WORCESTER POLYTECHNIC INSTITUTE
MECHANICAL ENGINEERING DEPARTMENT

INTRODUCTION TO THE FEM with ANSYS v9.0

COURSE No.:	ME-3320, Term A, 2005	**NAME:** ____________________________
COURSE NAME:	Design of machine elements	**INSTRUCTOR:** C. Furlong HL-151
DATE:	04 October 2005	(508) 831-5126 *cfurlong@wpi.edu*
References:	1) Ansys. User's Guide, v9.0 2) Ch. 16: R. L. Norton, *Machine Design*. 3th Ed., Prentice-Hall, 2005 3) H. Grandin, Jr., *Fundamentals of the Finite Element Method*, Macmillan, 1986	**TA:** M. Rodgers HL-310 (508) 831-5976 *mrodgers@wpi.edu*

The finite element method (FEM) is a numerical solution method that originated in the field of structural engineering. Because of rapid advances in computer technologies, FEM has become a method widely used in many science and engineering fields for solving problems governed by complex partial differential equations.

In the FEM, the domain of interest is divided into small subregions (elements) and the governing equations are solved for each subregion. Very much simpler functions are utilized to represent the solution in each subregion than the functions required to represent the solution for the entire domain. The subregions are joined together mathematically by applying conditions that make each element boundary compatible with each of its neighbors while satisfying the overall imposed boundary conditions and loads.

Detailed discussions about the mathematical formulations used in FEM are beyond the scope of this tutorial. If interested, students are strongly encouraged to take courses on FEM at WPI, including

ME-4512, Introduction to the Finite Element Method;
ME-515, Computational Methods for Partial Differential Equations;

In this tutorial, you will perform static and modal analysis with FEM by use of the ANSYS computer program available at WPI.

ANSYS is a complete, modular, self-contained finite element computer program developed by Ansys, Inc. for personal computers and workstations. The program includes modules to solve linear and nonlinear static and dynamic structural problems, in addition to problems of heat transfer, fluid mechanics, electromagnetics, and design optimization. Modules for such special analysis options as fatigue are available.

Typical Flowchart used for FEM Modeling
(used in this tutorial)

Pre-processing

Definition of:

a. Geometry (domain)
b. Loads
c. Boundary conditions
d. Material properties
e. Mathematical approximation to be used (e.g., element type)
f. Domain discretization (i.e., mesh generation)

↓

Solution

Definition of:

a. Type of analysis (e.g., physics to be considered)
b. Solution type (e.g., steady-state, time-dependent, optimization, etc.)
c. Selection of solvers for the governing equations
d. Parameters for the solvers
e. Solution of governing equations
f. Output options (e.g., writing solutions into files in specific format)

↓

Post-processing

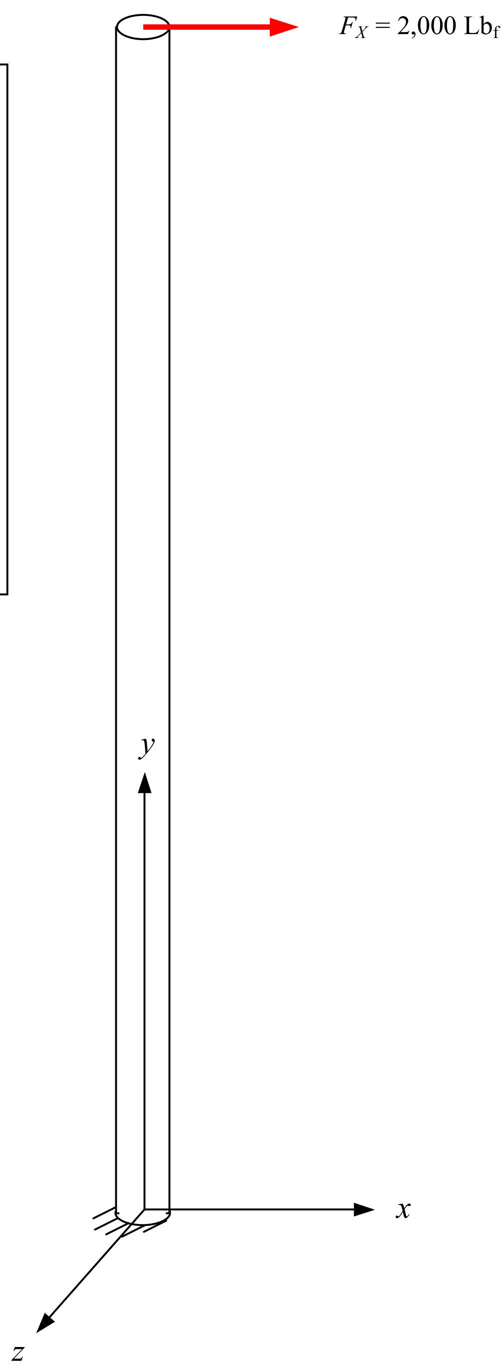
a. Reading results of the solution
b. Graphical display or results
c. Interpretation and validation of the results (***verifying the accuracy of the solutions***)

