2,2,1
FITEM,2,1
FITEM,2,4
E,P51X
FLST,2,2,1
FITEM,2,2
FITEM,2,4
E,P51X
FLST,2,2,1
FITEM,2,2
FITEM,2,5
E,P51X
FLST,2,2,1
FITEM,2,3
FITEM,2,4
E,P51X
FLST,2,2,1
FITEM,2,4
FITEM,2,5
E,P51X
/PNUM,KP,0
/PNUM,LINE,0
/PNUM,AREA,0
/PNUM,VOLU,0
/PNUM,NODE,1
/PNUM,SVAL,0
/NUM,0
!*
/PNUM,ELEM,1
/REPLOT
!*
EPLOT
SAVE
```

```
/SOLU
FINISH
/SOLU
FLST,2,1,1,ORDE,1
FITEM,2,5
D,P51X, , , , , ,ALL
FLST,2,1,1,ORDE,1
FITEM,2,2
D,P51X, , , , , ,UY
/SOLU
FLST,2,1,1,ORDE,1
FITEM,2,3
F,P51X,FY,-2000,
FLST,2,1,1,ORDE,1
FITEM,2,4
F,P51X,FY,-1000,
!*
/PSF,DEFA, ,1
/PBF,DEFA, ,1
/PSYMB,CS,0
/PSYMB,NDIR,0
/PSYMB,ESYS,0
/PSYMB,LDIR,0
/PSYMB,ECON,0
/PSYMB,PCONV,
/PSYMB,LAYR,0
!*
/PBC,ALL,,1
/PBC,NFOR,,0
/PBC,NMOM,,0
/PBC,RFOR,,0
/PBC,RMOM,,0
/REP
!*
SAVE
```

```
/STAT,SOLU
SOLVE
/POST1
FINISH
/POST1
PLDISP,1
ETABLE,Axial,LS,  1
ETABLE,Memfor,SM
ISC,  1
PRETAB,AXIAL,ME
MFOR
!*
PRRSOL,F
!*
/PSF,DEFA, ,1
/PBF,DEFA, ,1
/PSYMB,CS,0
/PSYMB,NDIR,0
/PSYMB,ESYS,0
/PSYMB,LDIR,0
/PSYMB,ECON,0
/PSYMB,PCONV,
/PSYMB,LAYR,0
!*
/PBC,U,0
/PBC,ROT, ,0
/PBC,TEMP, ,0
/PBC,PRES, ,0
/PBC,V, ,0
/PBC,ENKE, ,0
/PBC,ENDS, ,0
/PBC,VOLT, ,0
/PBC,MAG, ,0
```

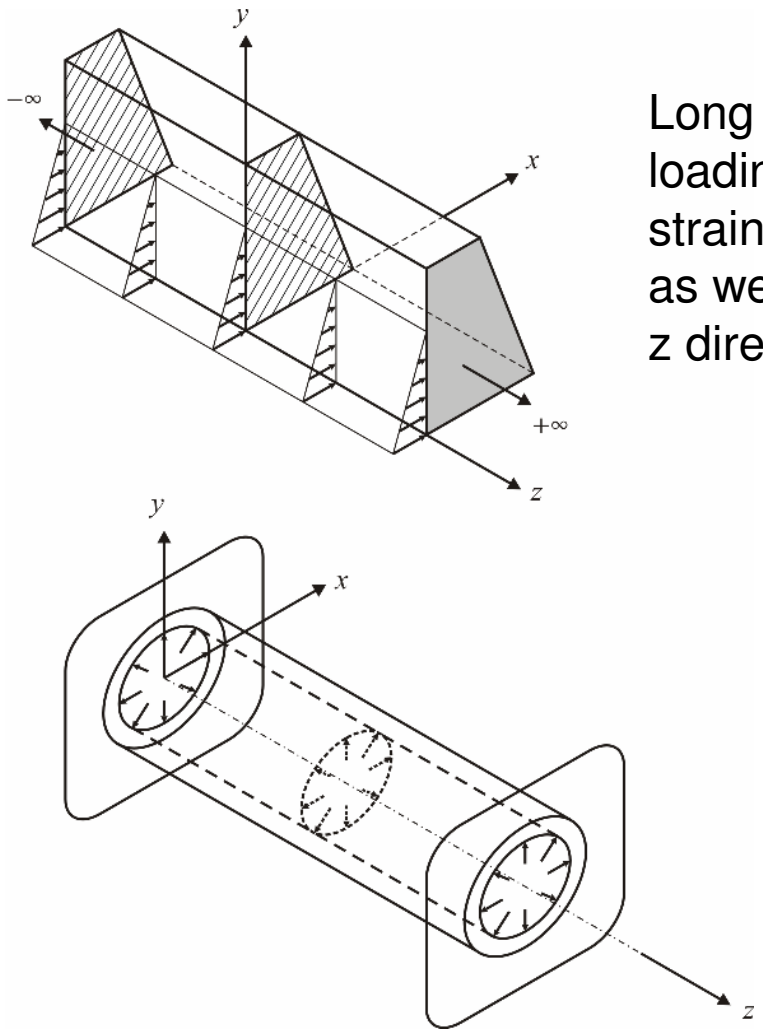
```
/PBC,A, ,0
/PBC,CHRG, ,0
/PBC,F, ,0
/PBC,M, ,0
/PBC,HEAT, ,0
/PBC,FLOW, ,0
/PBC,AMPS, ,0
/PBC,FLUX, ,0
/PBC,CSG, ,0
/PBC,ACEL, ,0
/PBC,OMEG, ,0
/PBC,MAST, ,0
/PBC,CP, ,0
/PBC,CE, ,0
/PBC,NFOR, ,0
/PBC,NMOM, ,0
/PBC,RFOR, ,1
/PBC,RMOM, ,0
/PBC,PATH, ,0
/REP

PRETAB,AXIAL
```

Plane Stress Conditions



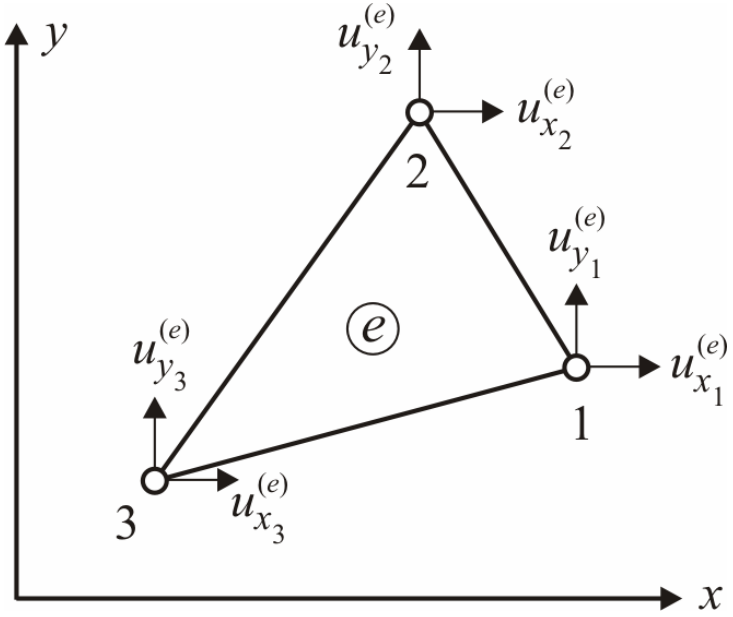
Plane stress is used for thin components subjected only to in-plane external loading (no lateral loads and as a result only u_x and u_y displacements exist)

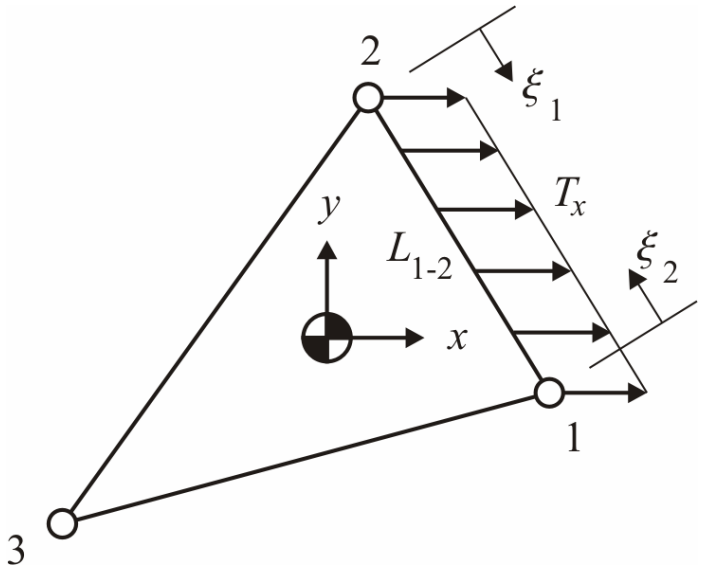
Plane Strain Conditions



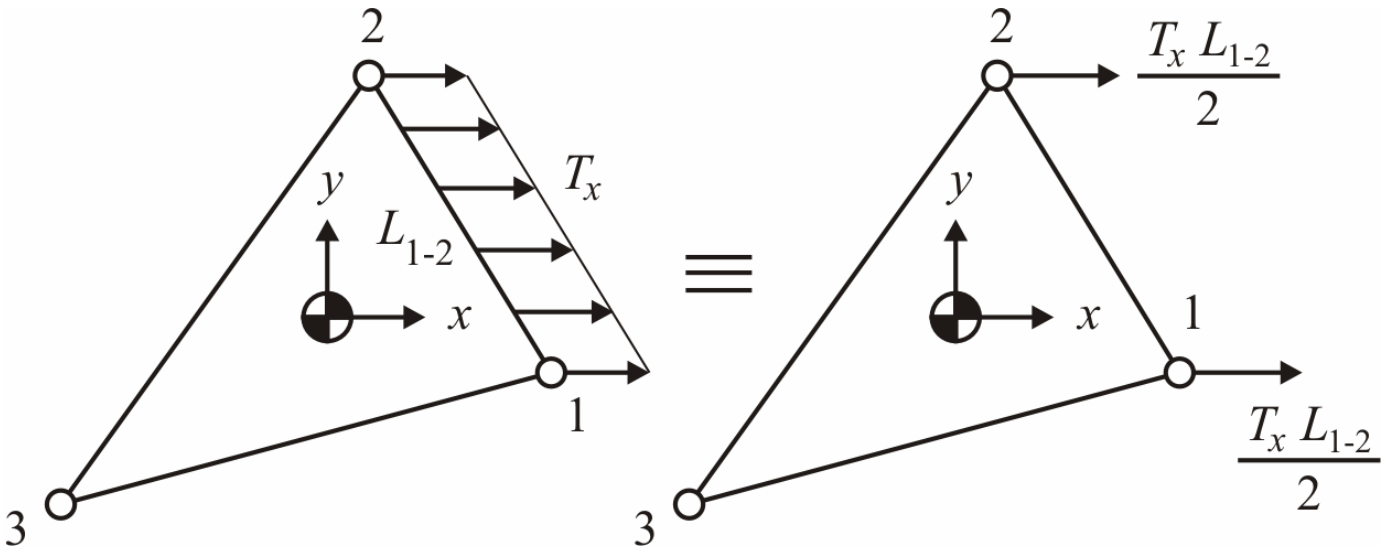
Long bodies with in-plane loading are suitable for plane strain idealization (u_z vanishes as well as any variation along z direction)

Typical Linear Triangular Element

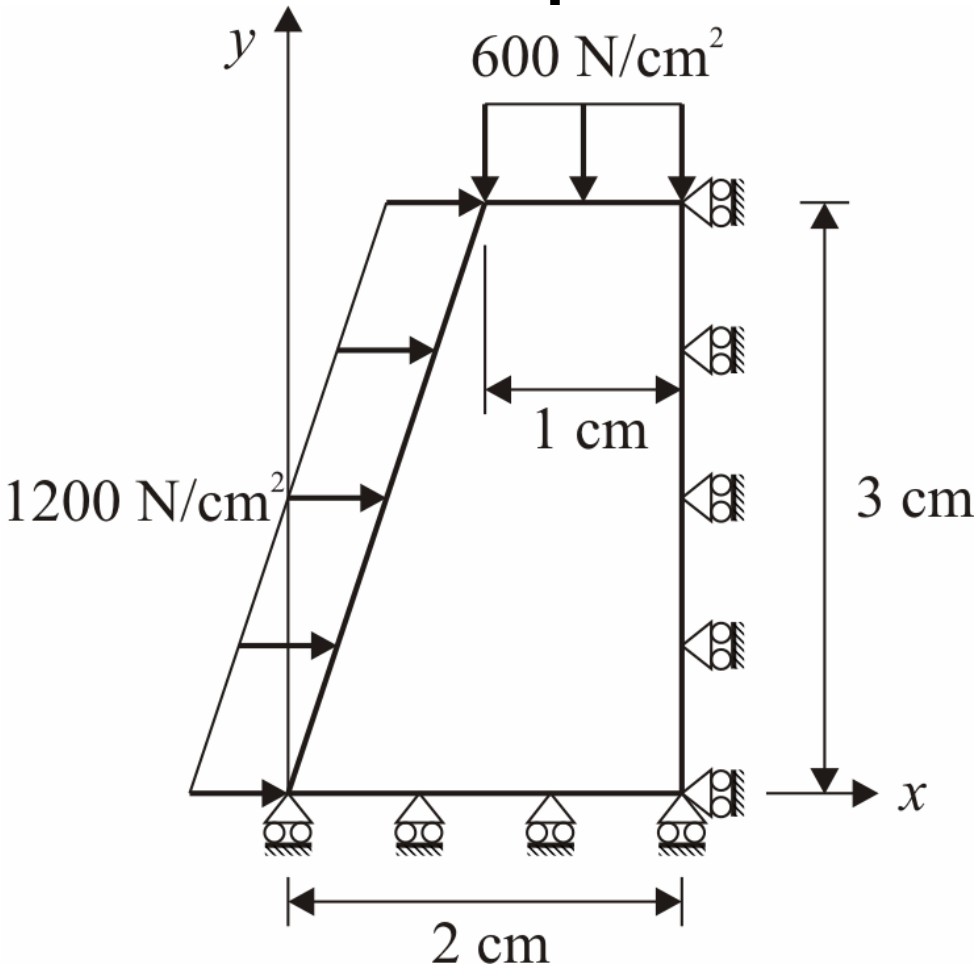




Equivalent Nodal Forces



Example#1



Example#1

- Using linear triangular elements determine the nodal displacements and the element stresses in a thin plate subjected to displacements constraints and surface tractions as shown in the figure. The plate is exposed to a temperature change of 10° C from the reference temperature. The plate thickness is 0.5cm. N=0.25; E=15e6 N/cm^2;the coefficient of thermal expansion is α=6e-6 /^{o}C.

Batch Mode Approach (*.Log)

```
/PREP7              ! ENTER PREPROCESSOR
ET,1,PLANE42        ! USE PLANE42 ELEMENT
KEYOPT,1,3,3        ! USE PLANE STRESS WITH          THICKNESS OPTION
R,1,5E-3,           ! SPECIFY THICKNESS THROUGH REAL CONSTANTS
MP,EX,1,150E9       ! SPECIFY ELASTIC MODULUS
MP,PRXY,1,0.25      ! SPECIFY POISSON'S RATIO
MP,ALPX,1,6E-6      ! SPECIFY COEFFICIENT OF THERMAL EXPANSION
N,1,0,0             ! CREATE NODES
N,2,0.02,0          !
N,3,0.02,0.03       !
N,4,0.01,0.03       !
E,1,2,4             ! CREATE ELEMENTS
E,2,3,4             !
FINISH              ! EXIT PREPROCESSOR
```

Batch Mode Approach (*.Log) (continued)

```
/SOLU                ! ENTER SOLUTION PROCESSOR
TUNIF,10             ! SPECIFY UNIFORM TEMPERATURE INCREASE
D,1,UY,0             ! SUPPRESS Y-DISPLACEMENT AT NODE 1
D,2,UY,0             ! SUPPRESS Y-DISPLACEMENT AT NODE 2
D,2,UX,0             ! SUPPRESS X-DISPLACEMENT AT NODE 2
D,3,UX,0             ! SUPPRESS X-DISPLACEMENT AT NODE 3
F,1,FX,3E3*SQRT(0.1)    ! APPLY FORCE IN X-DIR AT NODE 1
F,4,FX,3E3*SQRT(0.1)    ! APPLY FORCE IN X-DIR AT NODE 4
F,4,FY,-150          ! APPLY FORCE IN Y-DIR AT NODE 4
F,3,FY,-150          ! APPLY FORCE IN Y-DIR AT NODE 3
SOLVE                ! OBTAIN SOLUTION
FINISH               ! EXIT SOLUTION PROCESSOR
/POST1               ! ENTER GENERAL POSTPROCESSOR
PLDISP,2             ! PLOT DISPLACED SHAPE
PRNSOL,DOF       ! LIST DISPLACEMENT RESULTS
```

Solution using GUI

- Specify the element type (**ET**):

  Main Menu>Preprocessor>Element Type>Add/Edit/Delete

  Add from **Structural Solid list** the **Quad 4node 42** element

  2D idealization as plane stress with thickness: click **Options,** select in the dialog box for **Element behavior K3: Plane stress with/thk**
- Specify real constants (**R**)

  Specify **5e-3** for the **Thickness THK**
- Specify material properties **(MP)** using **Structural, Linear, Elastic, Isotropic** models; EX=150E9;PRXY=0.25; for T**hermal Expansion, Secant Coefficient: Isotropic, APLX=6e-6; Material>Exit** close the dialog box.
