University of Sussex

Design, Modelling and Manufacture 3

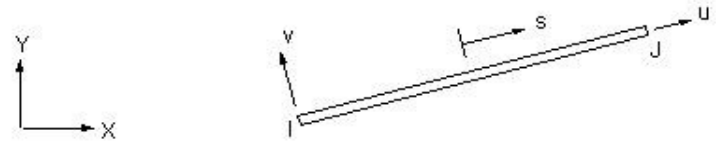
H7082

3

MODELLING AND ANALYSIS OF TRUSS AND BEAM STRUCTURES

TRUSS ELEMENT

14.1. LINK1 - 2-D Spar (or Truss)



Matrix or Vector	Shape Functions	Integration Points
Stiffness Matrix and Thermal Load Vector	Equation 12-1	None
Mass Matrix	Equation 12-1 and Equation 12-2	None
Stress Stiffness Matrix	Equation 12-2	None

Load Type	Distribution
Element Temperature	Linear along length
Nodal Temperature	Linear along length

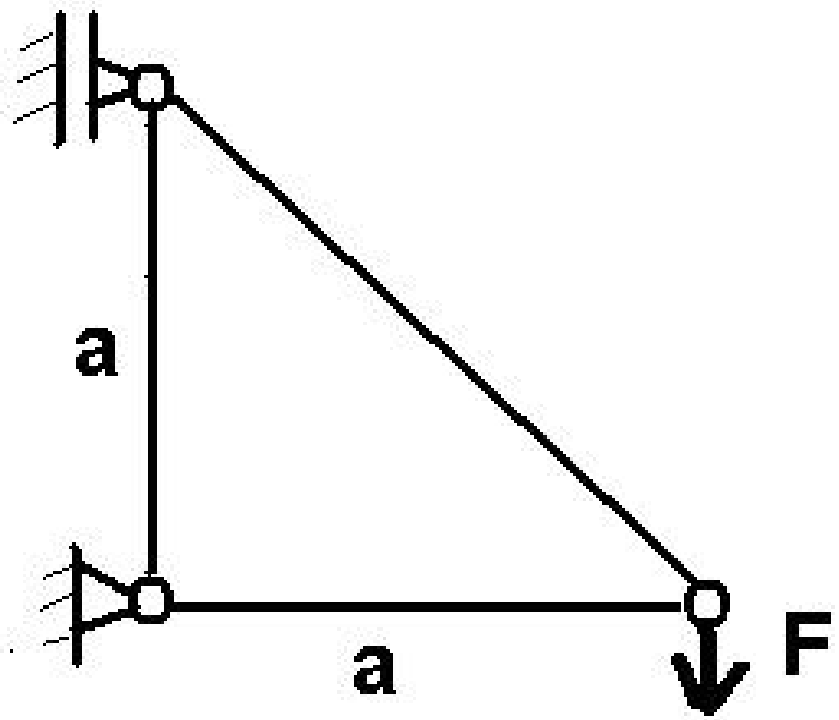
14.1.1. Assumptions and Restrictions

The element is not capable of carrying bending loads. The stress is assumed to be uniform over the entire element.

14.1.2. Other Applicable Sections

LINK8, the 3-D Spar, has analogous element matrices and load vectors described, as well as the stress printout.

Example #1



Data:
A=0.1m^2;
F=1000N;
E=0.7e11N/m^2.
Determine the nodal displacements.

ANSYS Steps

Element selection:
Preprocessor - Element Type – Link 2D Spar

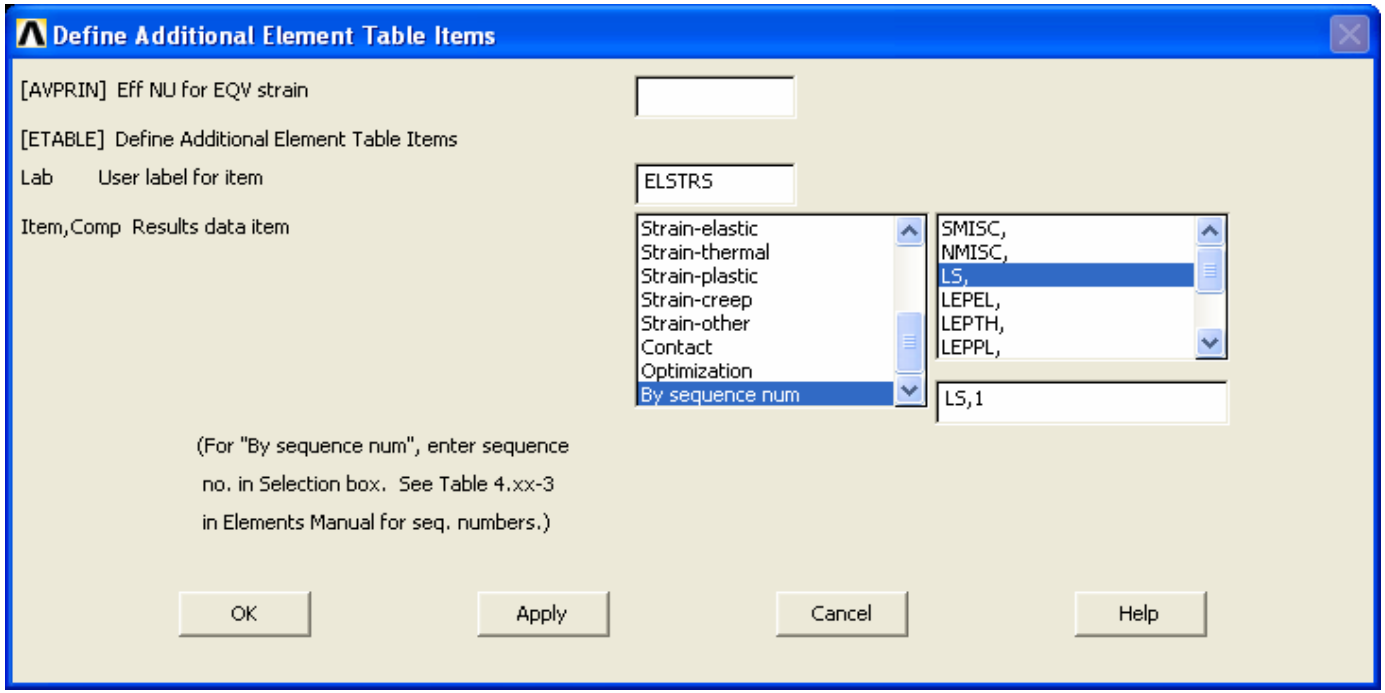
Element materials and gemometry:
Preprocessor – Real Constants - Add – Cross-section area, A, 2*A
Preprocessor – Material Models - ..E, 2E,3E, (v can be ignored in this case)

FE Model
Preprocessor – Modeling – Create –Nodes
Preprocessor – Modeling – Create –Elements –Elem Attributes (define different E and A for different elements)- Auto numbered –Through Nodes

Solve
Solution – Analysis Type –Static – Define Loads – Apply- Structural-Displacement/Force Moment