Definition of the Model

Problem: determine deformations, stresses, and the first 5 fundamental natural frequencies of vibration of the column shown.

Column has a constant diameter of 15" and a height of 350". Material is steel with E = 30 Mpsi, $\mu = 0.33$, Sy = 40 kpsi, Sut = 72 kpsi, and γ= 0.28 lb$_f$/in^3.

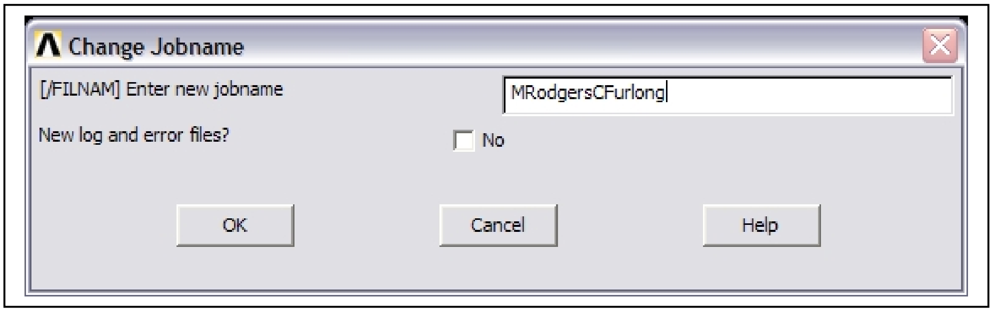
Compare your FEM solutions with corresponding analytical solutions.



Note: ANSYS can import geometry generated in CAD packages, such as Pro/E and SolidWorks through IGES

Setup the File:

1. Start Ansys:
   <Start> → <Programs> → <Ansys 9.0> → <Ansys>
2. Change the file name to your group members name (i.e. MRodgersCFurlong):
   <File> → <Change Jobname>



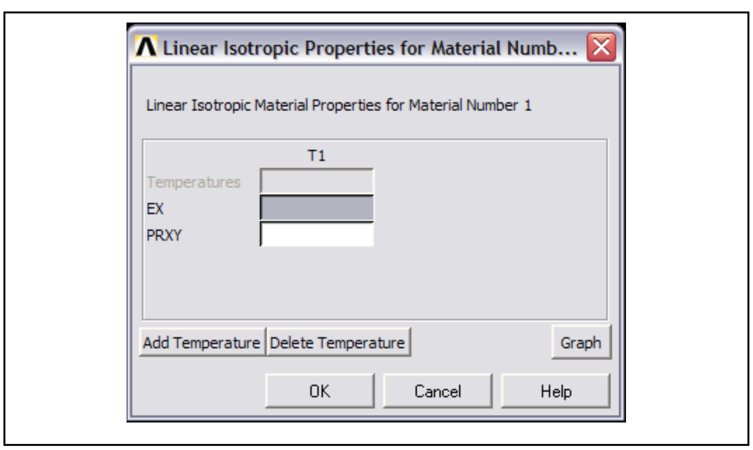
3. Save the file:
   <File> → <Save As> → Choose your directory on the network so you can access your file later.