- Nodes:

  Main Menu>Preprocessor>Modeling>Create>Nodes>IN Active CS
- Elements

  Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Thru Nodes

Solution using GUI

- Review model: review any geometrical and material properties
- elements: **Utility Menu>PlotCtrls>Numbering**
- **EPLOT** : **Utility Menu>Elements>Plot**
- **NPLOT** : **Utility Menu>Plot>Nodes**
- **Solution:**

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

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

***Pick Menu* appears;Pick nodes 1 and 4 along the slanted boundary ; click OK and enter 3e3*sqrt(0.1) for VALUE;click Apply; When the Pick Menu reappears pick nodes 4 and 3 along the top horizontal boundary and click OK; pull down the menu for direction of Force/Moment and select FY;enter -150 for VALUE;click OK**

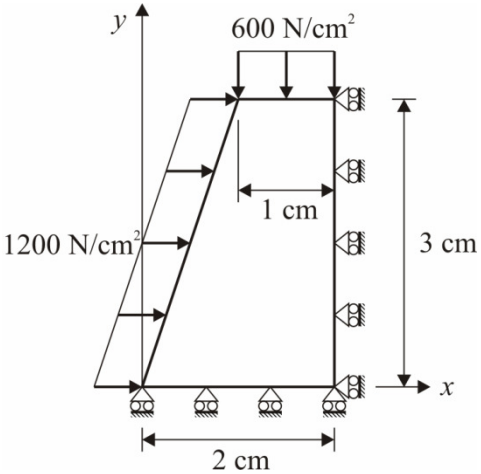
Solution using GUI

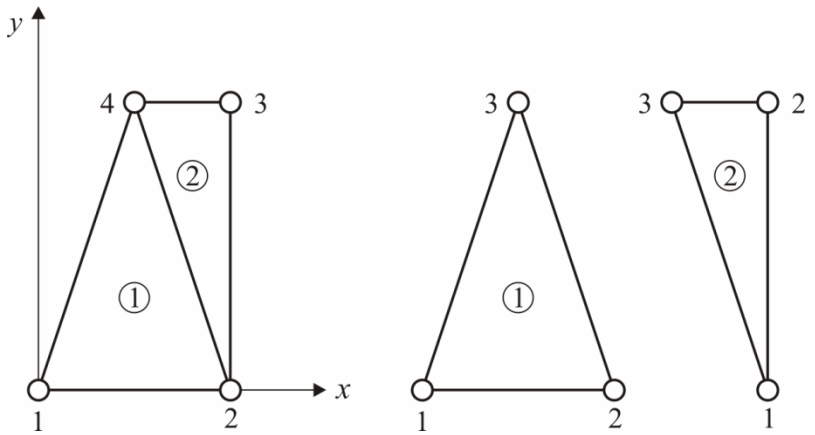
- Apply thermal load (**TUNIF**)

  Main Menu>Solution>Define Loads>Apply>Structural>Temperature>Uniform Temperature

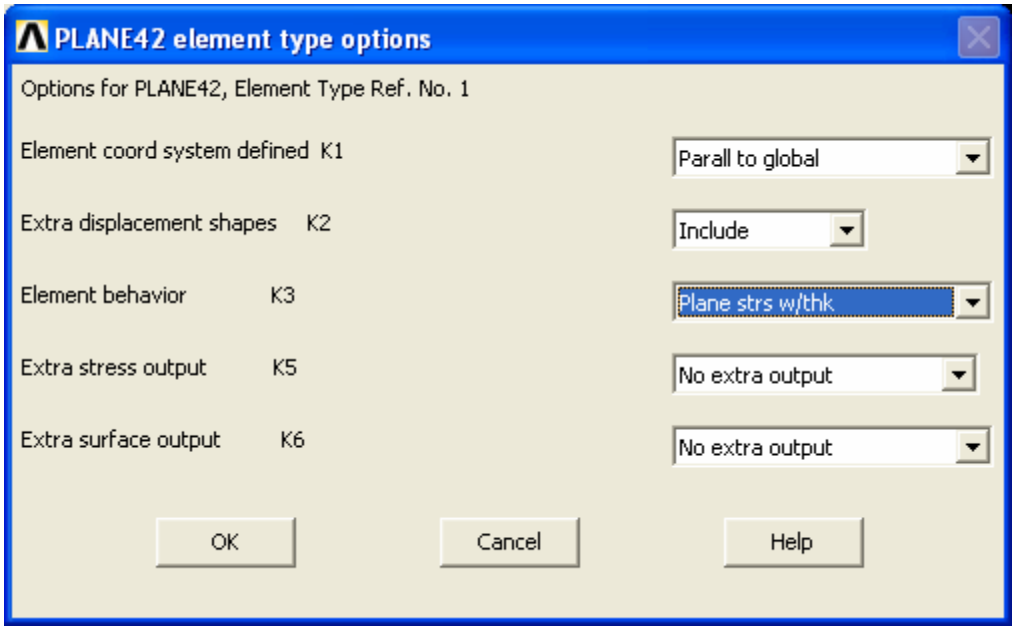
  Enter **10** for **Uniform temperature** in the **uniform temperature** dialog box;OK
- Solution: **Main Menu>Solution>Solve>Current LS**
- **Postprocessing (PLDISP and PRNSOL):**
- **Main Menu>General Postprocessing>Plot Results>Deformed Shape**
- **Main Menu>General Postprocessing>>List results>Nodal Solution**

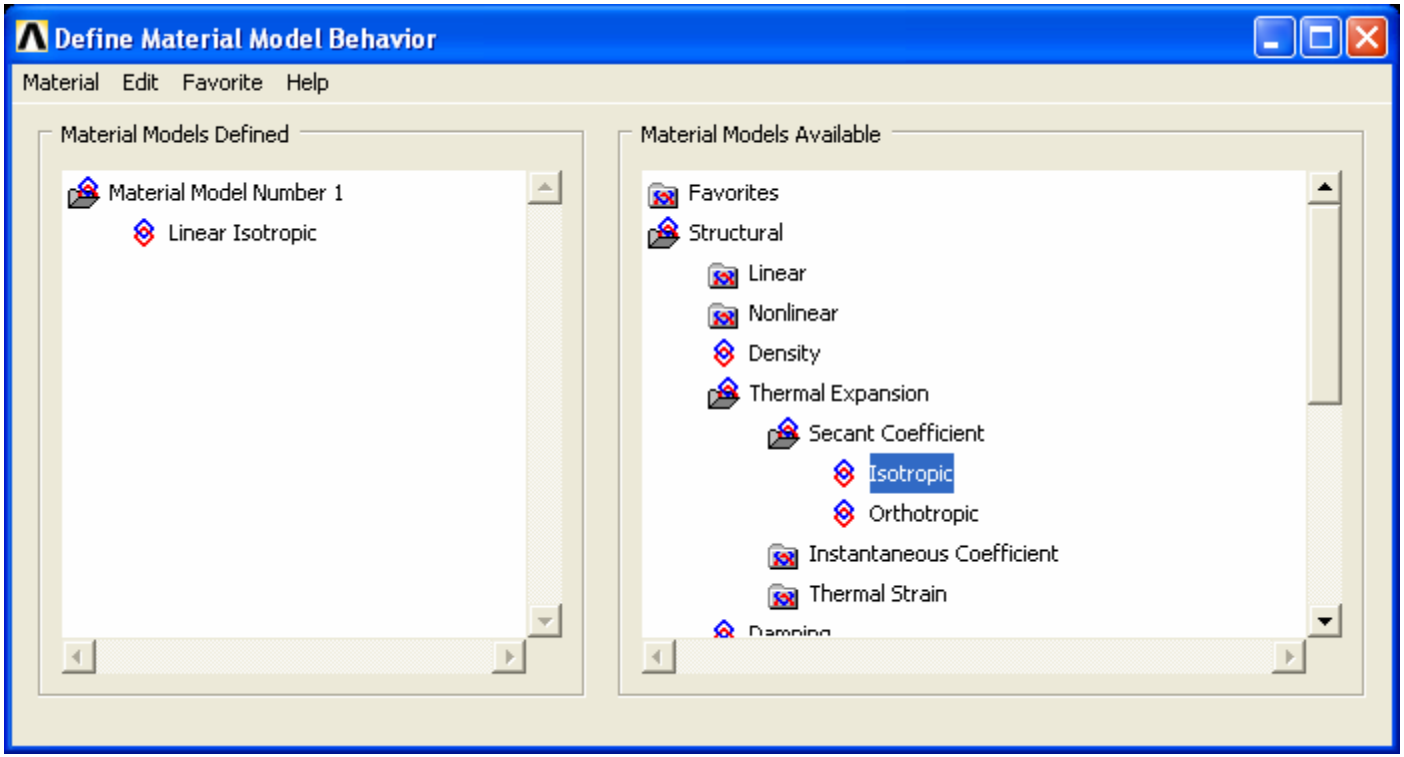
Model



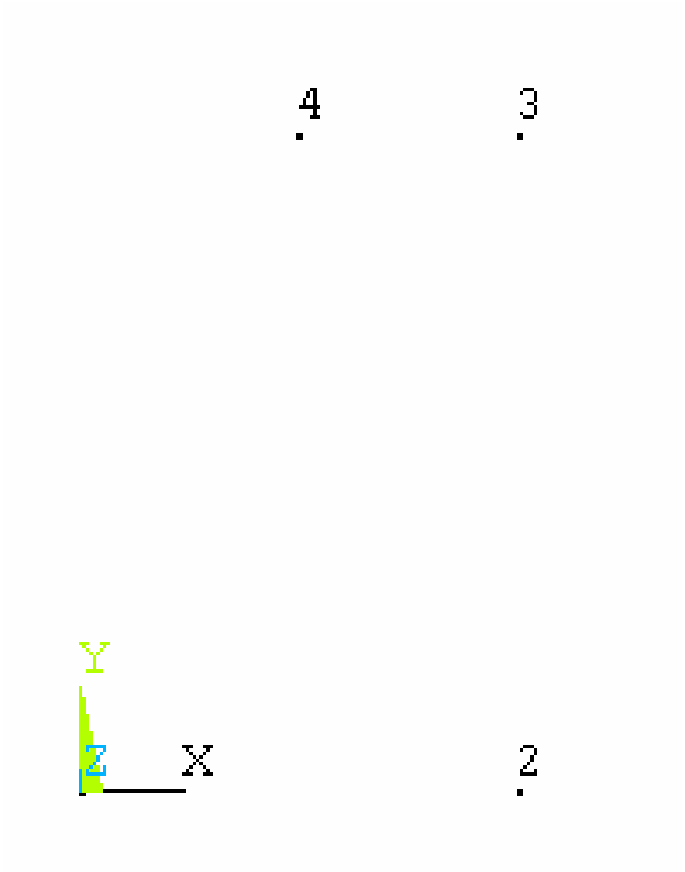


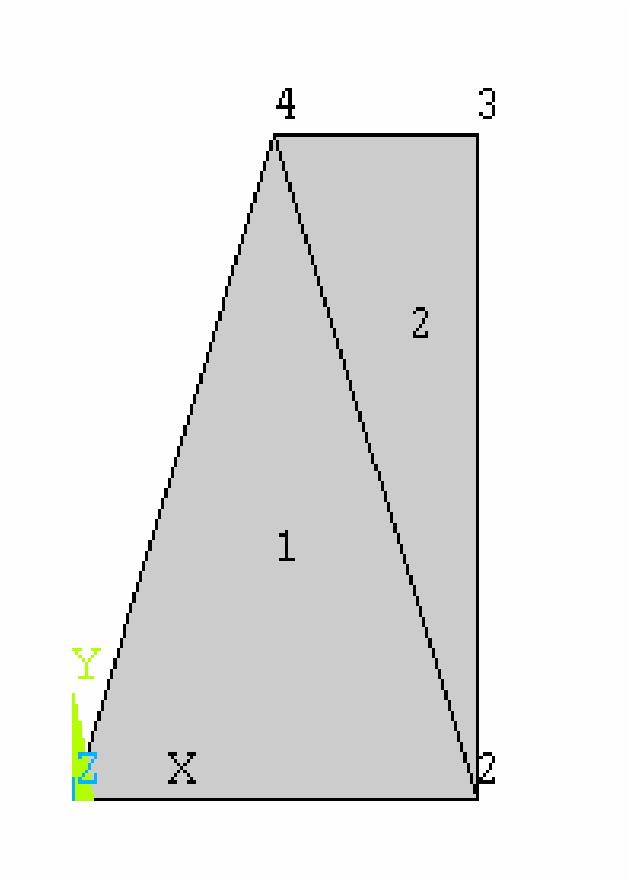
Model


PLANE42 element type options
Options for PLANE42, Element Type Ref. No. 1
Element coord system defined K1
Parall to global
Extra displacement shapes K2
Include
Element behavior K3
Plane strs w/thk
Extra stress output K5
No extra output
Extra surface output K6