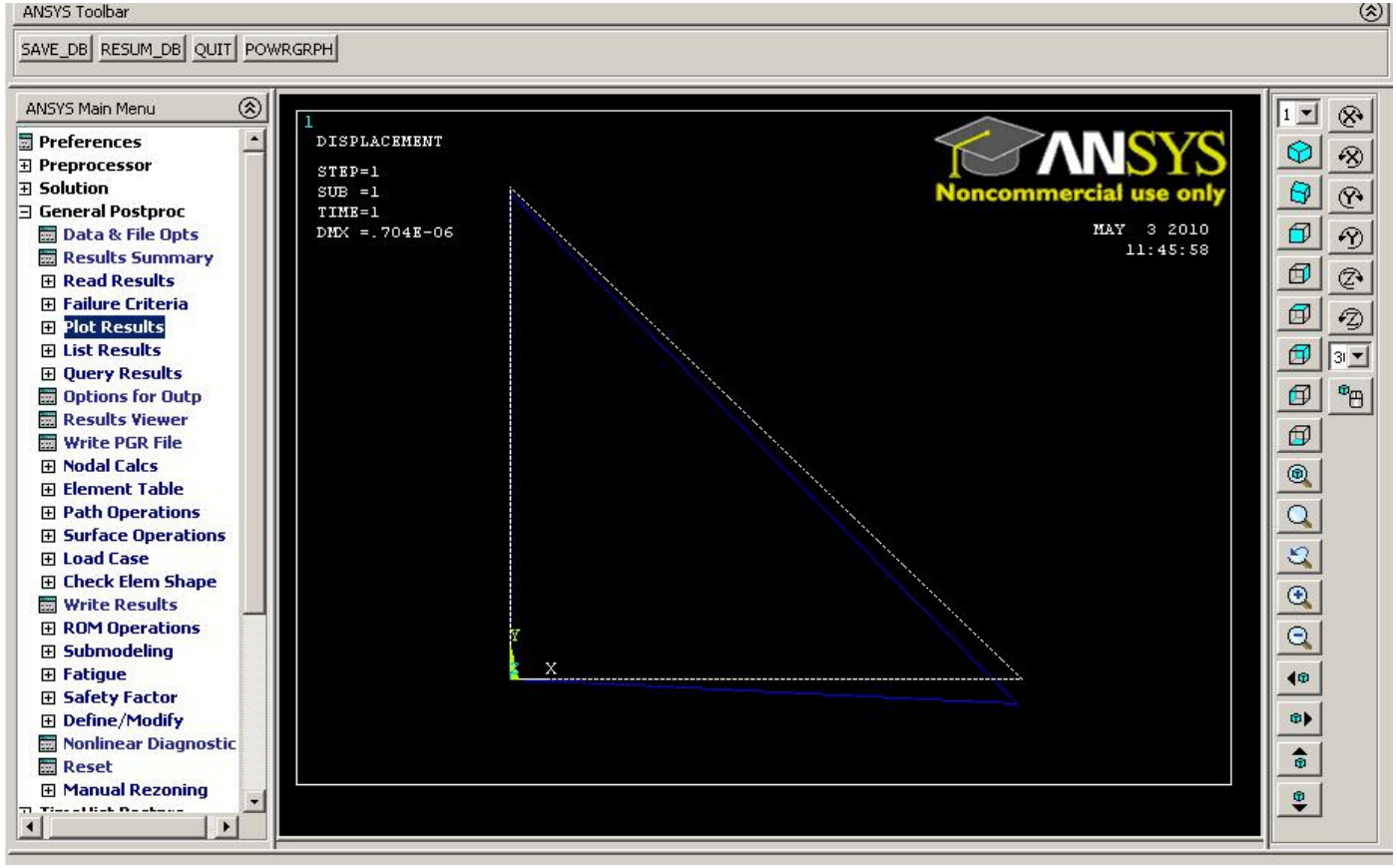
Display/List Results
General PostProc > Plot Results > Deformed Shape…
Plot Ctrls –Numbering –Node numbers – on – Elem –Element numbers

Dialog box for retrieving elements based on sequence numbers

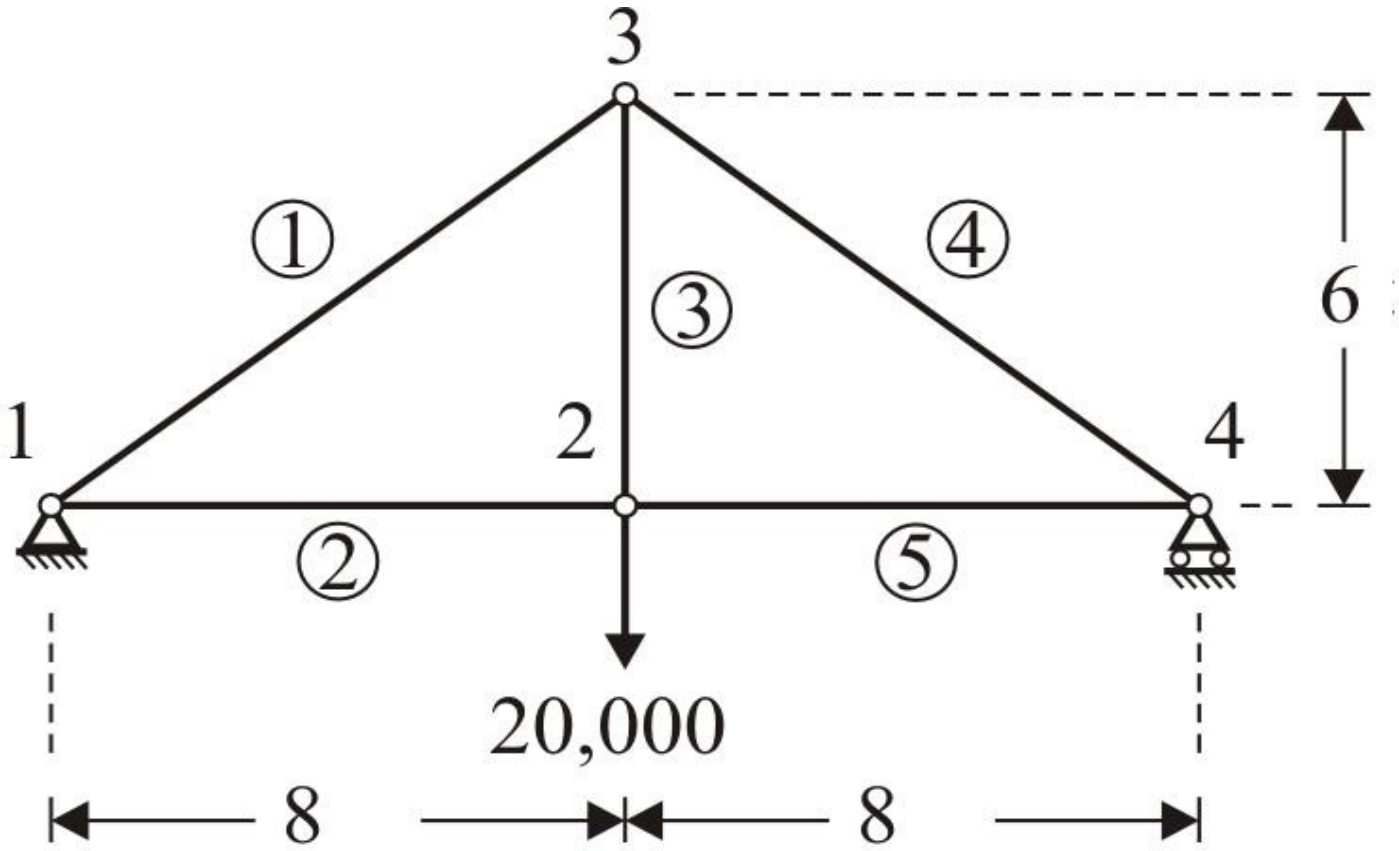


Solution

```
PRINT U    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 THE GLOBAL COORDINATE SYSTEM

   NODE      UX          UY          UZ          USUM
      1   0.0000      0.0000      0.0000      0.0000
      2 -0.14286E-06-0.68978E-06  0.0000     0.70441E-06
      3   0.0000    -0.14286E-06  0.0000     0.14286E-06

MAXIMUM ABSOLUTE VALUES
NODE          2           2           0 |         2
VALUE  -0.14286E-06-0.68978E-06  0.0000     0.70441E-06
```