Setup Element Type and Material Properties

<Preprocessor> → <Element Type> → <Add/Edit/Delete>
<Add> → Structural Mass; Solid; Brick 8-node (#185) → <OK>
Close the Element Types window

<Preprocessor> → <Material Properties> → <Material Models>
In "Define Material Model Behavior" box → <Double-click>
Structural; Linear; Elastic; Isotropic
Set EX (Modulus of Elasticity) & PRXY (Poisson's Ratio)

In "Define Material Model Behavior" box
Structural; Density (Set the Density)

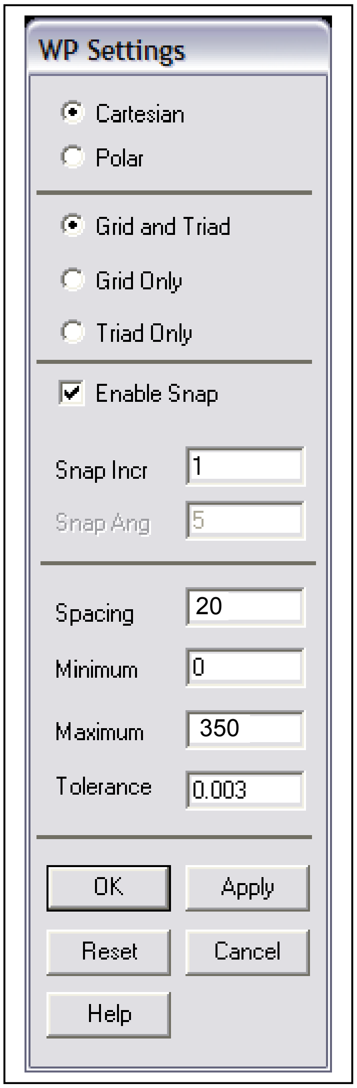


*Note: www.matweb.com is an excellent source of material properties

Define Geometry of the Component:

1. Lets look at our working plane:

   <Workplane> → Make sure <Display Working Plane) is checked
   <Workplane> → <WP Settings> …
   Change the settings so that your rod will fit within the grid.



Note: Ansys does not use units explicitly. Whatever system used in defining your material use the same system here.

<Workplane> → <Offset Workplane by Increments> → Rotate around the negative X-axis until it is perpendicular to your view

2. Lets Make the Cantilevered Beam:

<Preprocessor> → <Modeling> → <Create> → <Keypoints> →
<In Active CS>

In the Box type in the following values:

Keypoint Number	X	Y	Z
1	0	0	0
2	0	350	0
3	7.5	350	0
4	7.5	0	0

<PlotCtrls> → <Pan Zoom Rotate> → Fit

Outline the Area

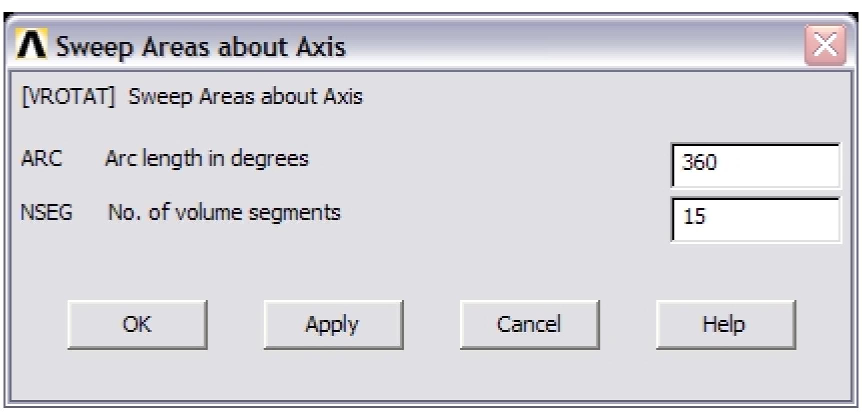
<Preprocessor> → <Modeling> → <Create> → <Lines> →
<Lines> → <Straight Lines>
Pick each point in order: 1,2 2,3 3,4 4,1