No extra output
OK
Cancel
Help

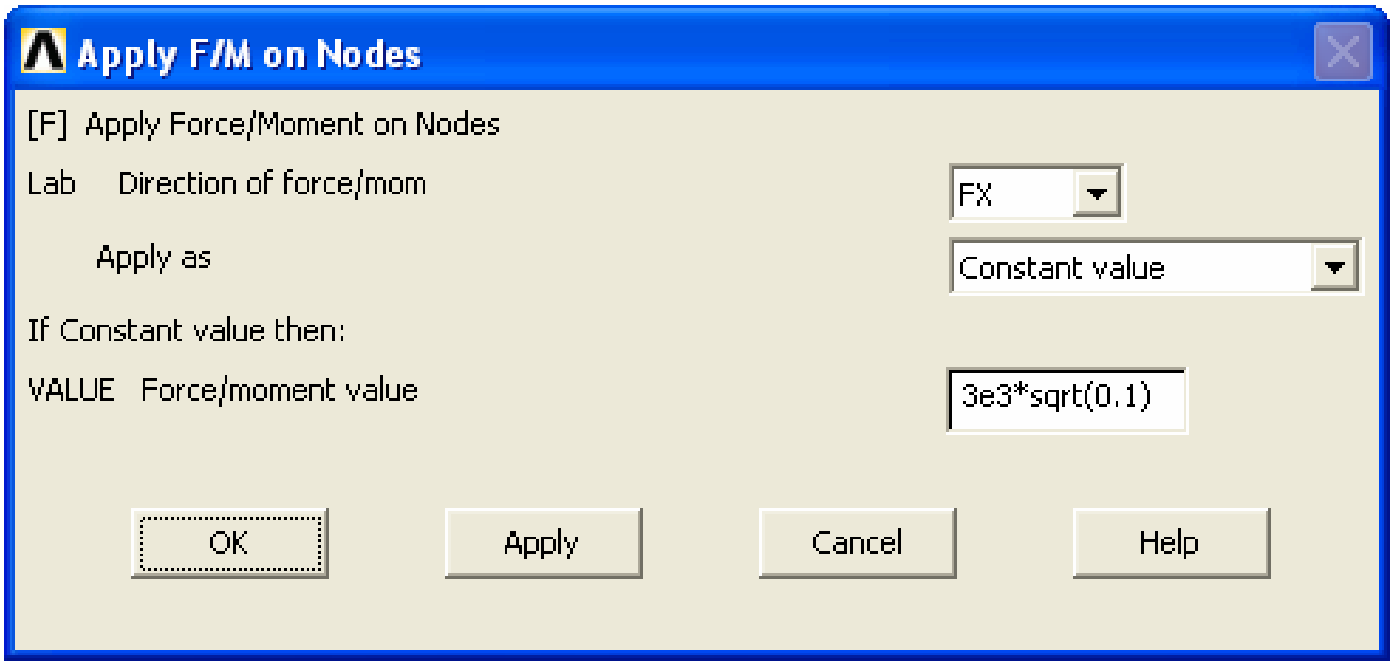

Define Material Model Behavior
Material
Edit
Favorite
Help
Material Models Defined
Material Model Number 1
Linear Isotropic
Material Models Available
Favorites
Structural
Linear
Nonlinear
Density
Thermal Expansion
Secant Coefficient
Isotropic
Orthotropic
Instantaneous Coefficient
Thermal Strain

Model

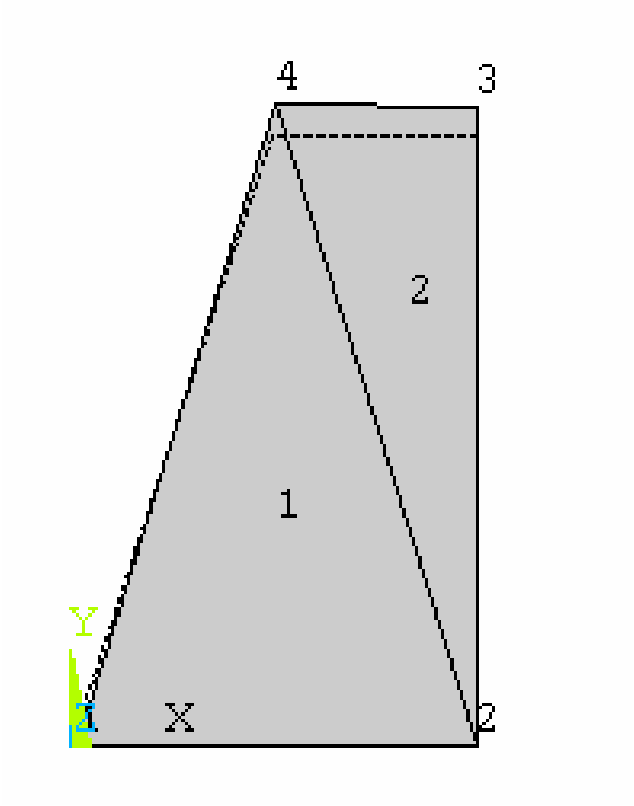




Model


Apply F/M on Nodes
[F] Apply Force/Moment on Nodes
Lab Direction of force/mom
FX
Apply as
Constant value
If Constant value then:
VALUE Force/moment value
3e3*sqrt(0.1)
OK
Apply
Cancel
Help

Model

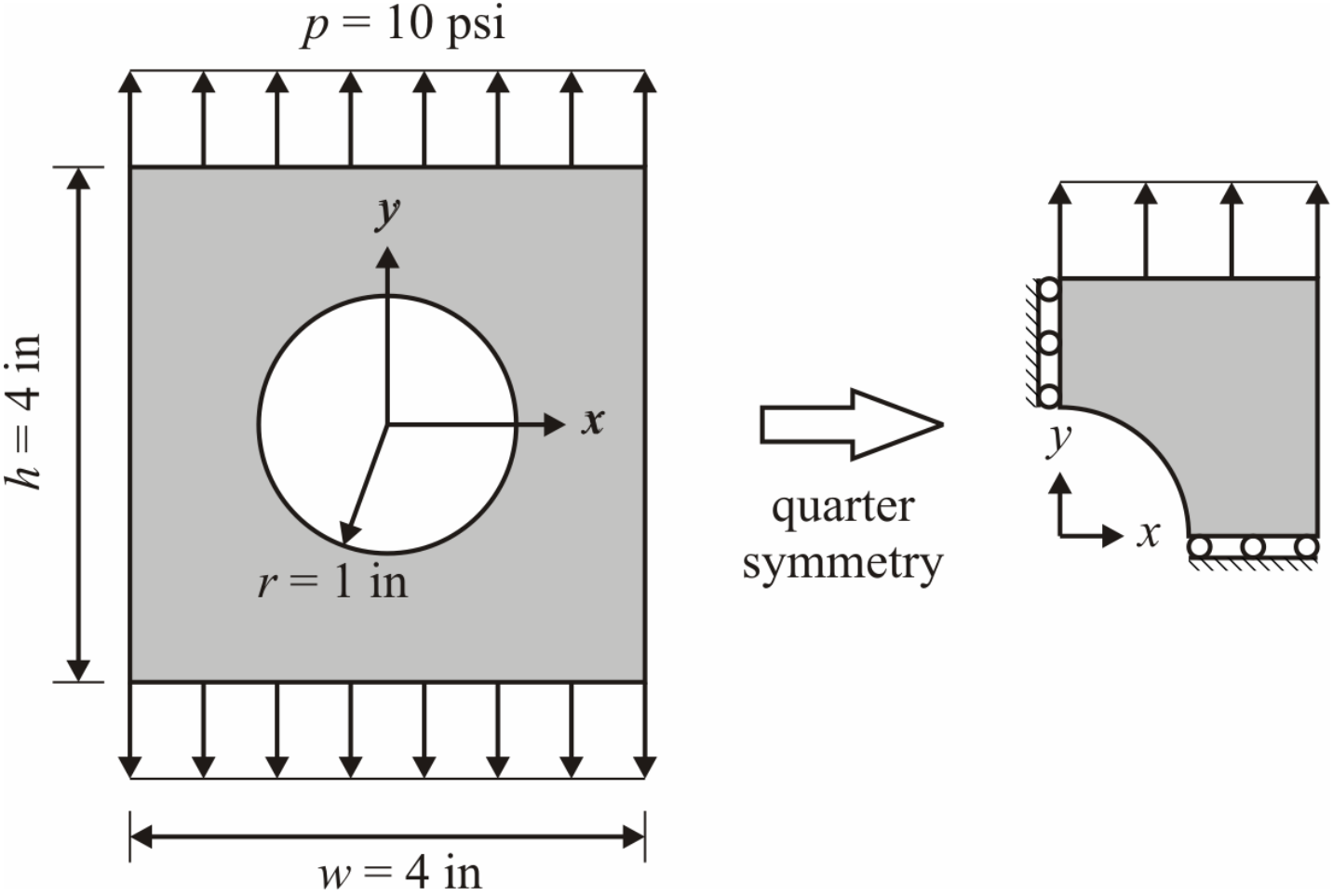


PRNSOL Command

File

```
 PRINT DOF  NODAL SOLUTION PER NODE

  ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000       LOAD CASE=  0

 THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

    NODE       UX           UY
       1  0.35784E-06  0.0000
       2   0.0000       0.0000
       3   0.0000      0.15700E-05
       4  0.17198E-06 0.16637E-05

 MAXIMUM ABSOLUTE VALUES
 NODE          1            4
 VALUE   0.35784E-06 0.16637E-05
```

Example#2



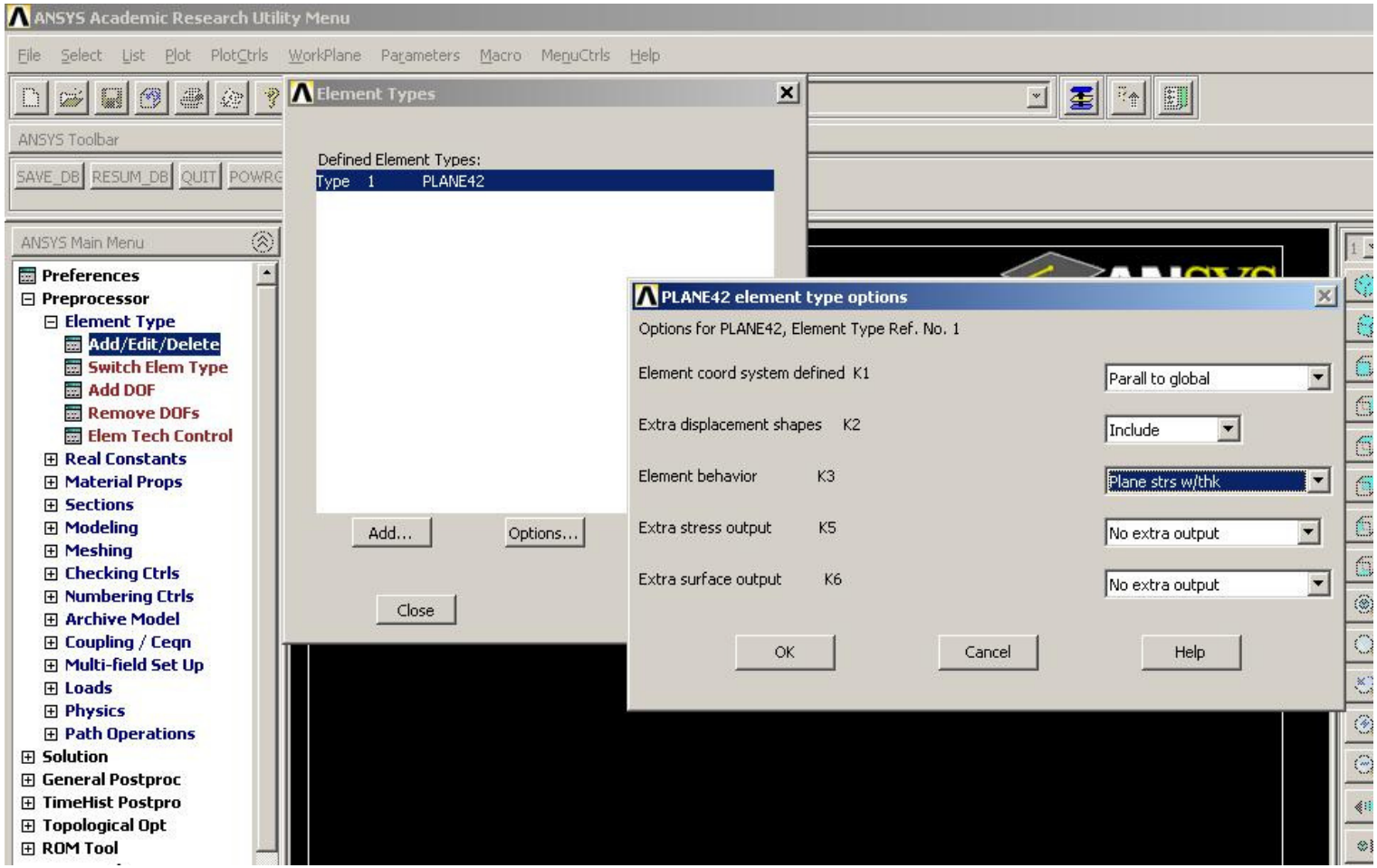
A thin, square structure with a centric circular hole subjected to a tensile loading in the y-direction is considered, as above.

Example#2

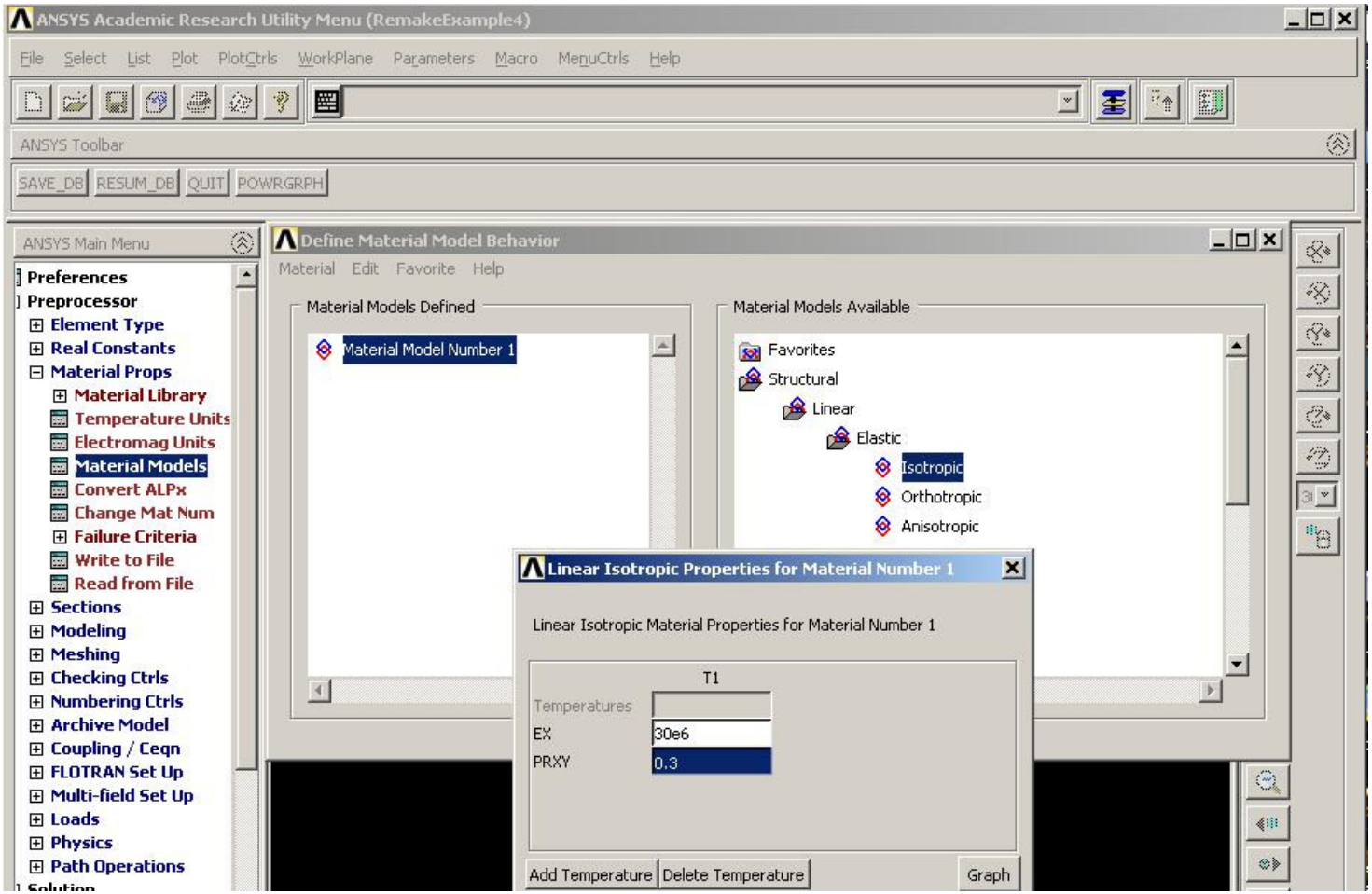
- The length of the square and the radius of the circular hole are w=4in, and r=1in, respectively. The geometry and the material possess quarter-symmetry, therefore only one-fourth of the domain is modeled. Because the thickness is significantly smaller than the in-plane dimensions, a plane stress assumption is used.v=0.3,E=30.0e+06 psi. The distributed load is q=10lbs/in applied in the form of a pressure loading with q=-10.