ANSYS Toolbar
SAVE_DB
RESUM_DB
QUIT
POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
List Results
Query Results
Options for Outp
Results Viewer
Write PGR File
Nodal Calcs
Element Table
Path Operations
Surface Operations
Load Case
Check Elem Shape
Write Results
ROM Operations
Submodeling
Fatigue
Safety Factor
Define/Modify
Nonlinear Diagnostic
Reset
Manual Rezoning
1
DISPLACEMENT
STEP=1
SUB =1
TIME=1
DMX =.704E-06
ANSYS
Noncommercial use only
MAY 3 2010
11:45:58
Y
X

Exercise #1

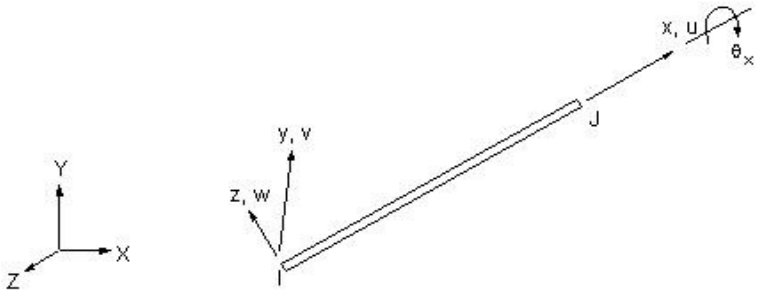


Exercise #1

- Determine the displacements and the stresses in the elements for the steel structure E=2.0e11N/m^2 with the cross section area A=6.4514e-4m^2 for all elements except A3=2A.

BEAM ELEMENT

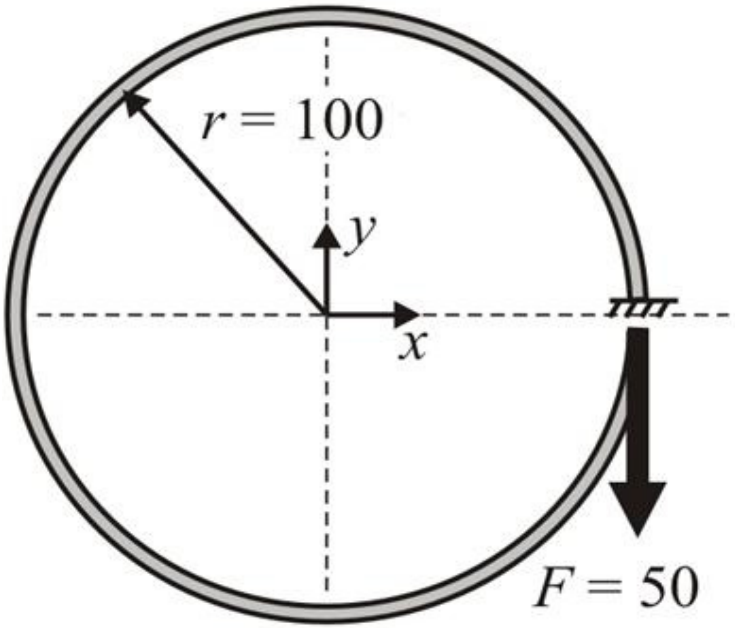
14.4. BEAM4 - 3-D Elastic Beam

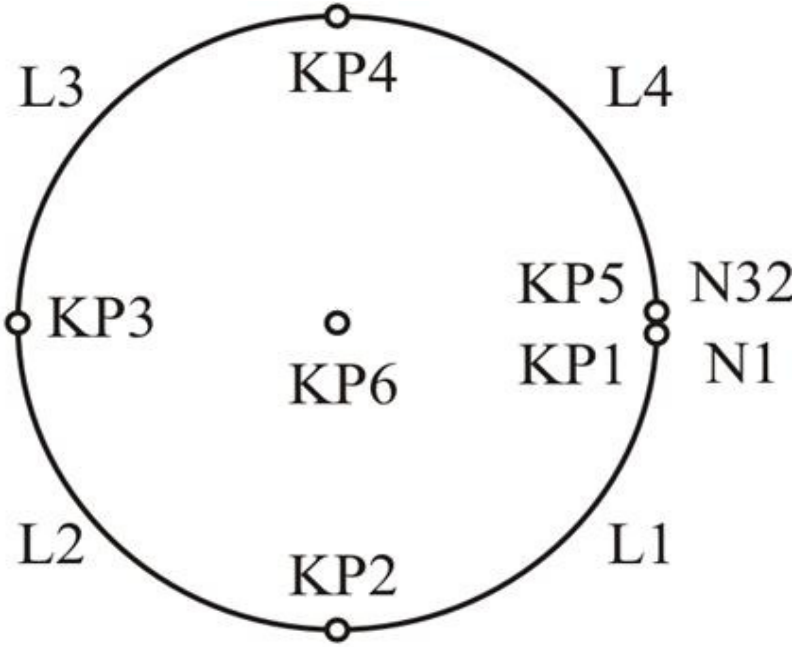


Matrix or Vector	Shape Functions	Integration Points
Stiffness and Mass Matrices	Equation 12-15, Equation 12-16, Equation 12-17, and Equation 12-18	None
Stress Stiffness and Damping Matrices	Equation 12-7 and Equation 12-8	None
Pressure Load Vector and Temperatures	Equation 12-15, Equation 12-16, and Equation 12-17	None

Load Type	Distribution
Element Temperature	Bilinear across cross-section, linear along length
Nodal Temperature	Constant across cross-section, linear along length
Pressure	Linear along length