Create the Area

<Preprocessor> → <Modeling> → <Create> → <Areas> →
<Arbitrary> → <By Lines>
Select the lines surrounding your area
Click <Ok>

Create the Volume

<Preprocessor> → <Modeling> → <Operate> → <Extrude> →
<Areas> → <About Axis>
Select your Area, Press <Ok>
Select 2 points on the Axis (i.e., along the y-axis), Press <Ok>
In the box that appears type in the following:



You should now see a cylinder on your screen.

Make it one object

<Preprocessor> → <Modeling> → <Operate> → <Booleans> → <Add> → <Volumes>
Select all volumes and click <ok>

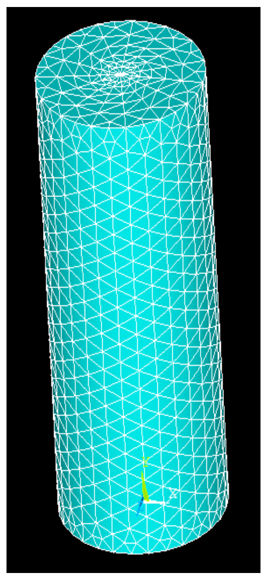
Mesh the Geometric Model:

3. Time to Mesh the beam

<Meshing> → <MeshTool> → Check Smart Size box → Move slider to **5**
Under Size Controls:

Global → <Set>
In the box that appears Type 2.5 into the SIZE box

Press Mesh → Select Volume → Press Ok
Your cylinder should now look like this:



Set Boundary Conditions:

We need to set how the cylinder is allowed to move. For this model we will give this structure a fixed base.

<Preprocessor> → <Loads> → <Define Loads> → <Apply> → <Structural> → <Displacement> → <On Areas>

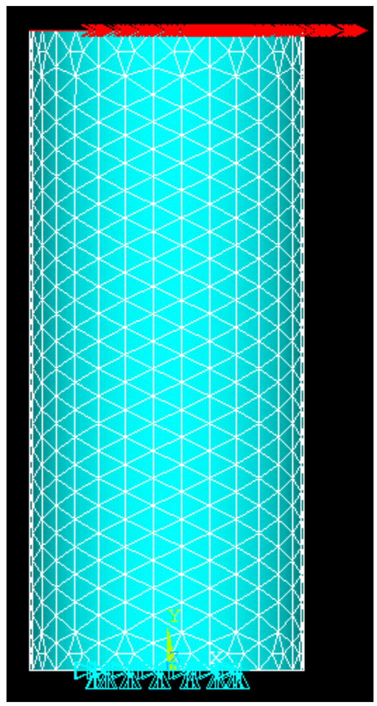
Select the base.
In the box that comes up, select All DOF, Apply as: Constant Value, Displacement Value: 0
This will prevent the base of the beam from moving.

We also have an applied force to the top of this beam.

<Preprocessor> → <Loads> → <Define Loads> → <Apply> → <Structural> <Force/Moment> → <On Nodes>

Using box select, select the top surface.
In the box that comes up, select Fx, Apply as: Constant Value,
Force/Moment Value: 2,000 / # nodes selected: 2,000/91
(distribute the load in the 91 nodes that were selected)

Your beam should now look like this:



Analysis: static

We are going to do a Stress Analysis on this beam.