- *Solved using the GUI step by step. Input (log) file provided for comparison between APDL and GUI approaches.*

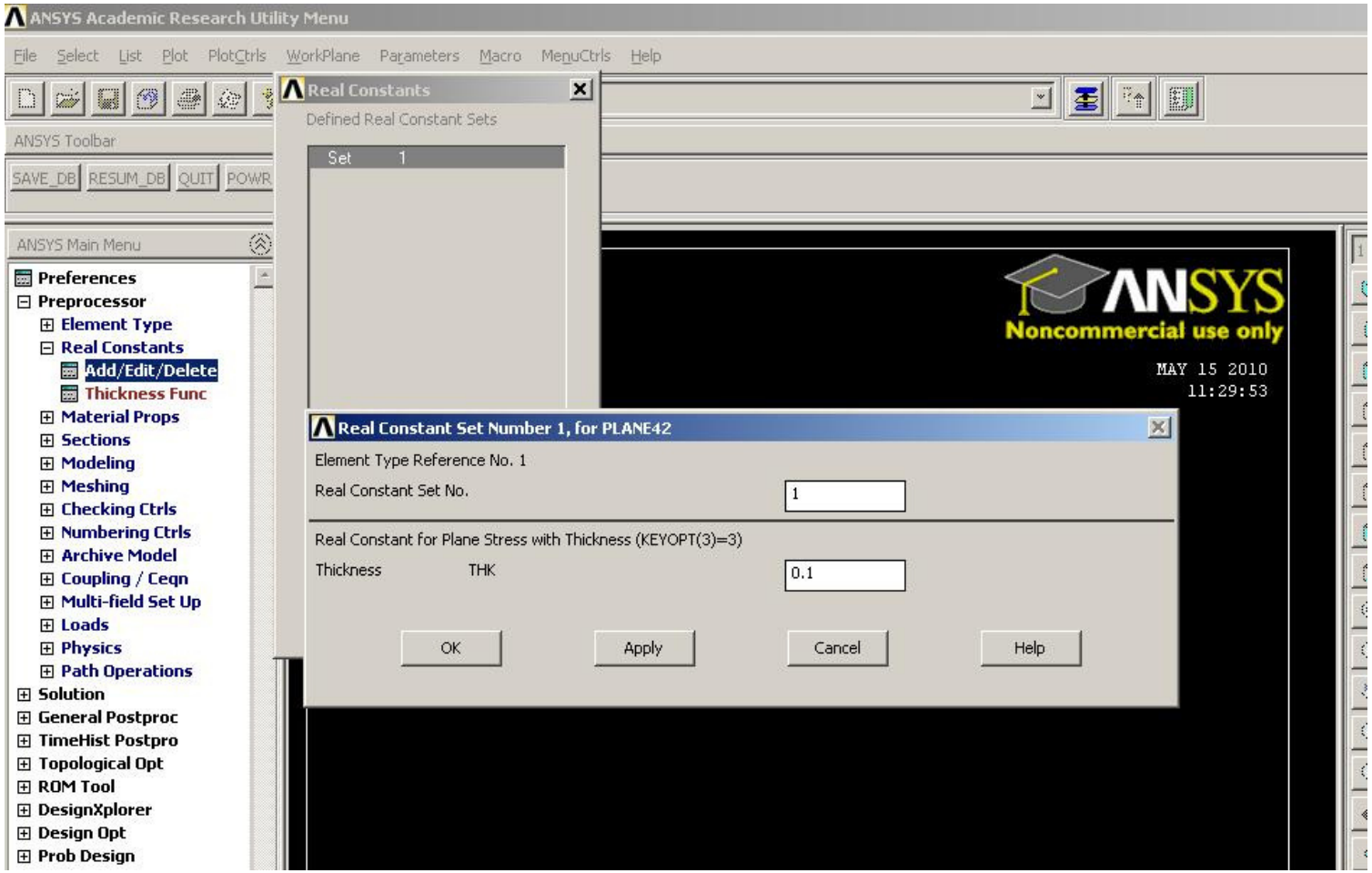
Select Element Type: PLANE 42



Choose Material Properties

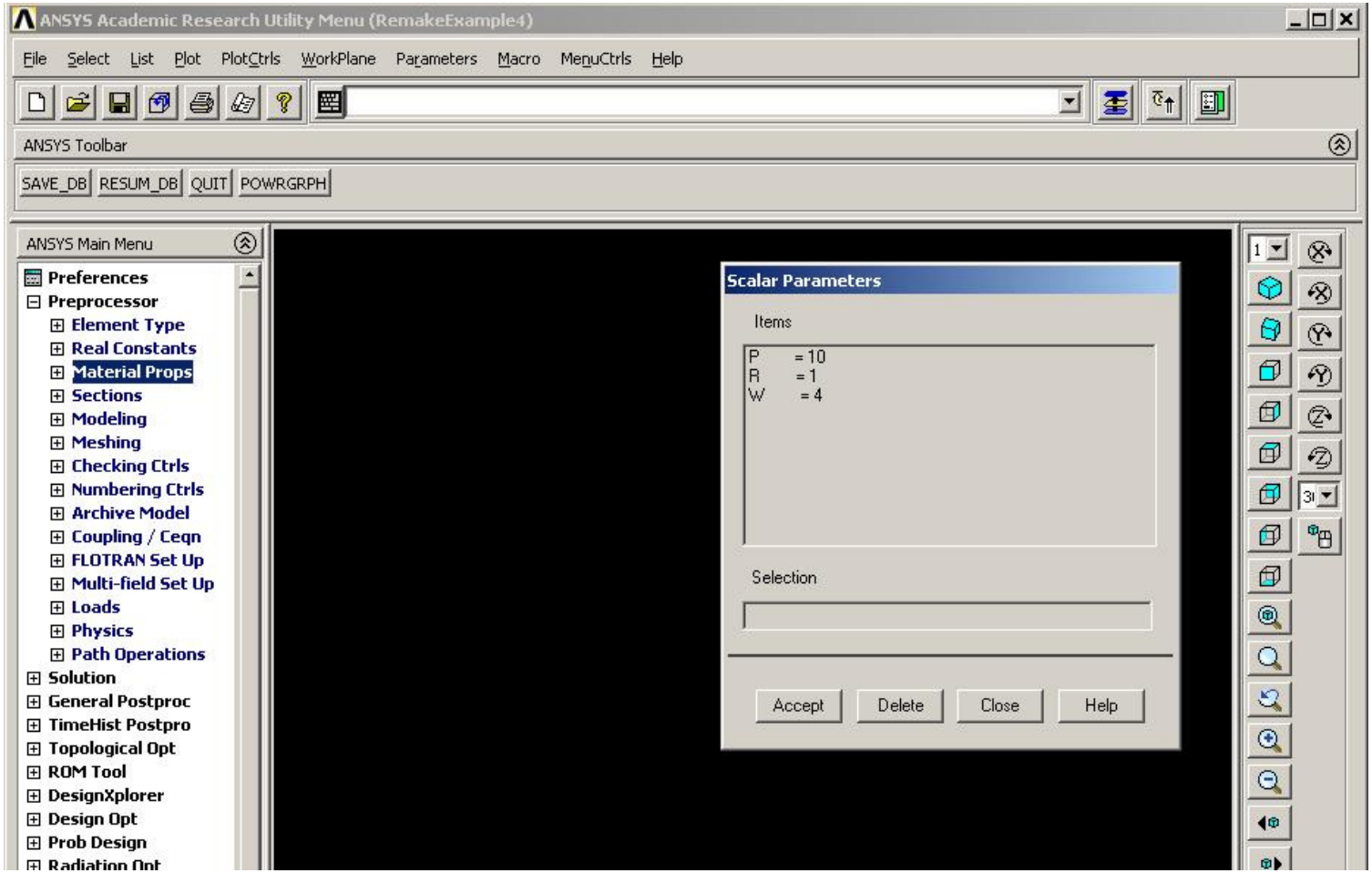


Choose Real Constants

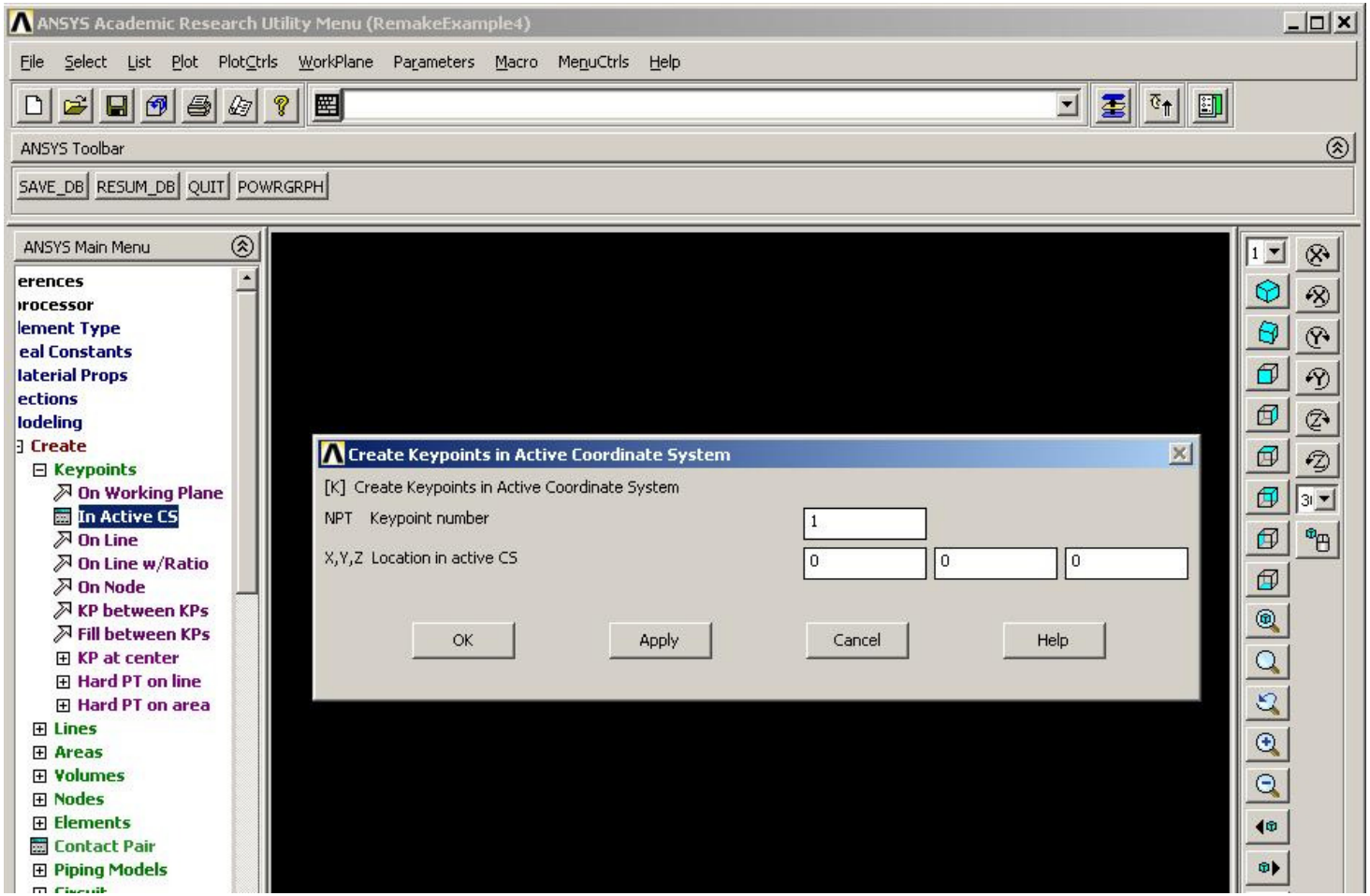


Scalar Parameters

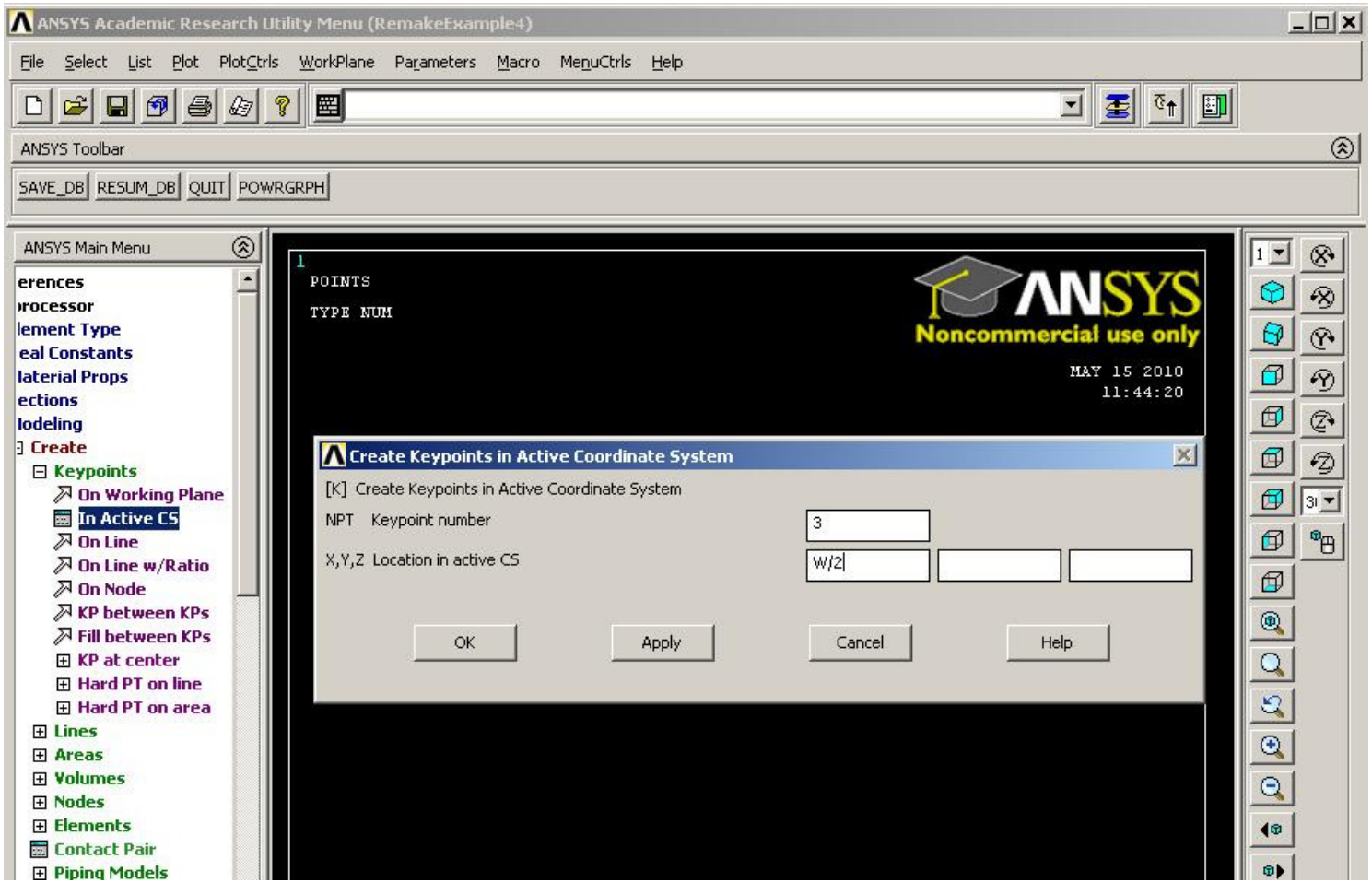
->parameters->



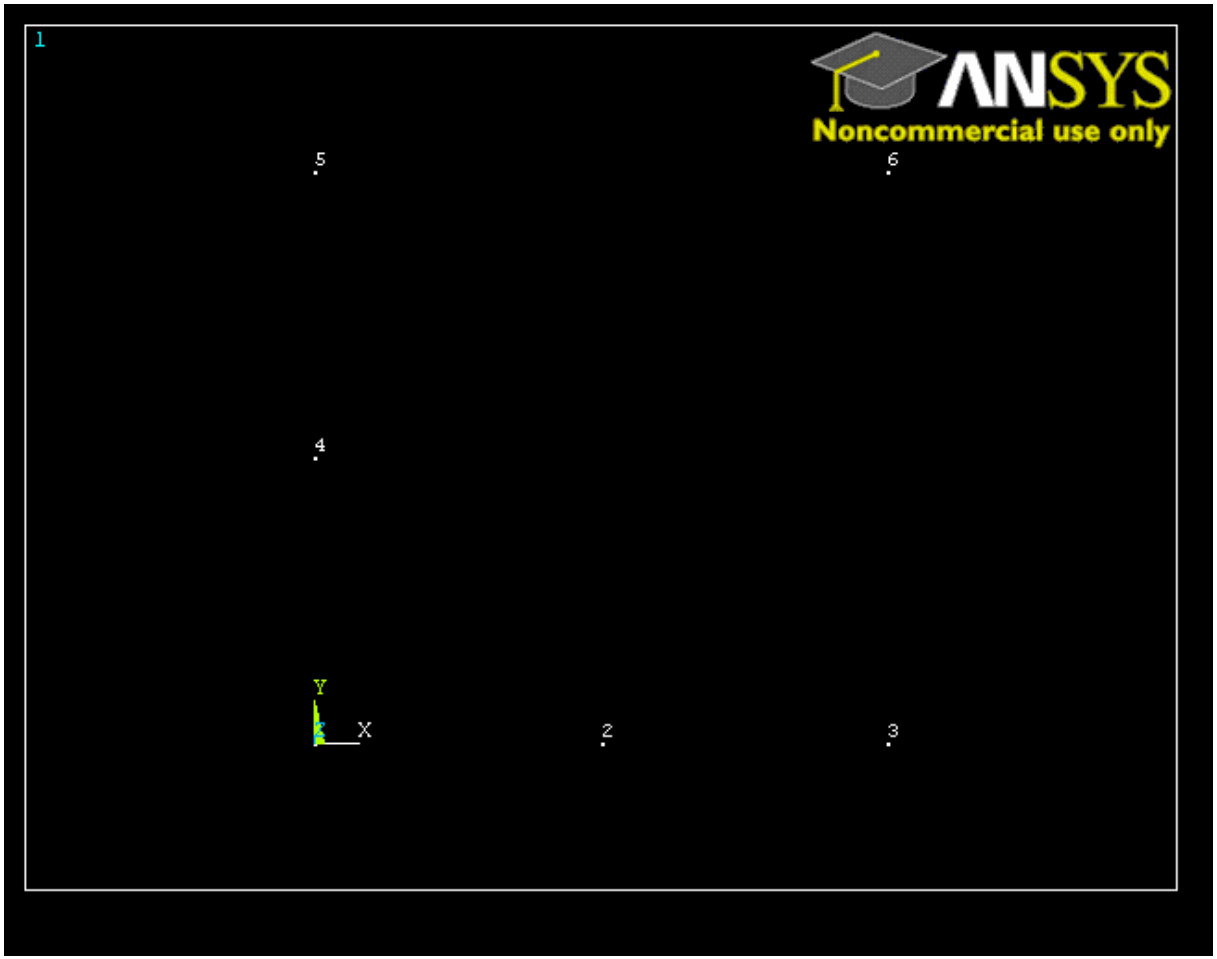
Create Key Points



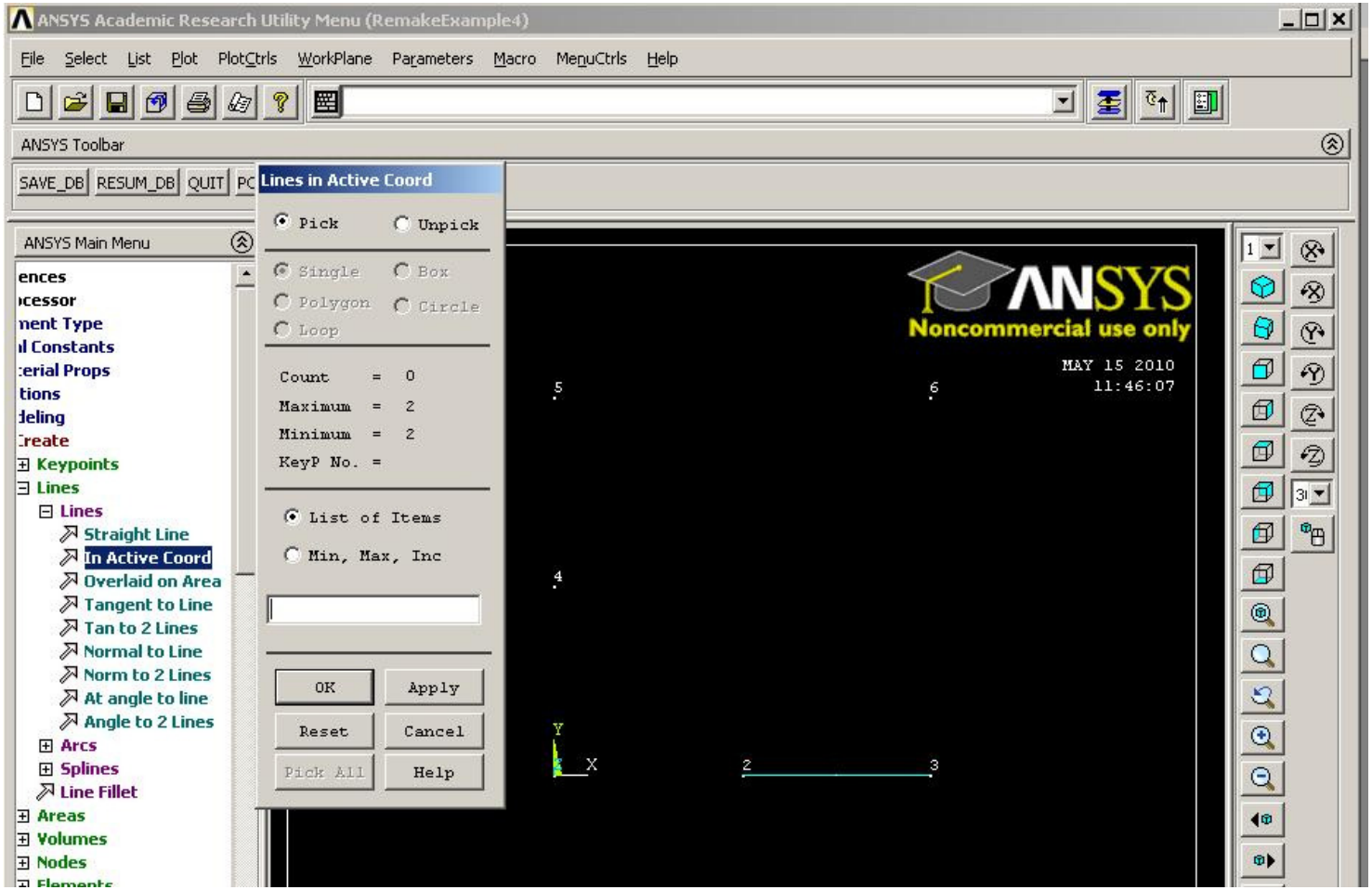
Key Points


ANSYS Academic Research Utility Menu (RemakeExample4)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
erences
rocessor
lement Type
eal Constants
laterial Props
ections
lodeling
Create
Keypoints
On Working Plane
In Active CS
On Line