Example #2

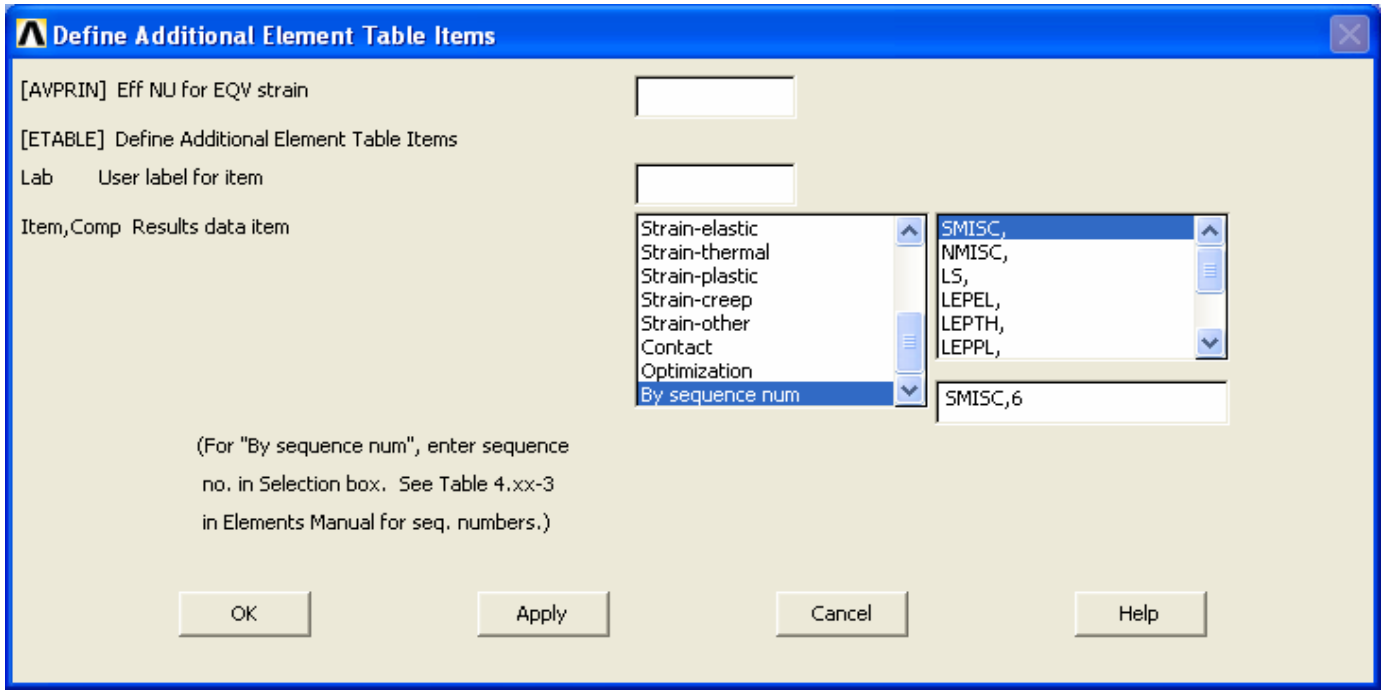




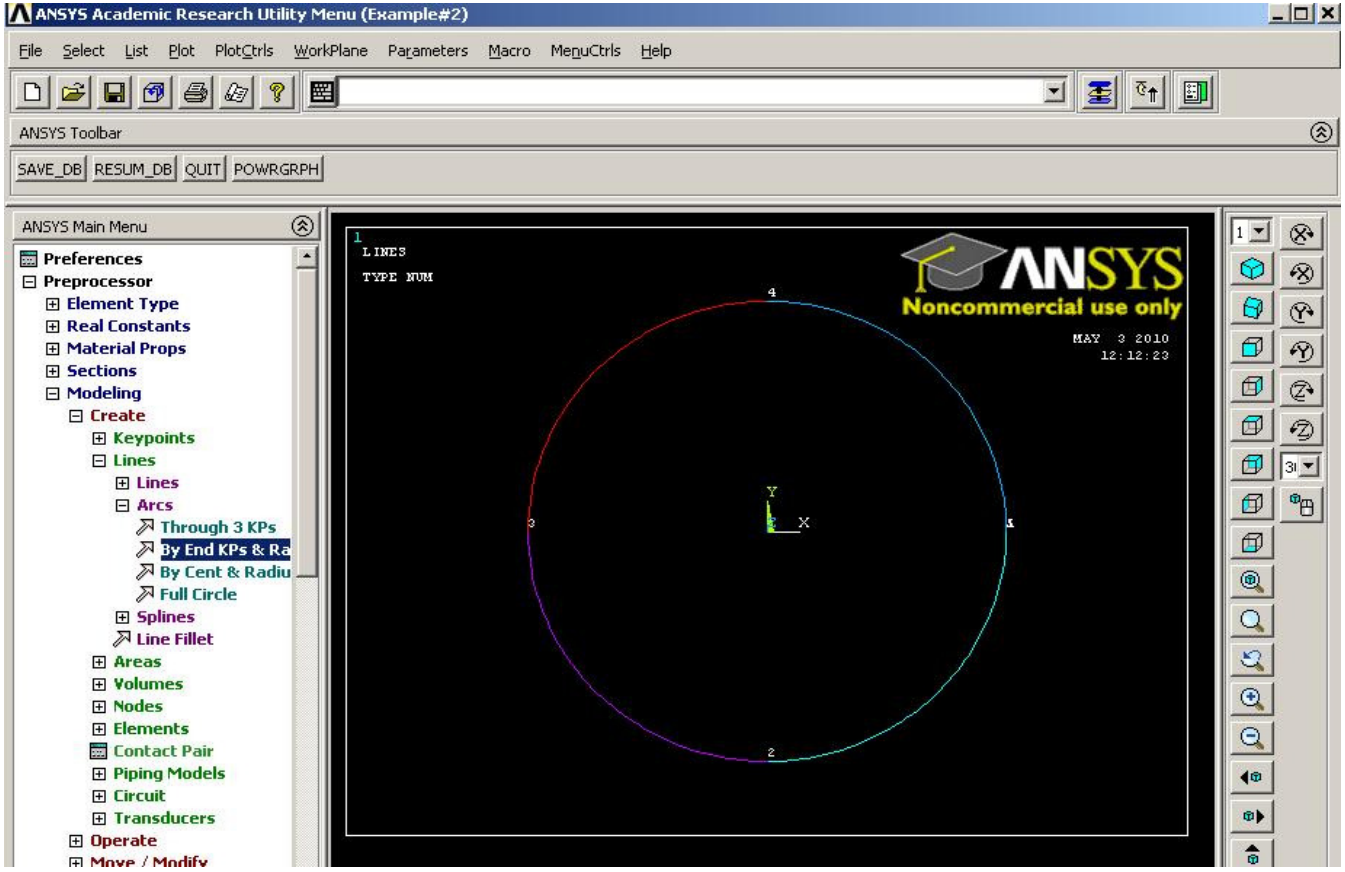
Example #2

- For the slit ring steel structure determine the moment diagram. The ring has a solid cross section with radius 2.54 cm. The structure is modeled using beam elements with the area A=20.2683e-4m^2, elastic modulus E=2.0e11N/m^2, Poisson's ratio 0.3 and moment of inertia Izz=32.6907e-8m^4

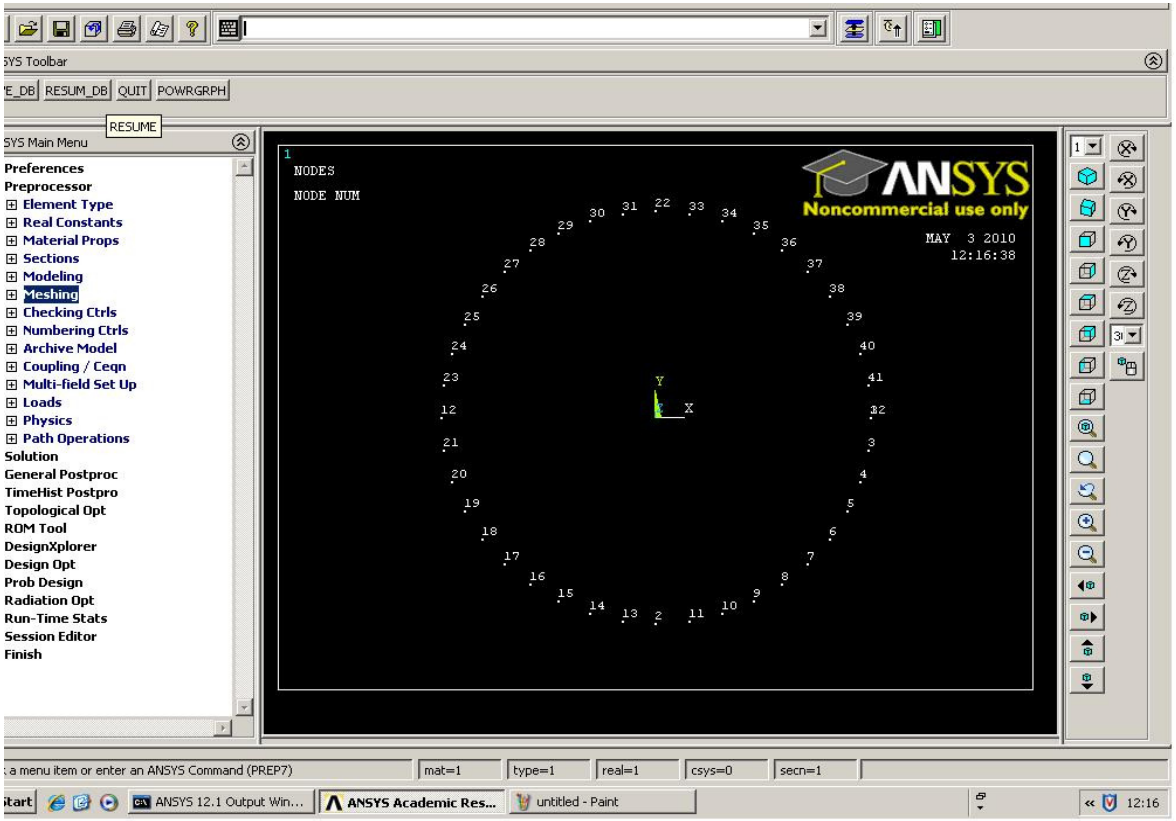
Dialog box for extracting nodal moment values