1. Stress Analysis:

<Solution> → <Analysis Type> → <New Analysis> → Static

Now we run the analysis:

<Solution> → <Solve > → <Current LS>

Time for the results:

<General PostProc> → <Plot Results> → <Contour Plot> →
<Nodal Solution> → Select what you would like to see plotted:
<u>Stress</u> → <u>Von Mises</u> → press <Ok>

There is your deformed shape with stresses in the beam. <u>Verify accuracy of your solution.</u>

MathCAD results:

End Conditions:

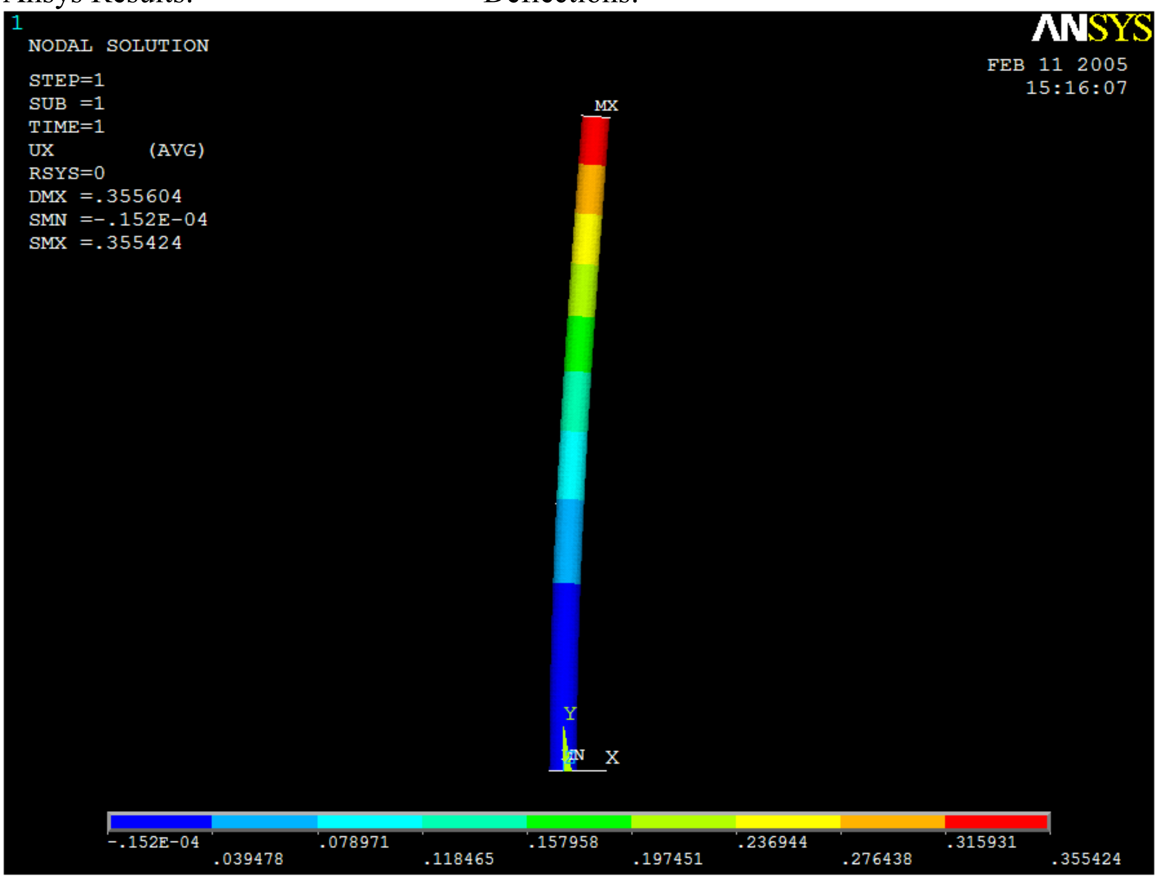
Deflection $y(l) = -0.383\ in$

Slope $\theta(l) = -1.643 \times 10^{-3}\ rad$

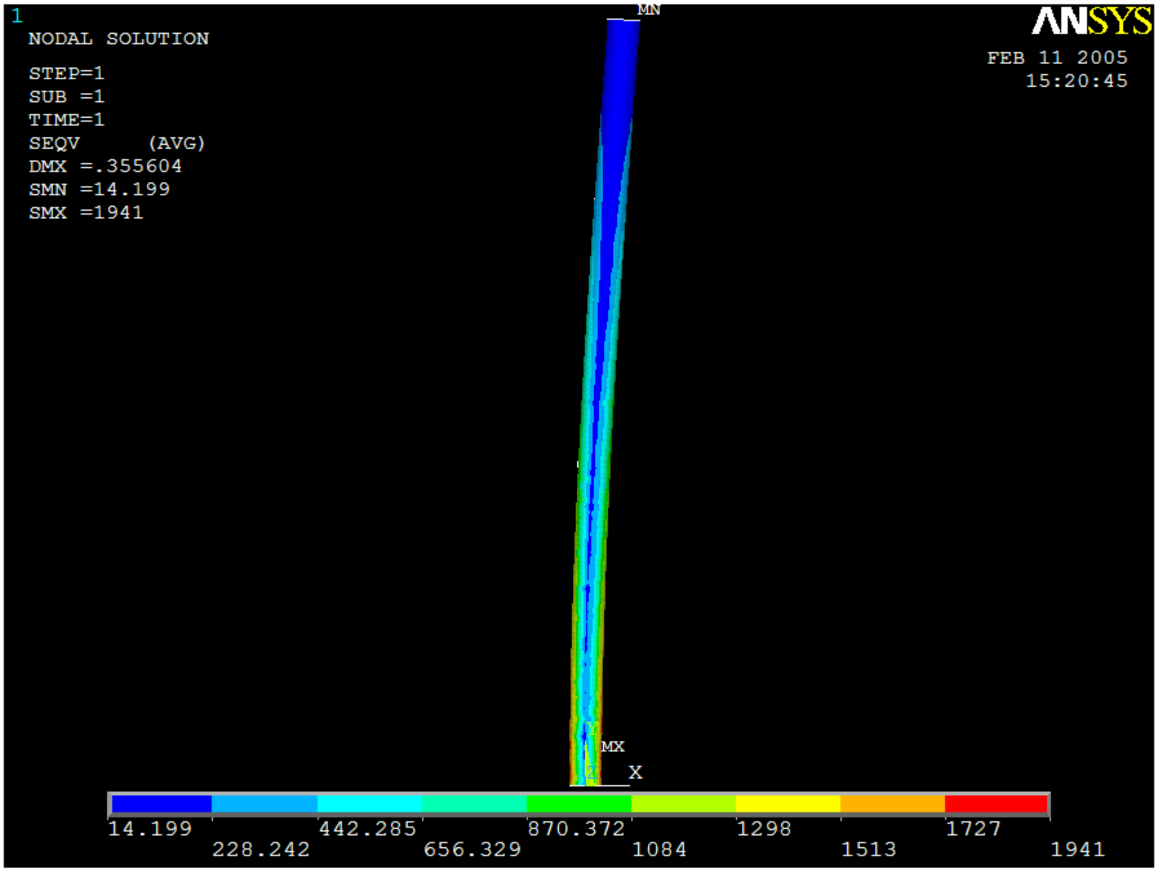
Moment $M(l) = 0.000\ lbf \cdot in$

Force $V(l) = 0.000\ lbf$

Ansys Results: Deflections:



Von Mises



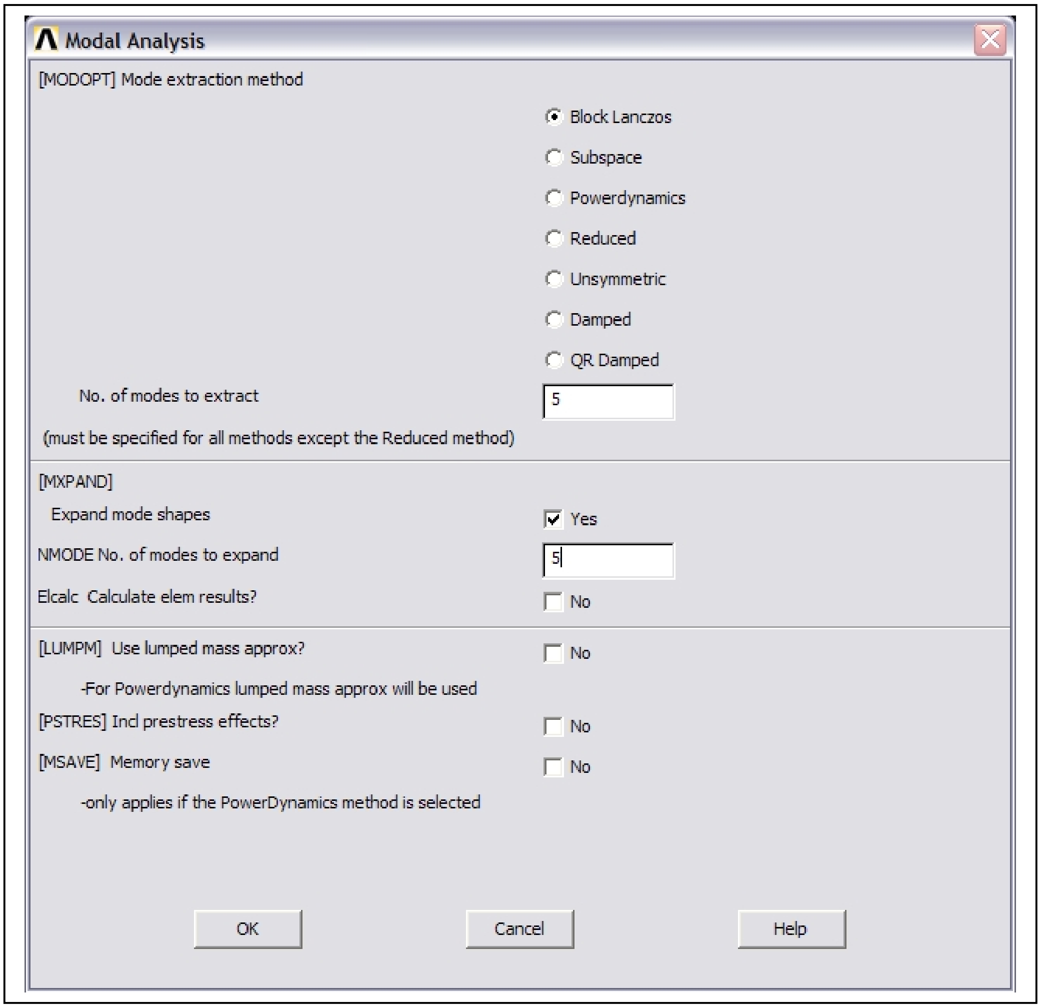
Analysis: modal

2. Specify Modal Analysis

<Solution> → <Analysis Type> → <New Analysis> → Modal

Then we need to tell Ansys how many of the infinite number of modes to find.

<Solution> → <Analysis Type> → <Analysis Options> → see box:



Press ok at the next window

Now we run the analysis:

<Solution> → <Solve > → <Current LS>

Time for the results:

<General PostProc> → <Results Summary> → And there are your results.

You can animate a mode by going to
<General PostProc> → <Read Results> → <By Pick>
Choose one of the points, press read.
Close the window
<PlotCtrls> → <Animate> → <Mode Shape> → press ok & watch the animation

Homework Problems:

1) Problem 16-7e,
2) Problem 16-10e.
3) Apply FEM to your final design project.

*   *   *