On Line w/Ratio
On Node
KP between KPs
Fill between KPs
KP at center
Hard PT on line
Hard PT on area
Lines
Areas
Volumes
Nodes
Elements
Contact Pair
Piping Models
POINTS
TYPE NUM
ANSYS
Noncommercial use only
MAY 15 2010
11:44:20
Create Keypoints in Active Coordinate System
[K] Create Keypoints in Active Coordinate System
NPT Keypoint number
3
X,Y,Z Location in active CS
W/2
OK
Apply
Cancel
Help

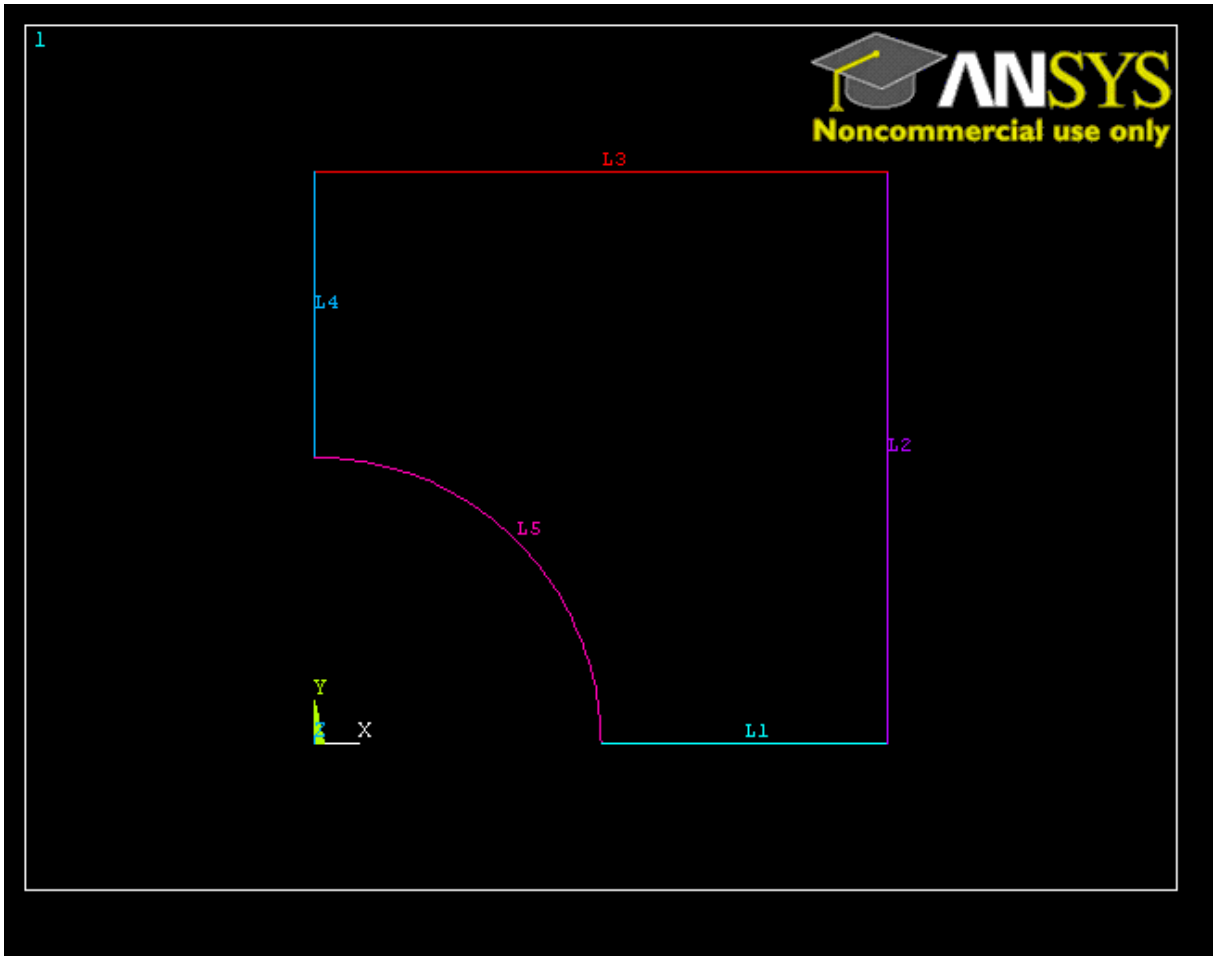
Key Points



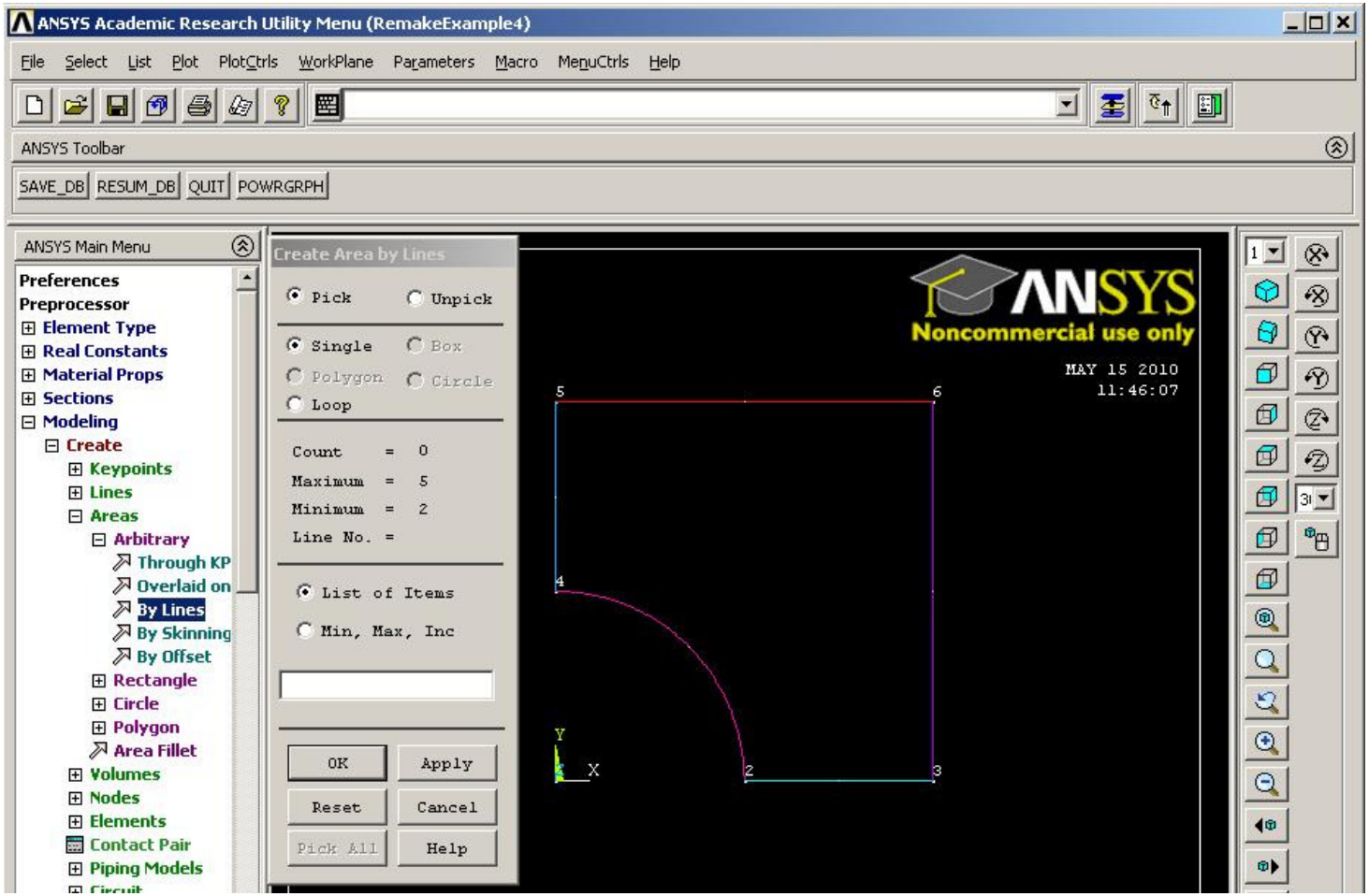
Create Lines



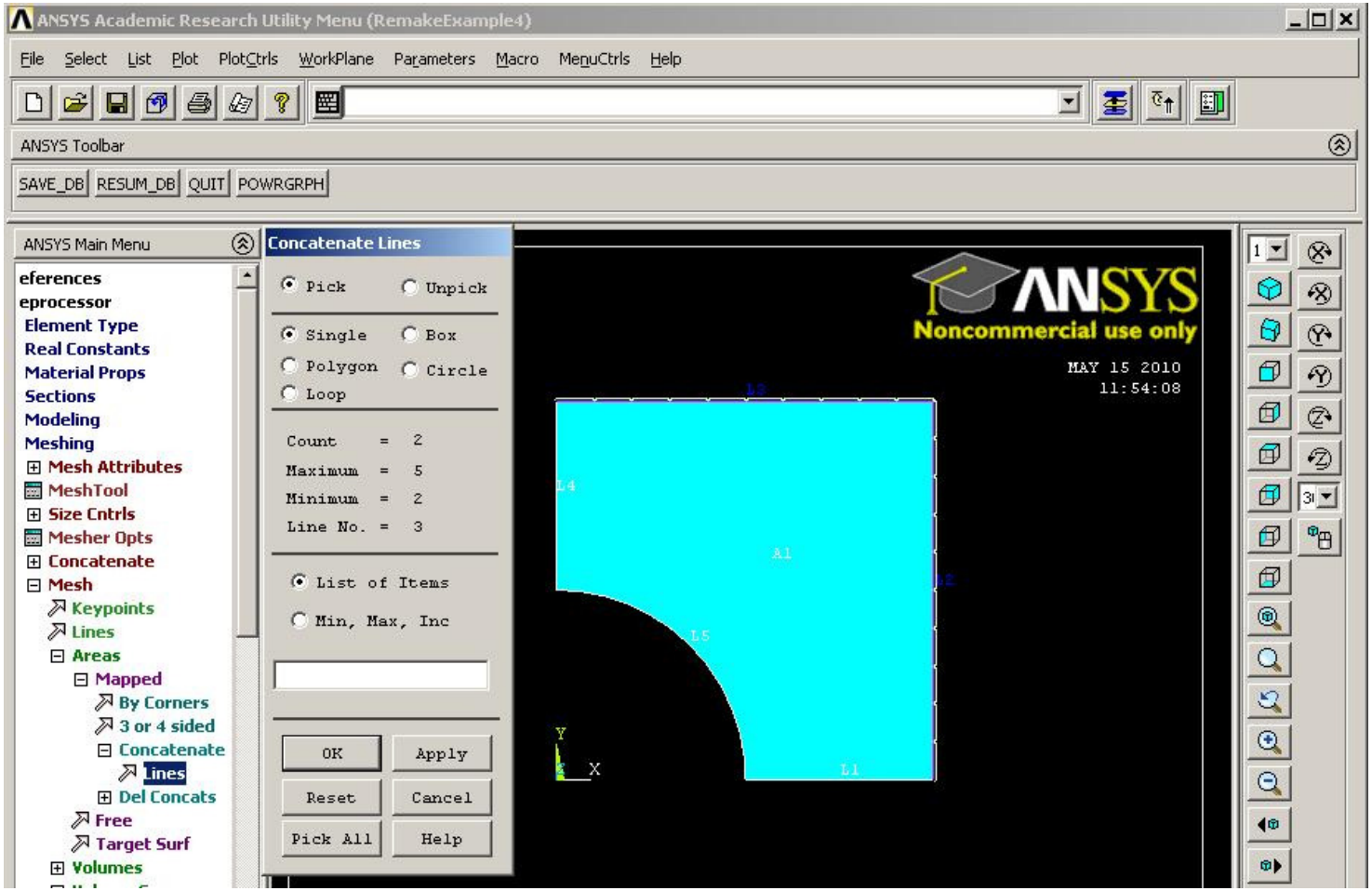
Create Lines



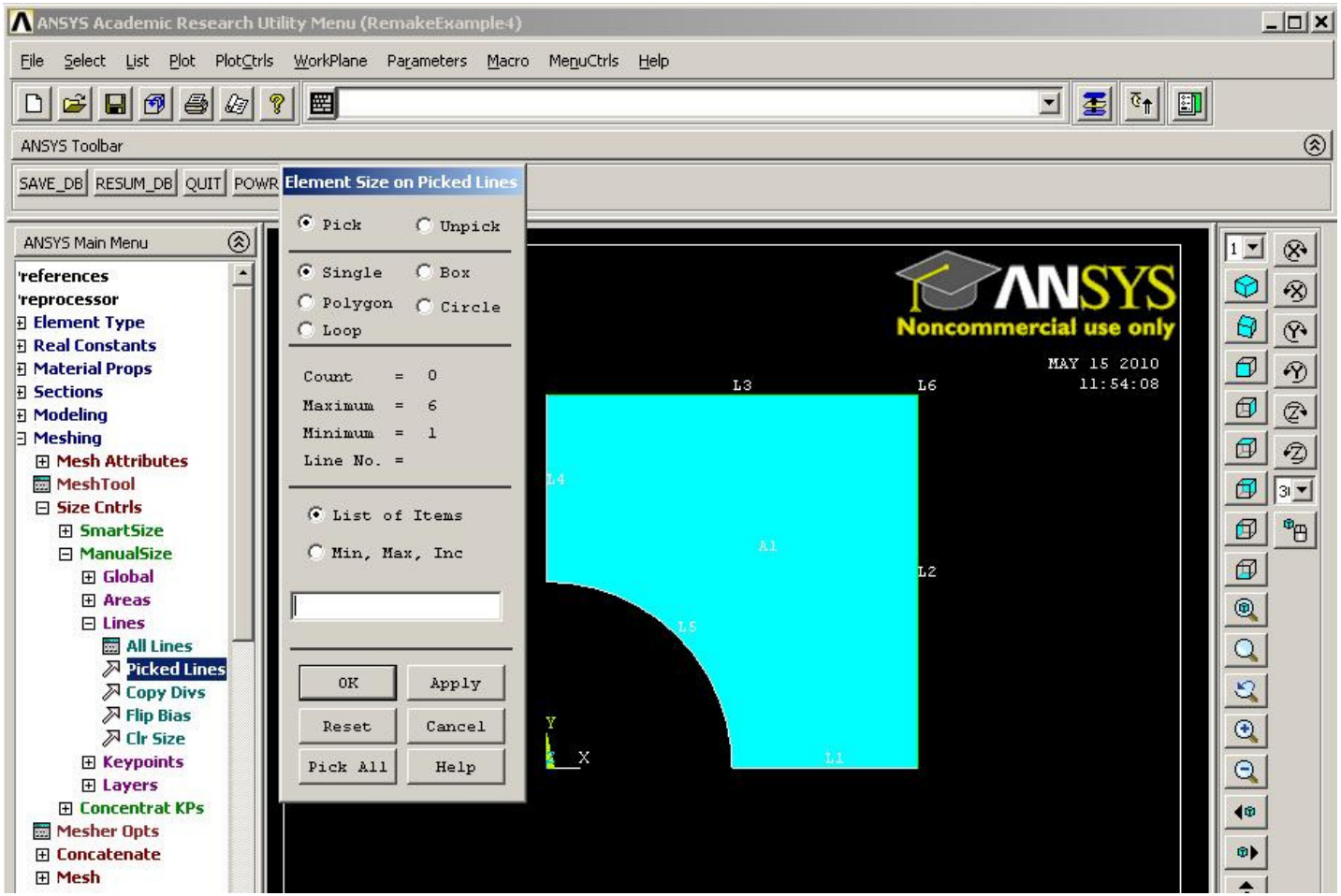
Create Area


ANSYS Academic Research Utility Menu (RemakeExample4)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
Lines
Areas
Arbitrary