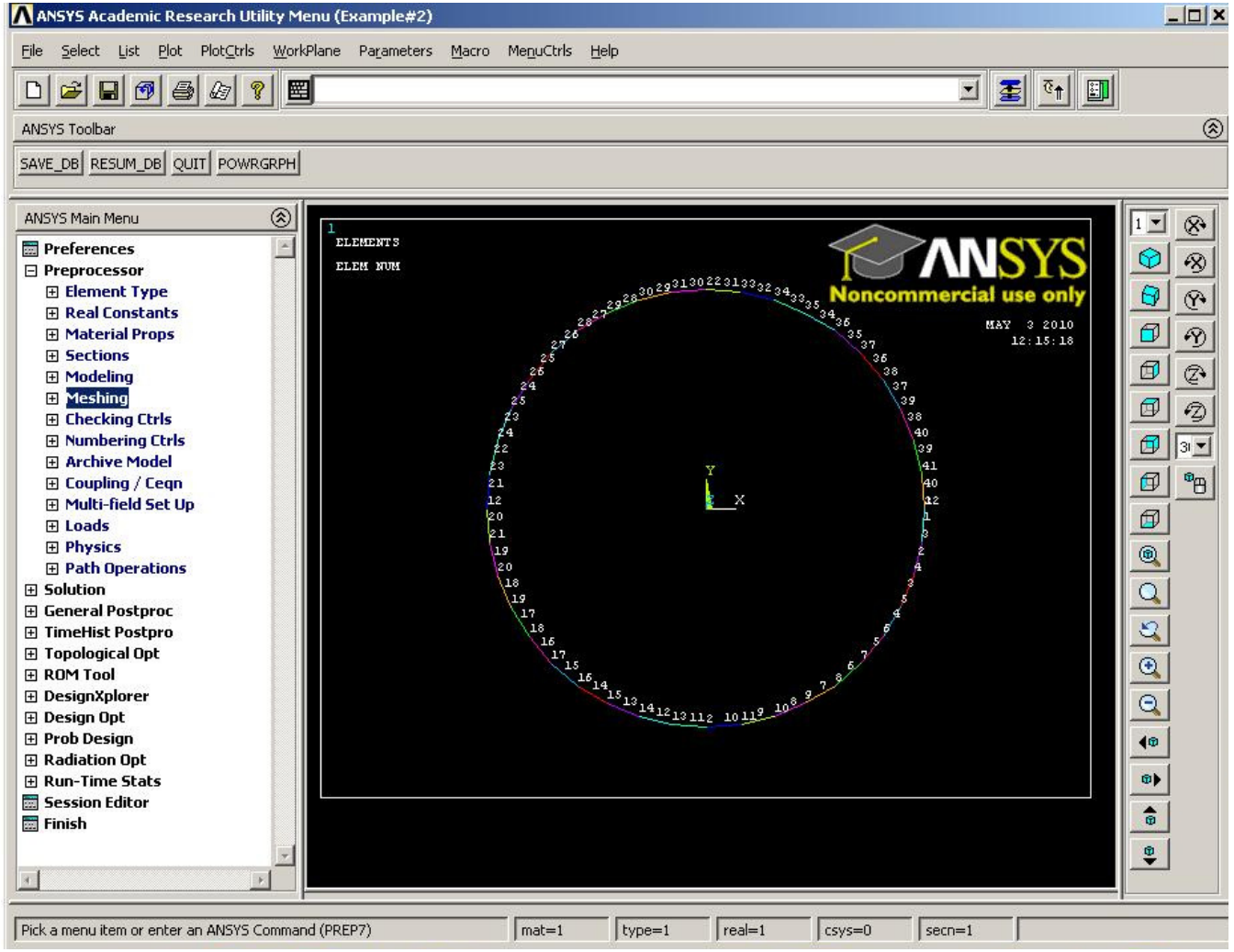
Lines & KT


ANSYS Academic Research Utility Menu (Example#2)
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
Lines
Arcs
Through 3 KPs
By End KPs & Ra
By Cent & Radiu
Full Circle
Splines
Line Fillet
Areas
Volumes
Nodes
Elements
Contact Pair
Piping Models
Circuit
Transducers
Operate
Move / Modify
1
LINES
TYPE NUM
ANSYS
Noncommercial use only
MAY 3 2010
12:12:23

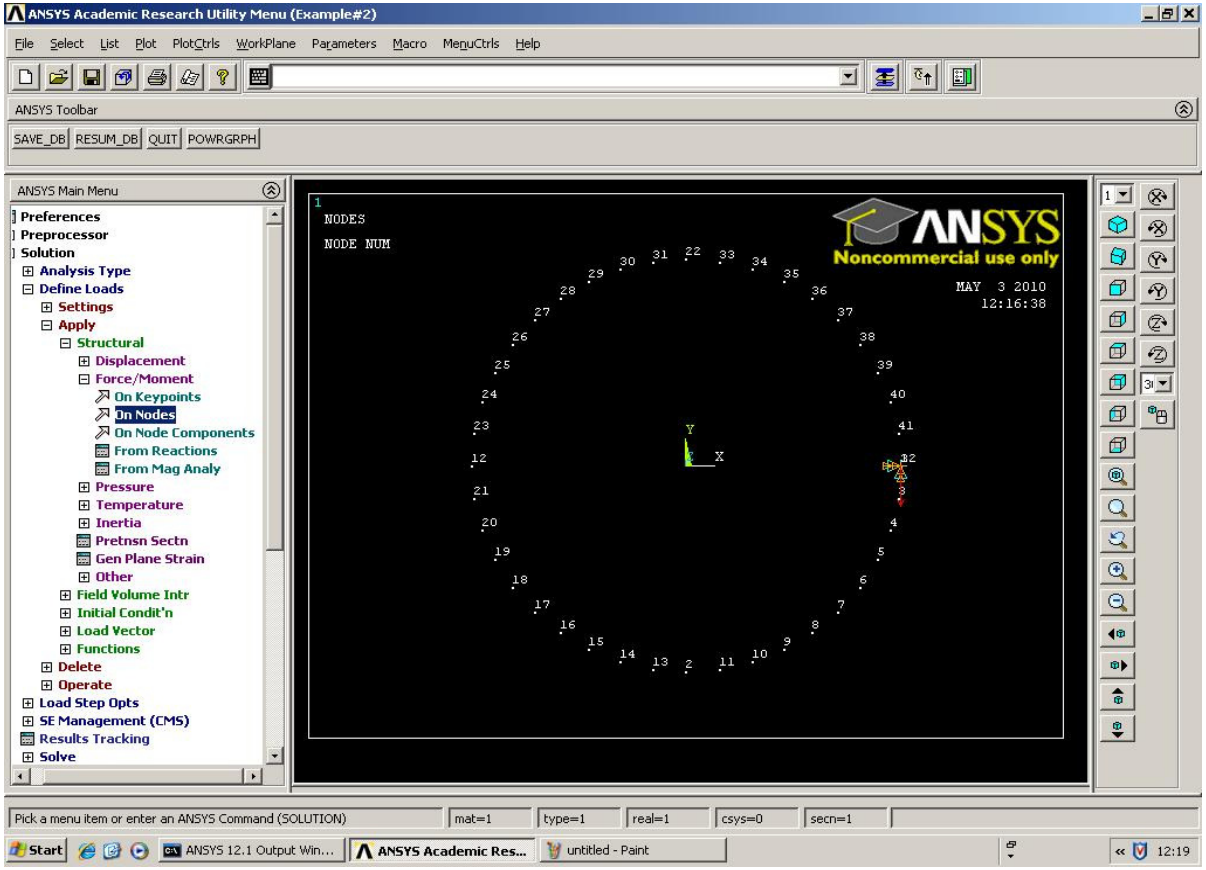
Nodes


SYS Toolbar
RESUME_DB
QUIT
POWRGRPH
RESUME
SYS Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Meshing
Checking Ctrls
Numbering Ctrls
Archive Model
Coupling / Ceqn
Multi-field Set Up
Loads
Physics
Path Operations
Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
DesignXplorer
Design Opt
Prob Design
Radiation Opt
Run-Time Stats
Session Editor
Finish
NODES
NODE NUM
ANSYS
Noncommercial use only
MAY 3 2010
12:16:38
a menu item or enter an ANSYS Command (PREP7)
mat=1
type=1
real=1
csys=0
secn=1
ANSYS 12.1 Output Win...
ANSYS Academic Res...
untitled - Paint
12:16

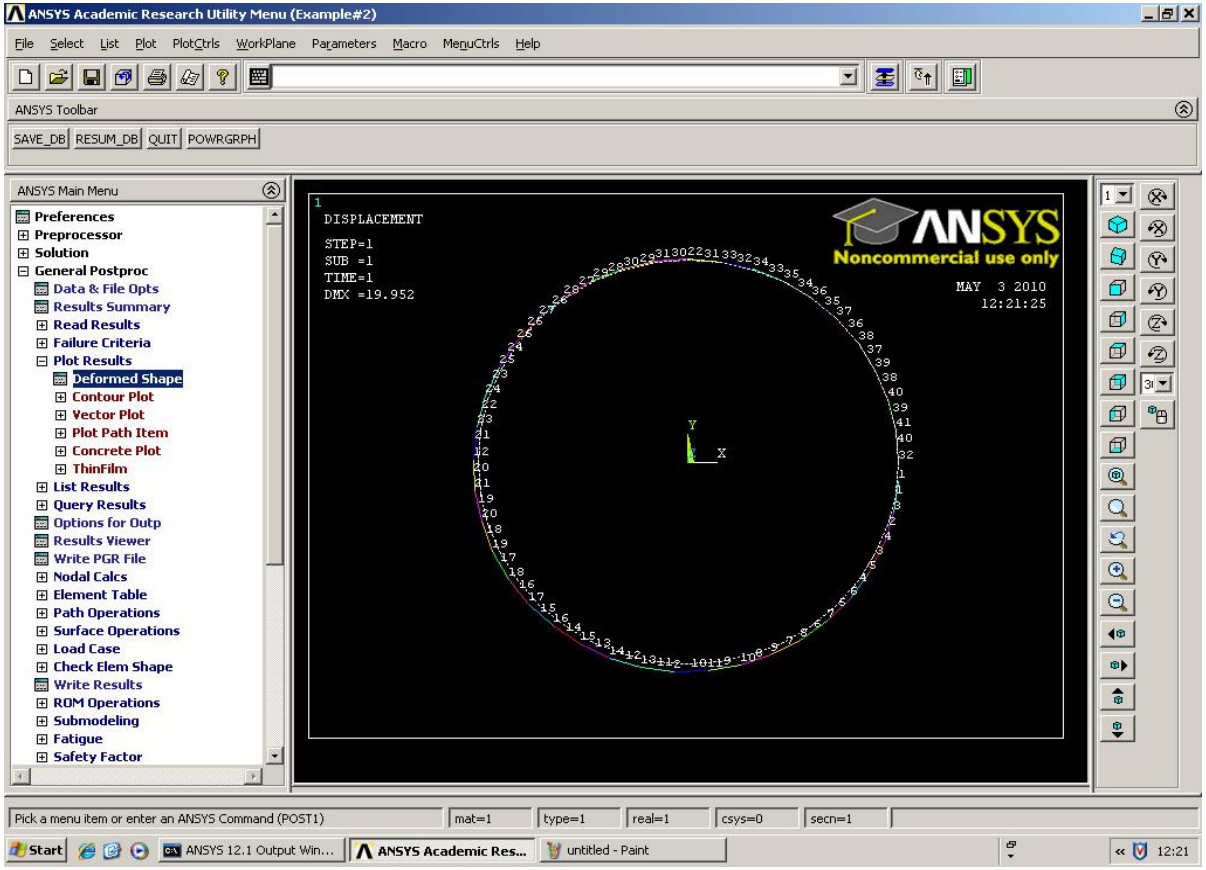
Elements


ANSYS Academic Research Utility Menu (Example#2)
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
Meshing
Checking Ctrls
Numbering Ctrls
Archive Model
Coupling / Ceqn
Multi-field Set Up
Loads
Physics
Path Operations
Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
DesignXplorer
Design Opt
Prob Design
Radiation Opt
Run-Time Stats
Session Editor
Finish
ELEMENTS
ELEM NUM
ANSYS
Noncommercial use only
MAY 3 2010
12:15:18
Pick a menu item or enter an ANSYS Command (PREP7)
mat=1
type=1
real=1
csys=0
secn=1