Through KP
Overlaid on
By Lines
By Skinning
By Offset
Rectangle
Circle
Polygon
Area Fillet
Volumes
Nodes
Elements
Contact Pair
Piping Models
Create Area by Lines
Pick Unpick
Single Box
Polygon Circle
Loop
Count = 0
Maximum = 5
Minimum = 2
Line No. =
List of Items
Min, Max, Inc
OK Apply
Reset Cancel
Pick All Help
ANSYS
Noncommercial use only
MAY 15 2010
11:46:07
5
6
4
2
3
Y
X

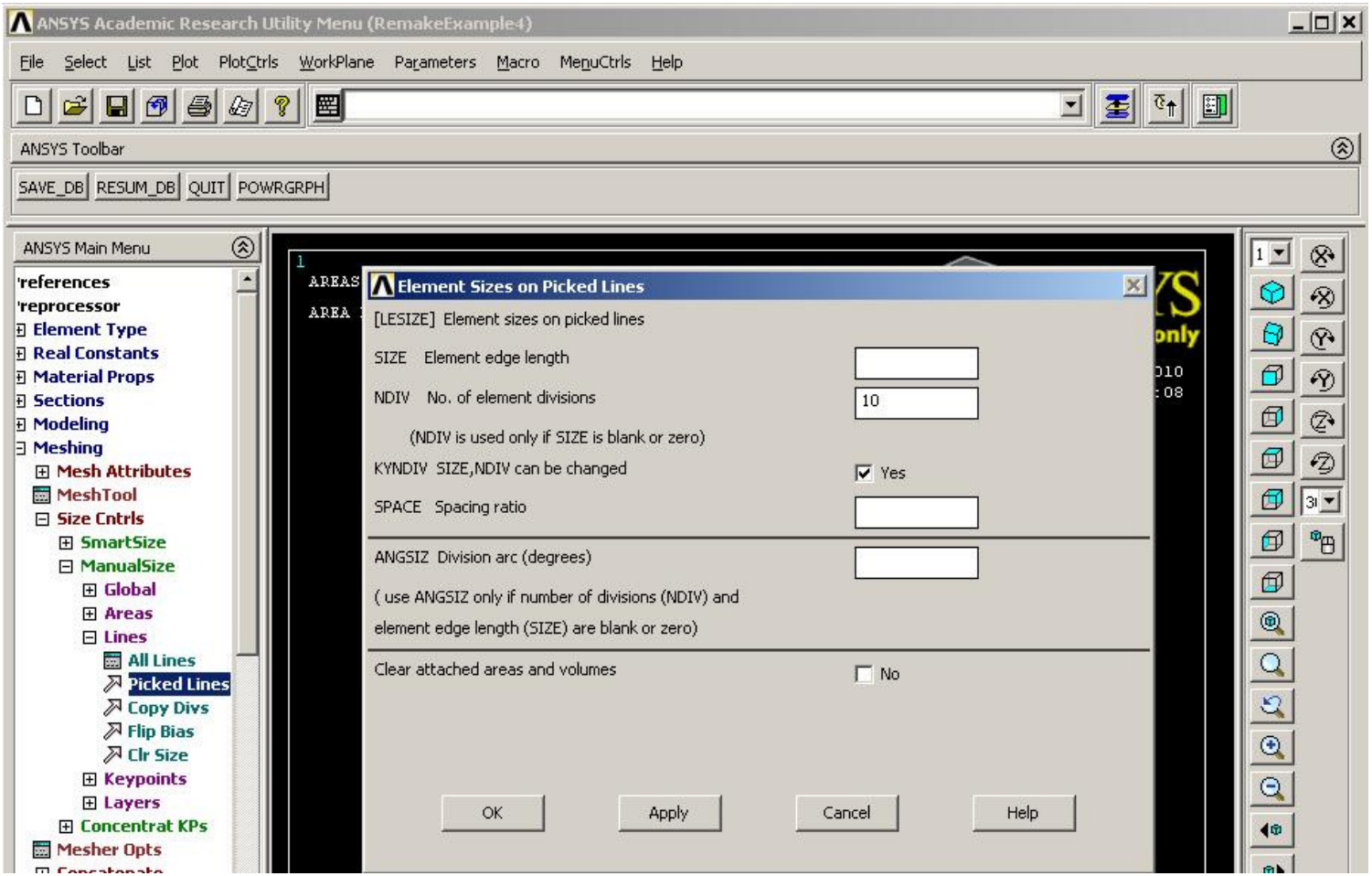
Concatenate Lines for Mapped Meshing



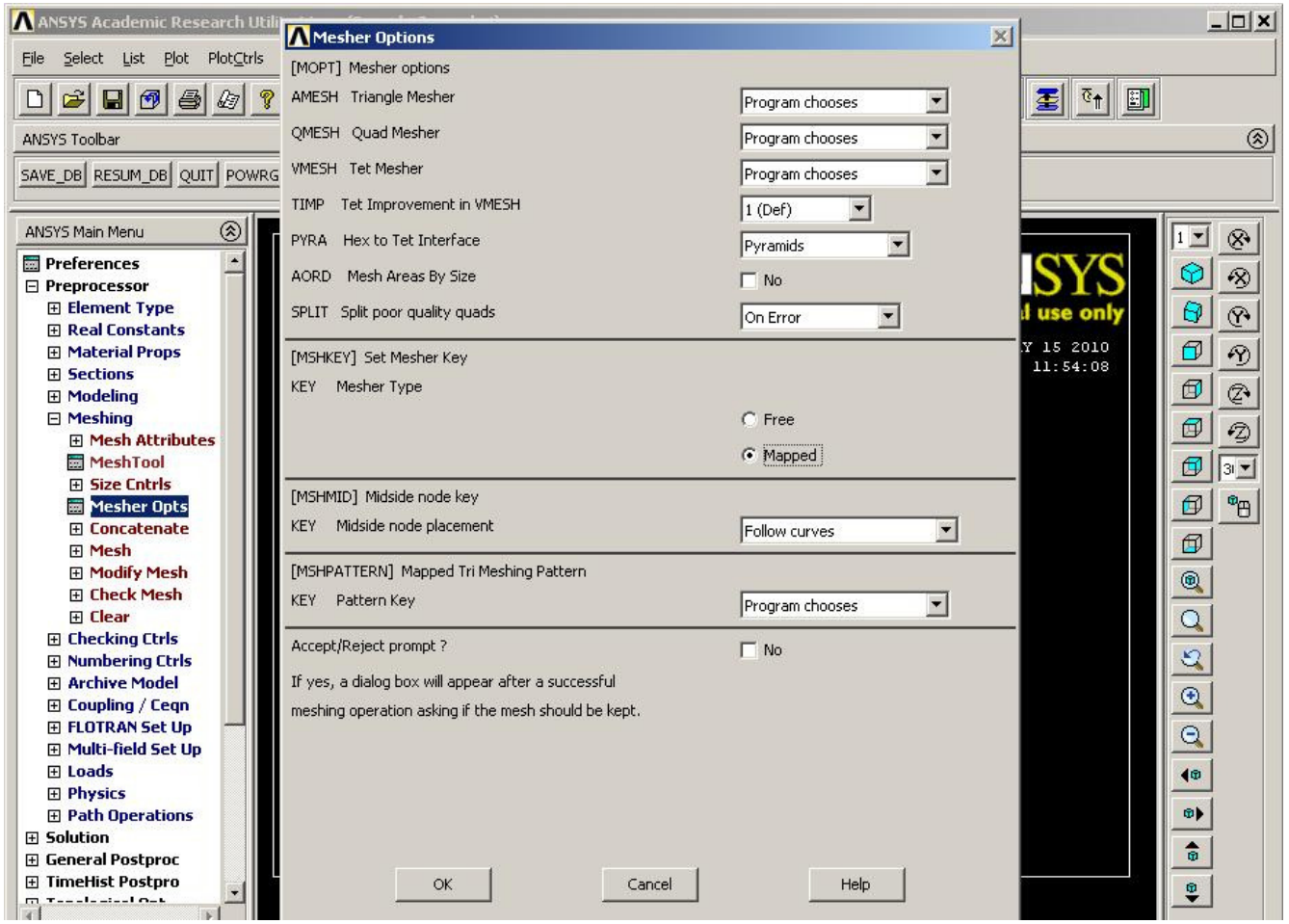
Meshing Control



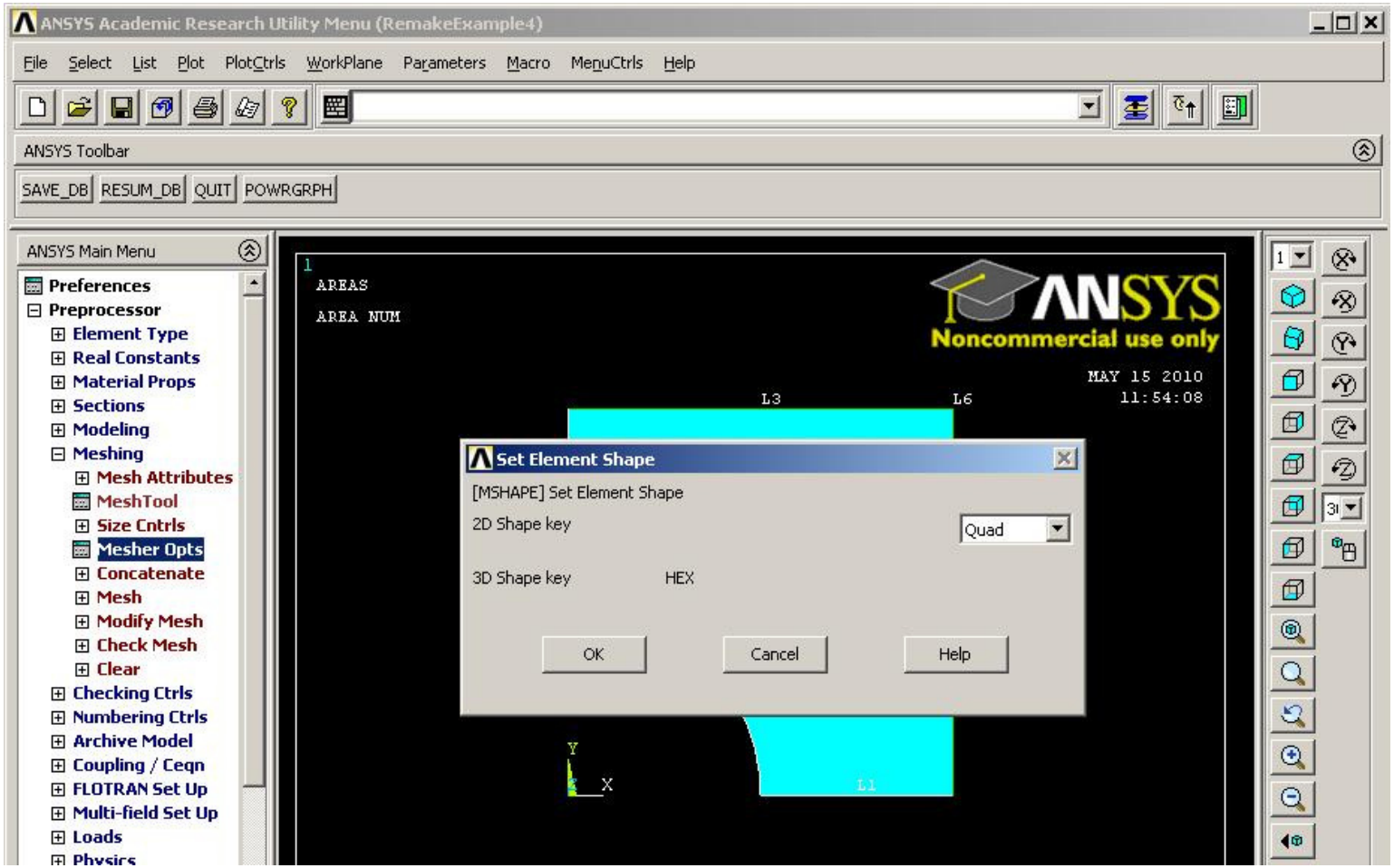
Meshing Control



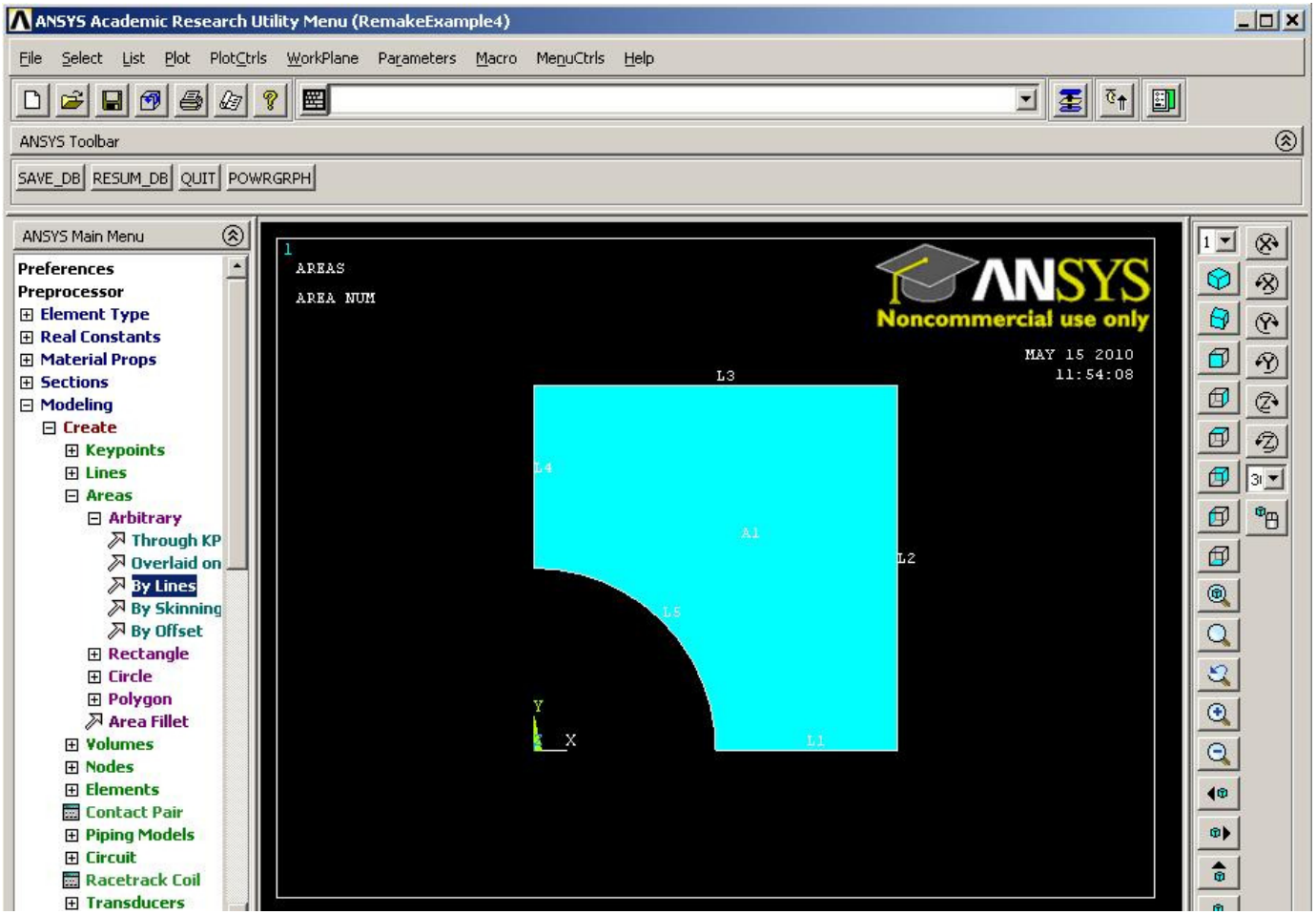
Meshing Type



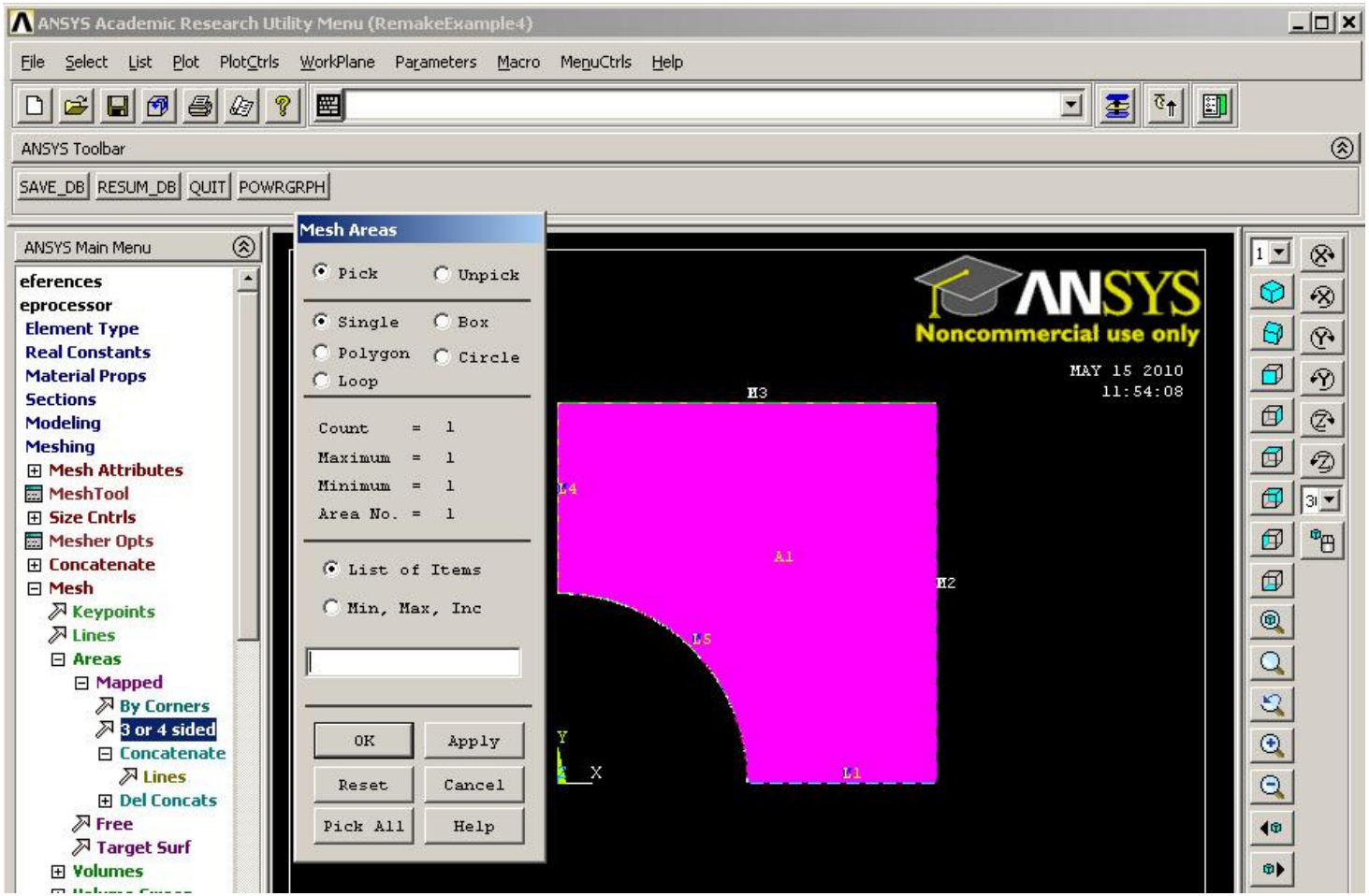
Meshing Type



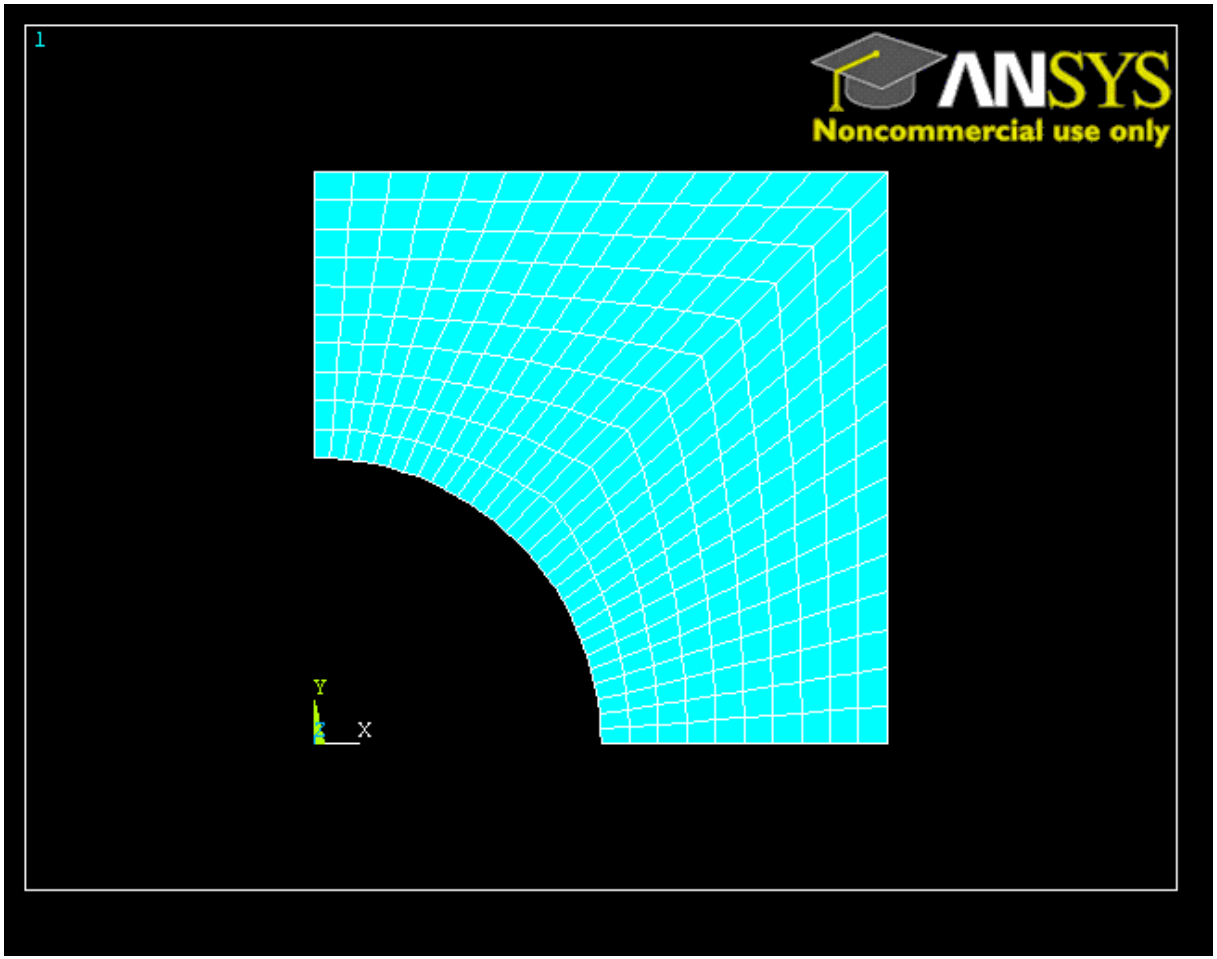
Create Area