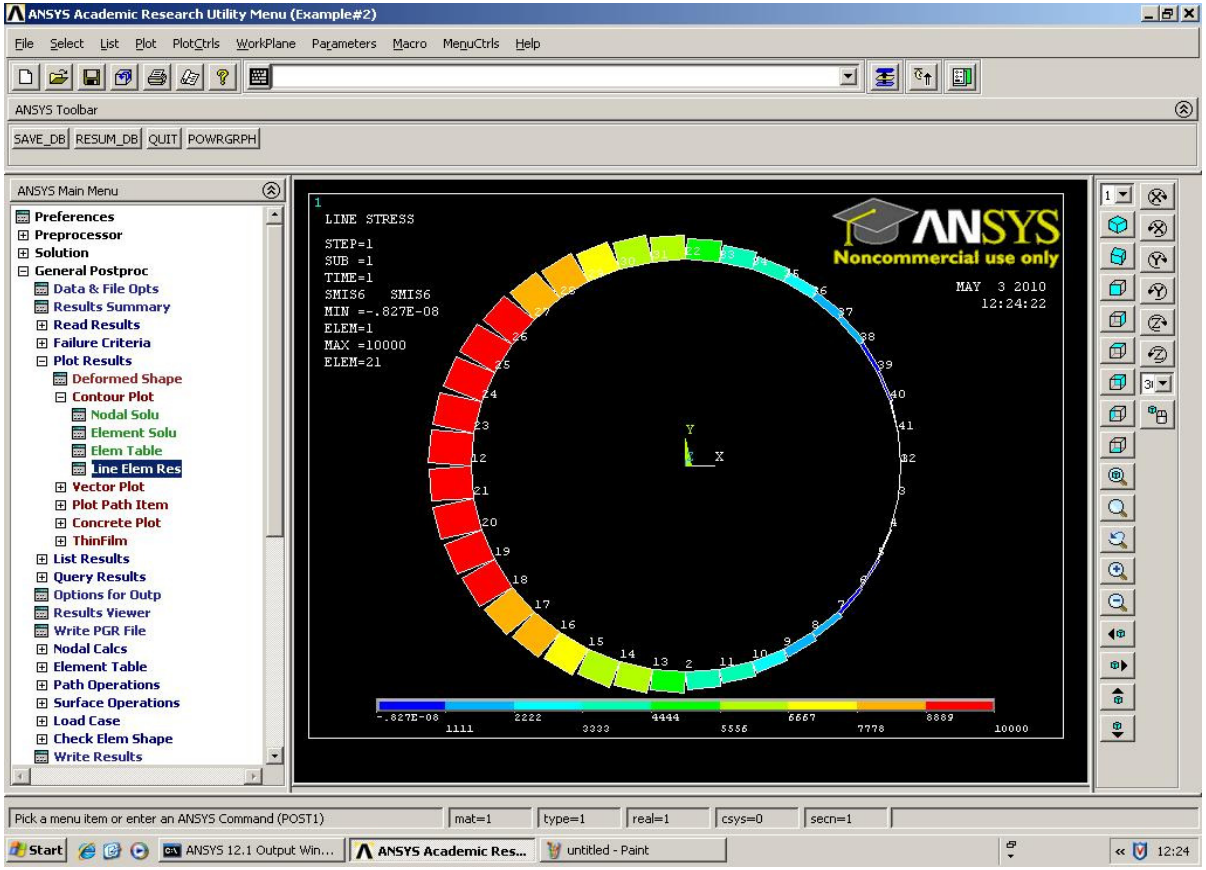
BC&Loads


ANSYS Academic Research Utility Menu (Example#2)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Solution
Analysis Type
Define Loads
Settings
Apply
Structural
Displacement
Force/Moment
On Keypoints
On Nodes
On Node Components
From Reactions
From Mag Analy
Pressure
Temperature
Inertia
Pretnsn Sectn
Gen Plane Strain
Other
Field Volume Intr
Initial Condit'n
Load Vector
Functions
Delete
Operate
Load Step Opts
SE Management (CMS)
Results Tracking
Solve
NODES
NODE NUM
ANSYS
Noncommercial use only
MAY 3 2010
12:16:38
Pick a menu item or enter an ANSYS Command (SOLUTION)
mat=1
type=1
real=1
csys=0
secn=1
Start
ANSYS 12.1 Output Win...
ANSYS Academic Res...
untitled - Paint
12:19

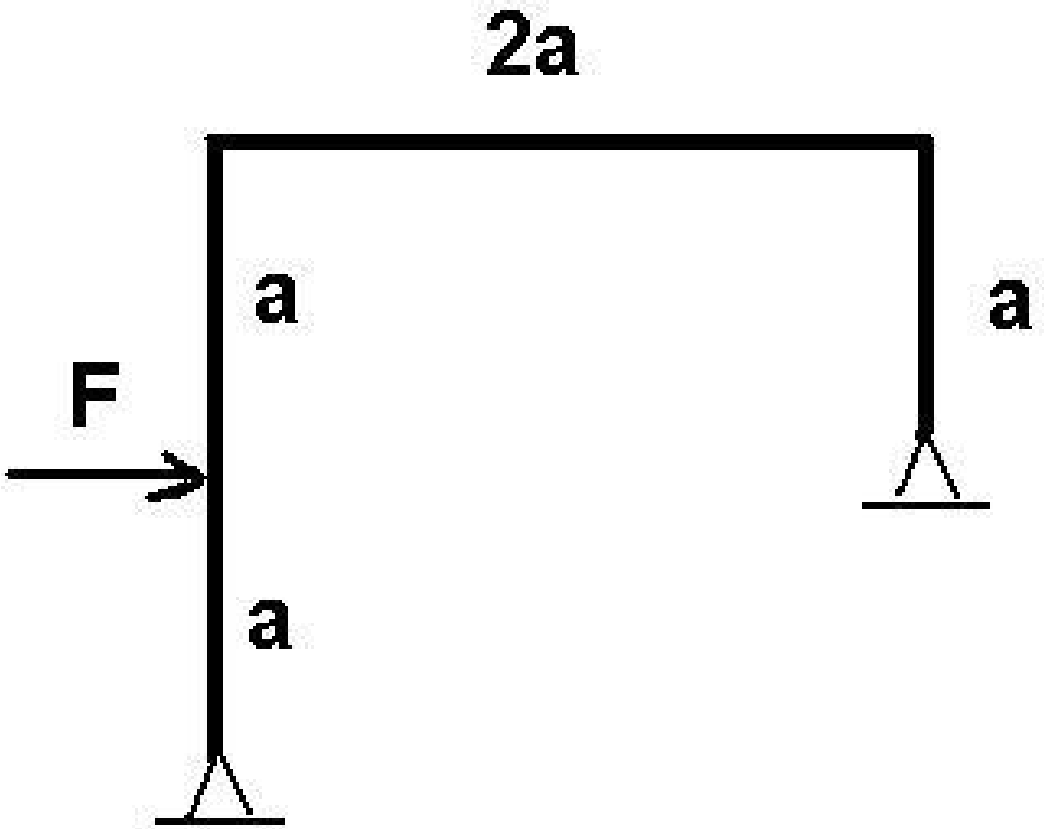
Deformed shape



Moment diagram



Exercise #2



Data:
F=1000N;a=1m;
E=2e11N/m^2
A=1600mm^2;
I=2.0e5mm^4.

Determine the moment diagram

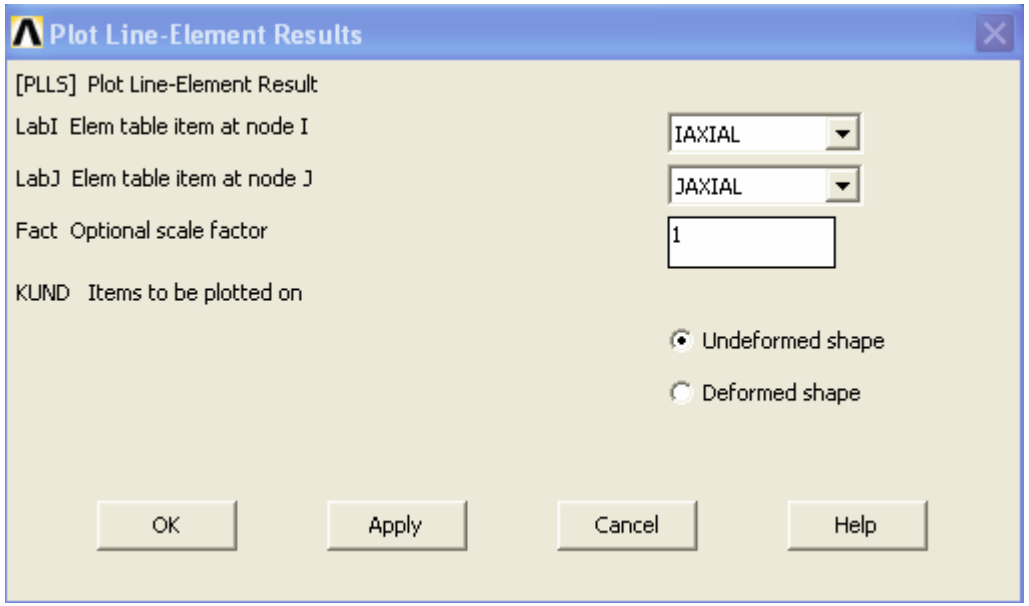
Beam's shear, axial forces and moment plots

Table 7.2 BEAM4 (KEYOPT(9) = 0) Item and Sequence Numbers for the ETABLE and ESOL Commands

MFORX	SMISC	-	1	7
MFORY	SMISC	-	2	8
MFORZ	SMISC	-	3	9
MMOMX	SMISC	-	4	10
MMOMY	SMISC	-	5	11
MMOMZ	SMISC	-	6	12
P1	SMISC	-	13	14
OFFST1	SMISC	-	15	16
P2	SMISC	-	17	18
OFFST2	SMISC	-	19	20
P3	SMISC	-	21	22