ANSYS Academic Research Utility Menu (RemakeExample4)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Create
Keypoints
Lines
Areas
Arbitrary
Through KP
Overlaid on
By Lines
By Skinning
By Offset
Rectangle
Circle
Polygon
Area Fillet
Volumes
Nodes
Elements
Contact Pair
Piping Models
Circuit
Racetrack Coil
Transducers
AREAS
AREA NUM
ANSYS
Noncommercial use only
MAY 15 2010
11:54:08
L3
L4
A1
L2
L5
L1
Y
X

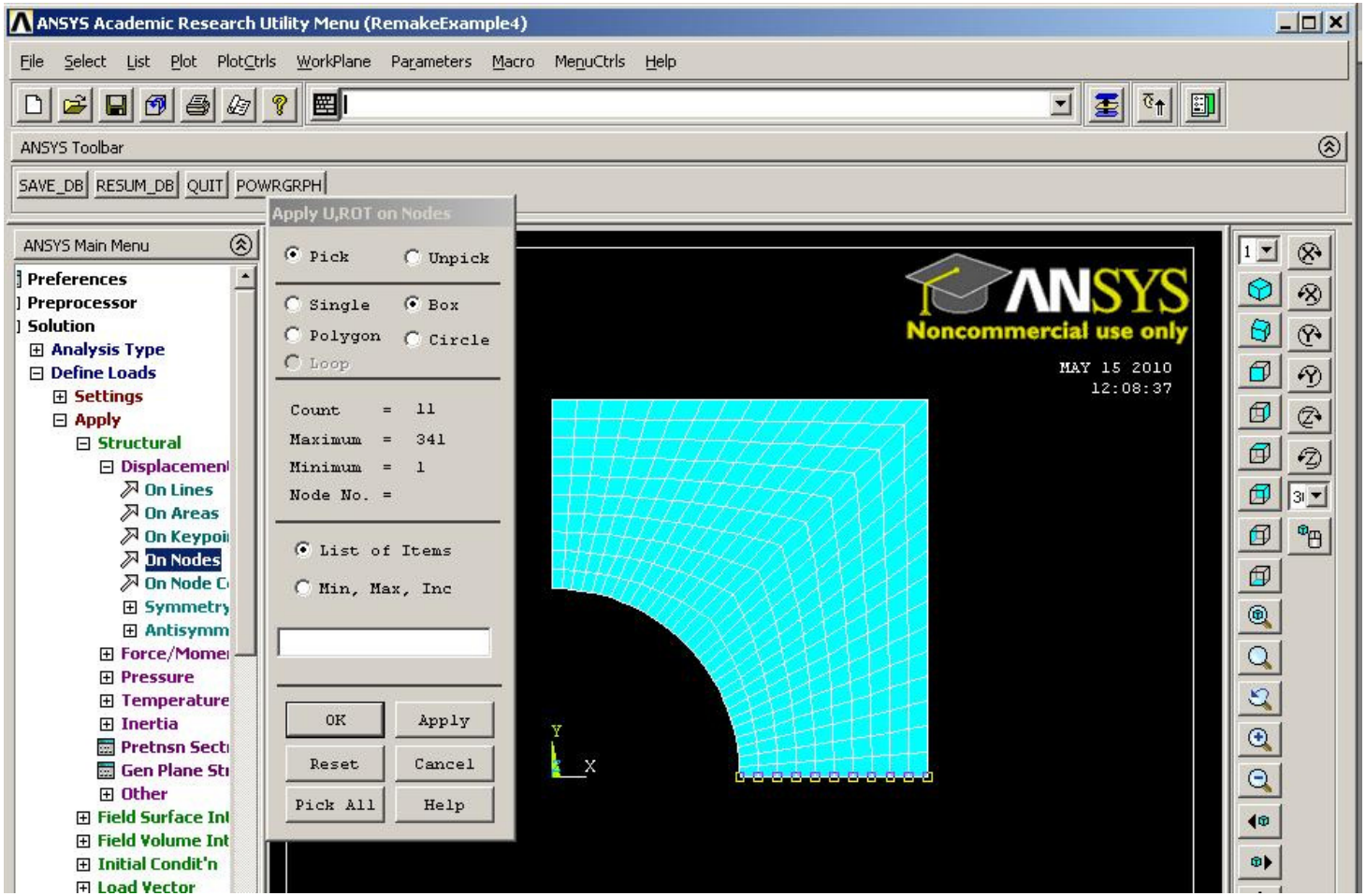
Meshing The Area

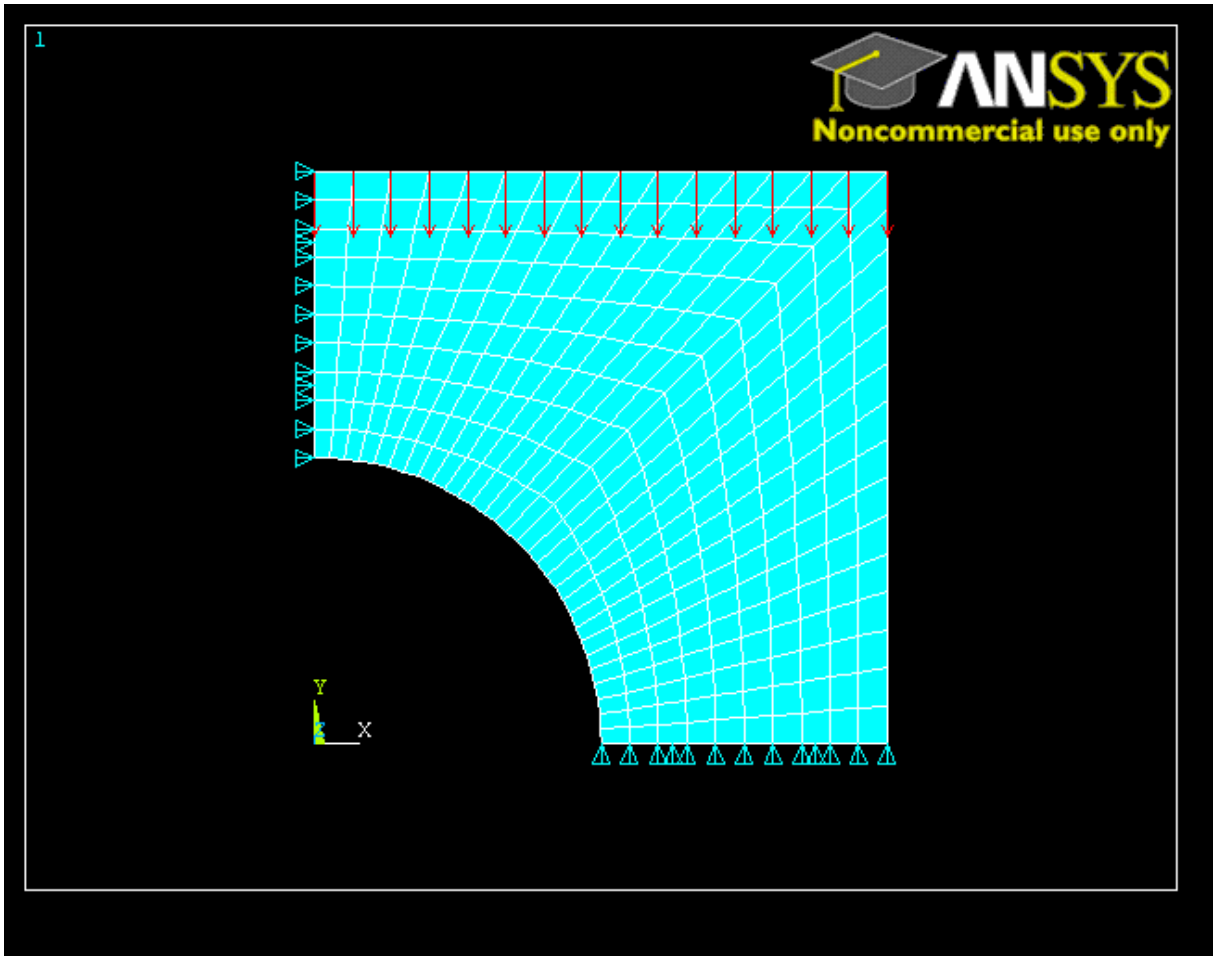


Mesh

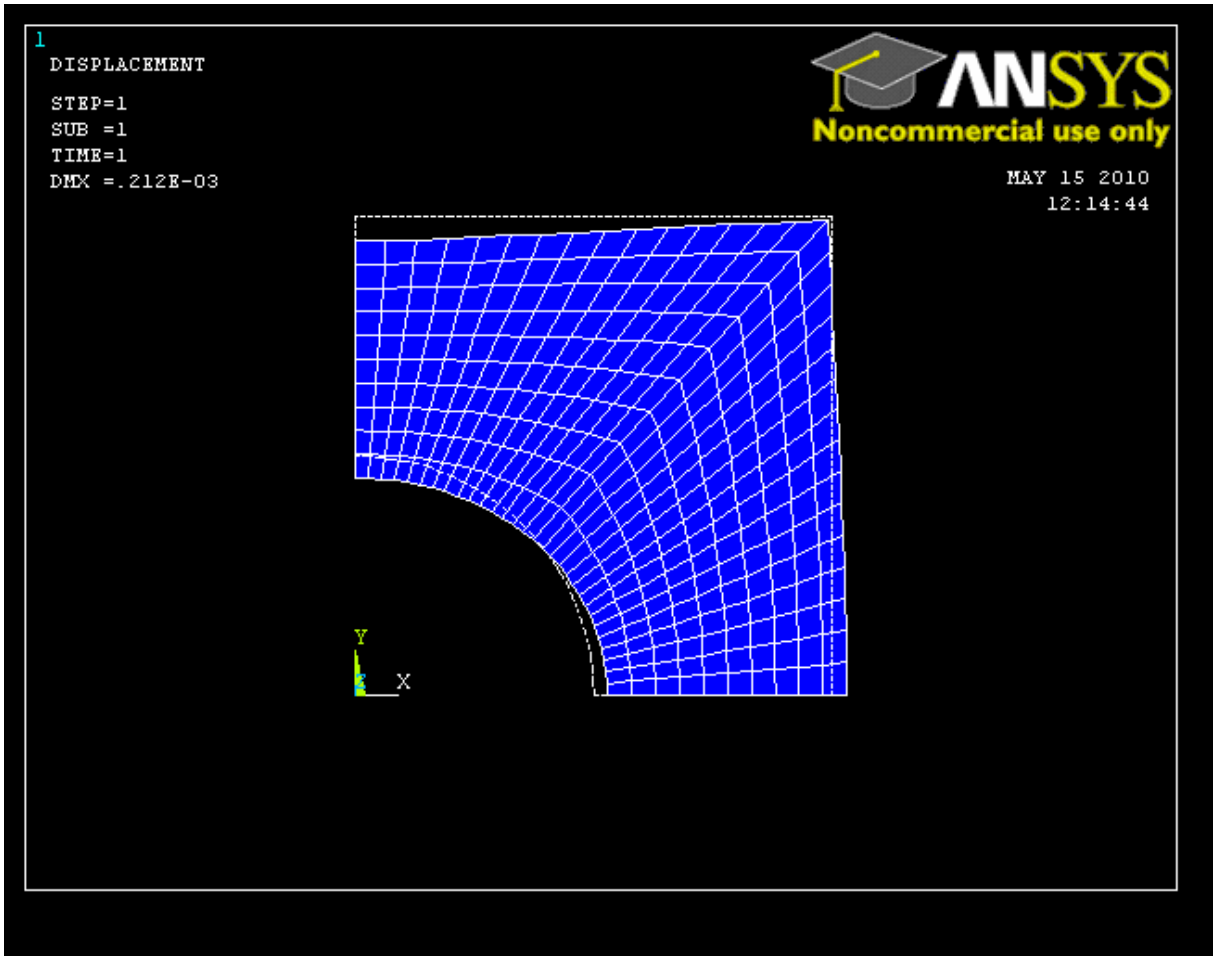


Boundary Conditions

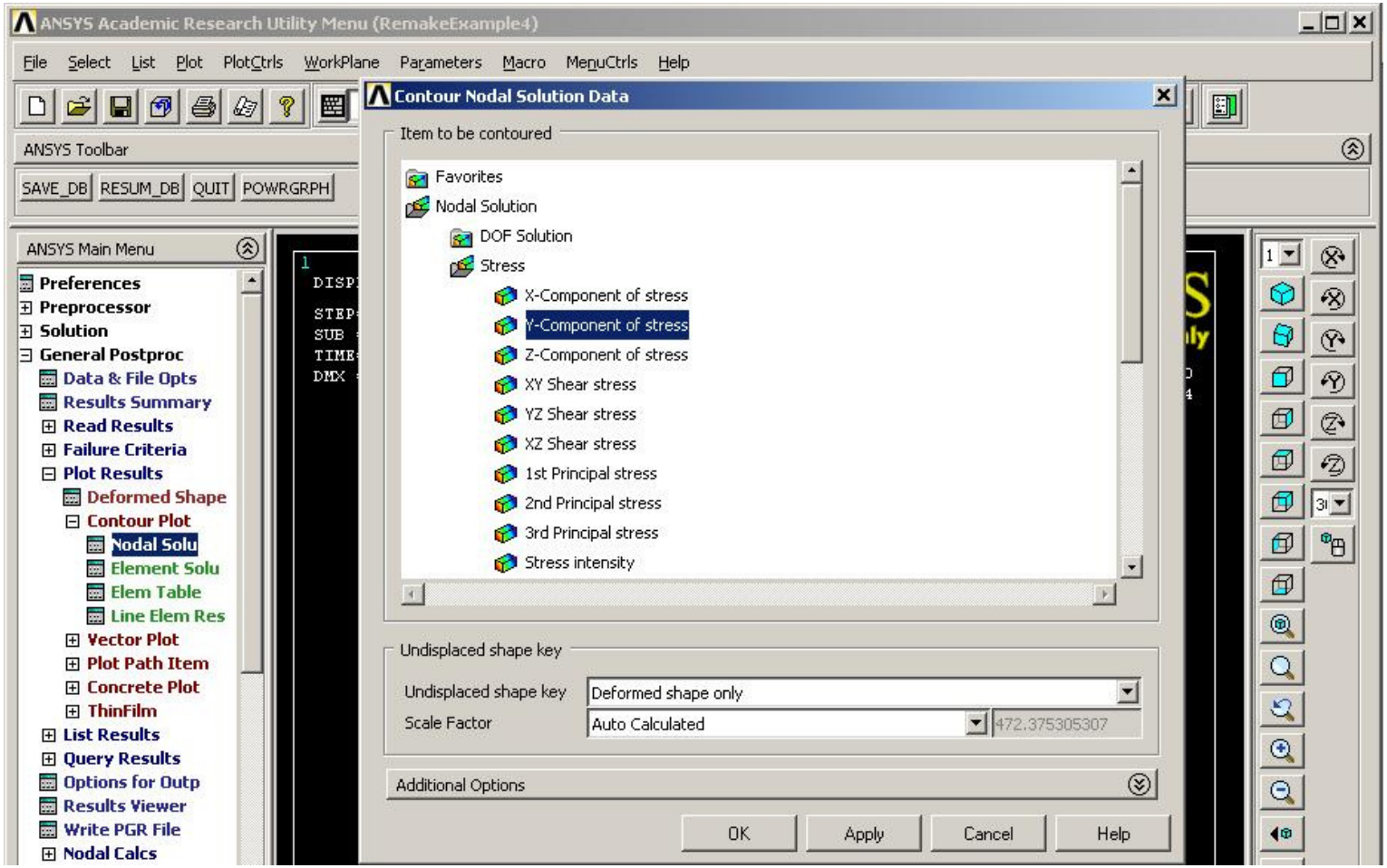



1
ANSYS
Noncommercial use only
Y
X

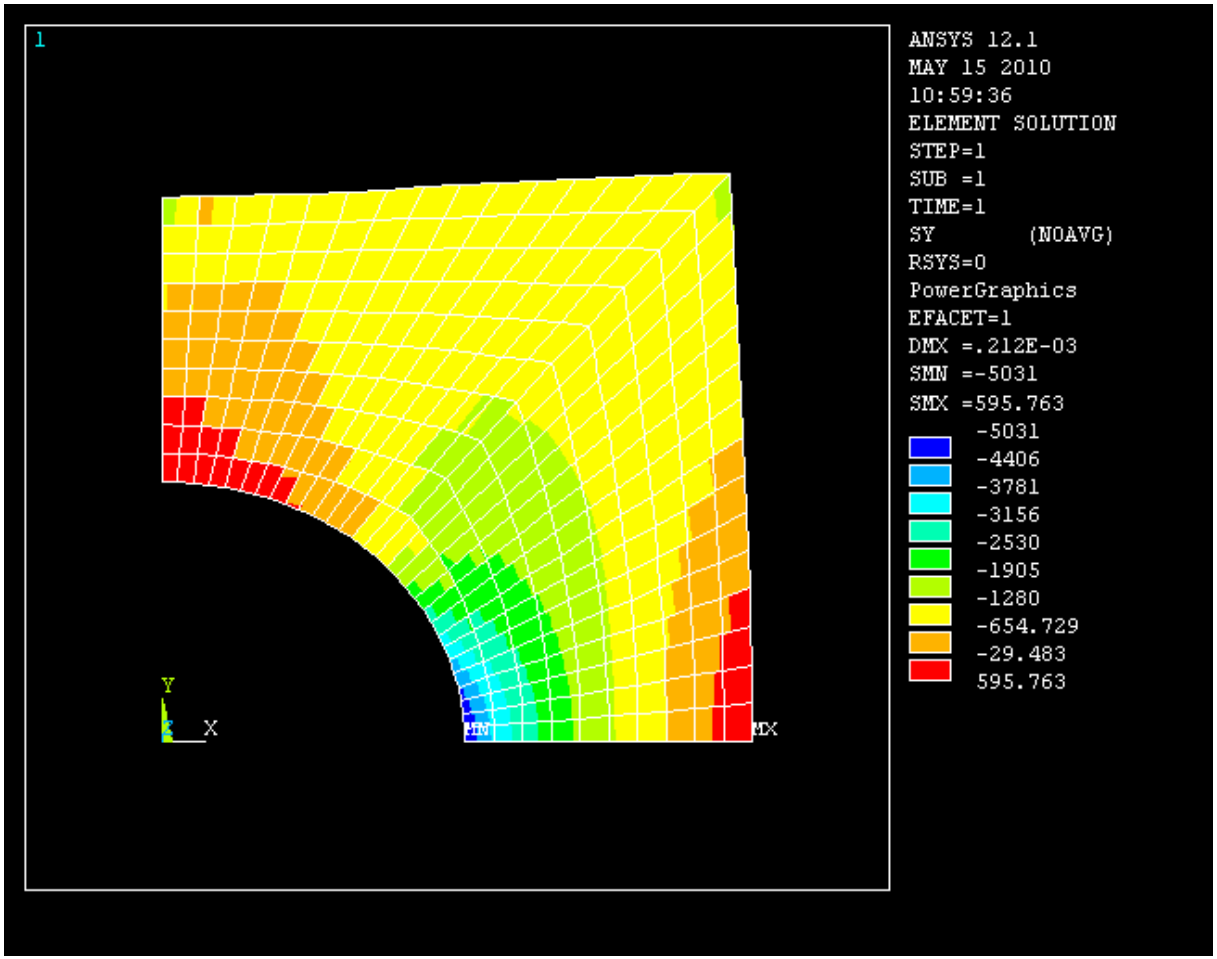
Plot Deformed Shape



Plot Y Element Stress



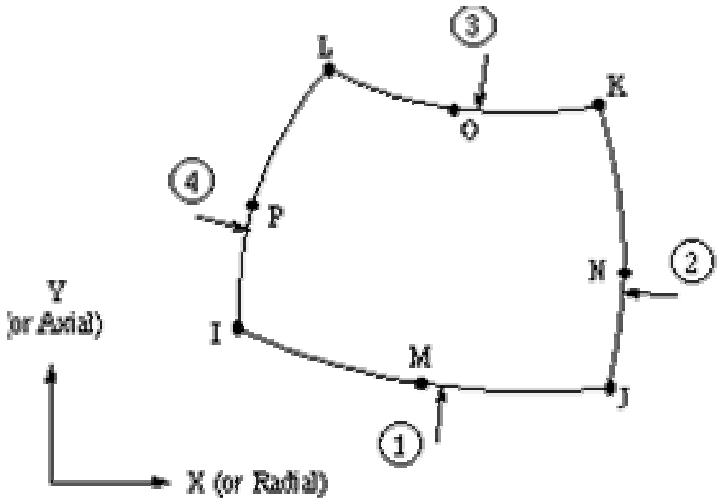
Stress Values

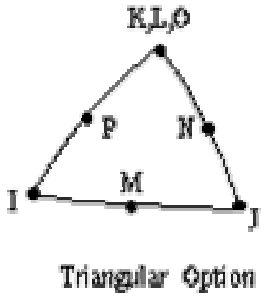


PLANE82 2-D 8-Node Structural Solid

8 –node elements

6 –node triangular elements (option in mesh control stage)





Von Mises Stress

A quantity called the equivalent tensile stress or Von Mises stress is commonly used in solid mechanics.

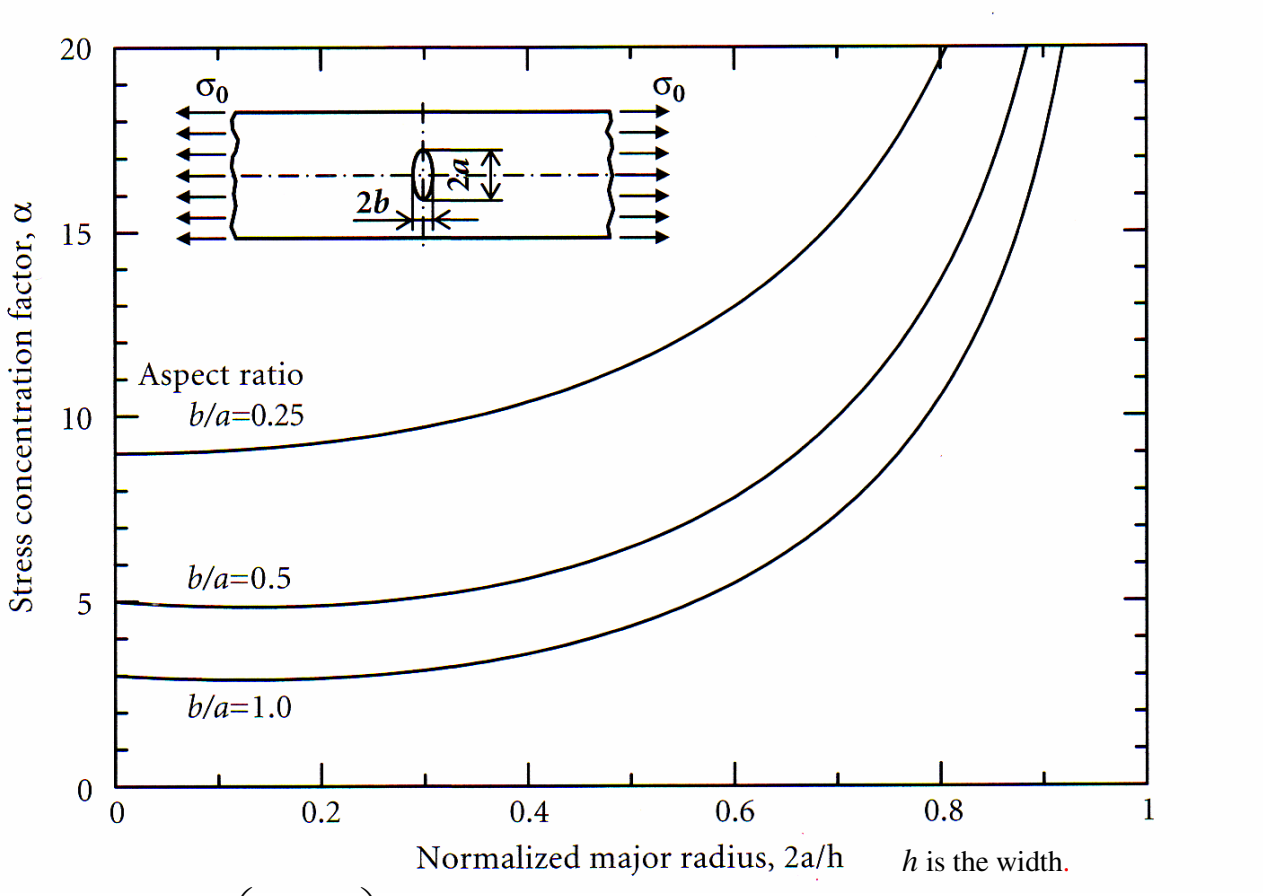
The equivalent tensile stress is defined as

$$\sigma_e = \sqrt{3\ J_2} = \sqrt{\tfrac{1}{2}\ [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}\ .$$

J_2 : second deviatoric stress invariant

$\sigma_{1\text{-}3}$: principal stresses

Theoretical Stress Concentration Formula



$$\sigma_{max} = \sigma_0\left(1+2\frac{a}{b}\right)$$

In the case a=b: $\sigma_{max} = 3\sigma_0$