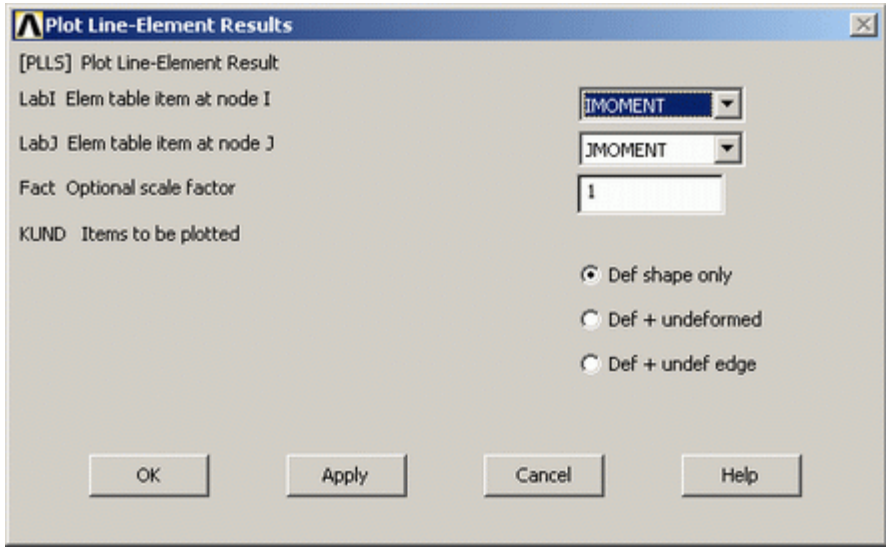
Axial Force

Genrral Prostproc > Element Table> Define Table> Add >
IAXIAL >By sequence num > SMISC, 1 > Apply ,
JXIAL> By sequence num> SMISC, 7 >Apply >OK>
Genrral Prostproc > Plot Results> Contour Plot > Line Element Res >Choose IAXIAL and JAXIAL> OK



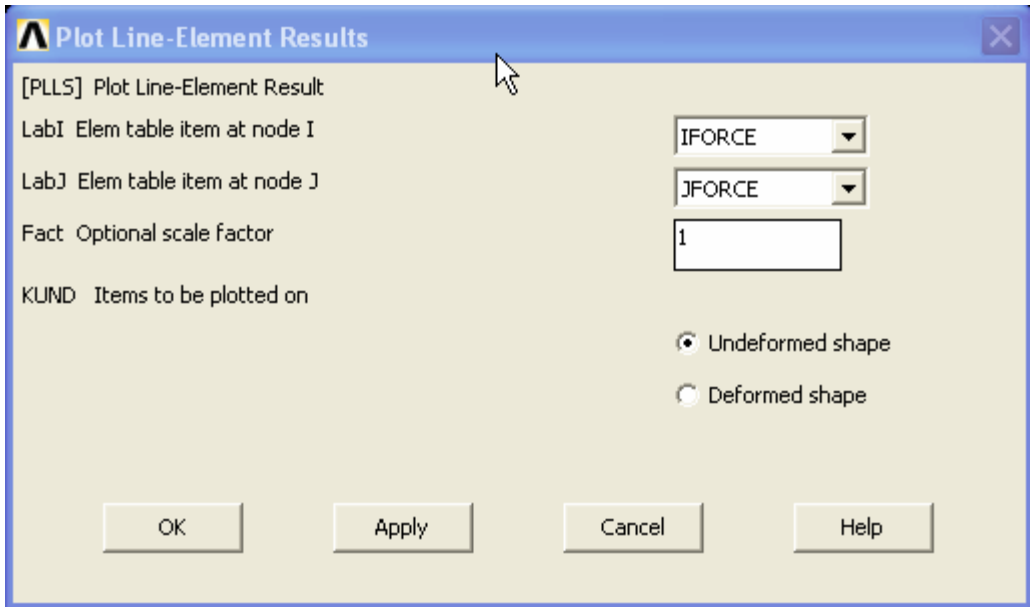
Bending Moment

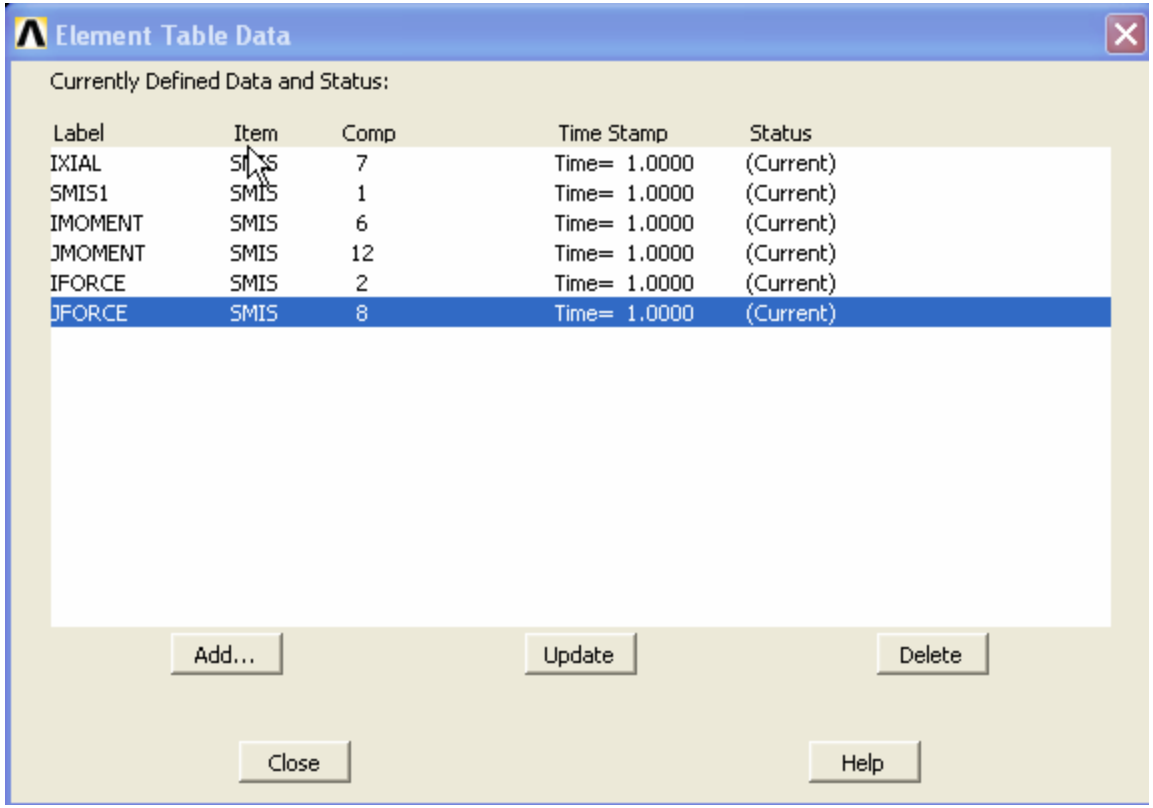
Genrral Prostproc > Element Table> Define Table> Add >
IMOMENT >By sequence num > SMISC, 6 > Apply ,
JMOMENT > By sequence num> SMISC, 12 >Apply >OK>
Genrral Prostproc > Plot Results> Contour Plot > Line Element Res >Choose IMOMENT and JMOMENT > OK



Shear force

Genrral Prostproc > Element Table> Define Table> Add >
IFORCE >By sequence num > SMISC, 2 > Apply ,
JFORCE > By sequence num> SMISC, 8 >Apply >OK>
Genrral Prostproc > Plot Results> Contour Plot > Line Element Res >Choose IFORCE and JFORCE > OK




Element Table Data
Currently Defined Data and Status:
Label Item Comp Time Stamp Status
IXIAL SMIS 7 Time= 1.0000 (Current)
SMIS1 SMIS 1 Time= 1.0000 (Current)
IMOMENT SMIS 6 Time= 1.0000 (Current)
JMOMENT SMIS 12 Time= 1.0000 (Current)
IFORCE SMIS 2 Time= 1.0000 (Current)
JFORCE SMIS 8 Time= 1.0000 (Current)
Add...
Update
Delete
Close
Help