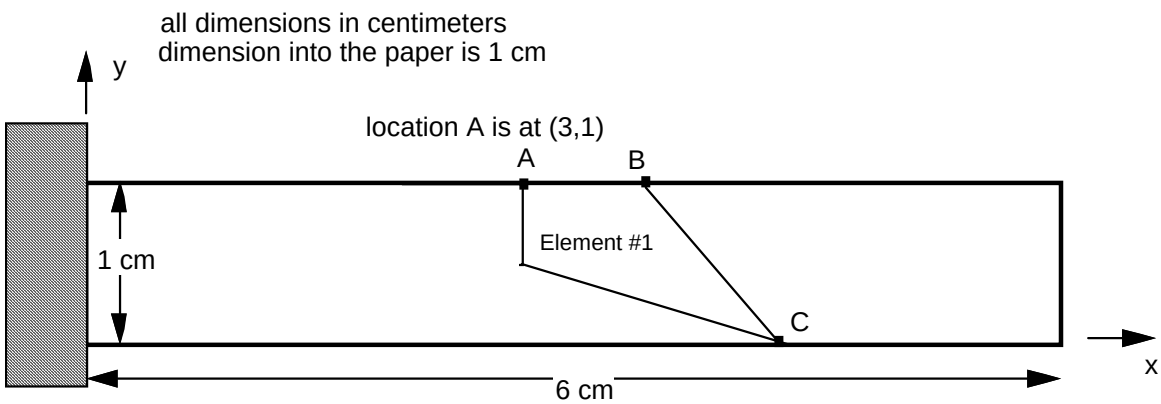
ME309 **Homework #3**

OBJECTIVES:
• Use of shape functions for an isoparametric element.
• Exercise theory and commercial finite element code to solve problems in vibration.
• Solution to an open-ended problem using a commercial FE code.

NOTES:
1) Problem I.1 does not require a FEM solver (i.e. ANSYS).
2) Problems II.1 and III.1 do require a FEM solver.
3) Review Tutorial 3 prior to tackling II.1

1.) Extrapolation of stress values to nodes.

1.1) (15%) Use the procedure described in class for plane isoparametric elements to find stress σ_{xx} at Point C in the beam below predicted according to Element 2. The load in each case is a 100N/cm distributed load acting vertically downward. Find the stress (based on beam theory) at the quadrature points to begin. Comment on the difference between the value predicted with element 2, and the predicted at directly at C from beam theory.



Element #1 is such that B is at (4,1) and C is at (5.25,0)

Figure 1

2.) Natural Frequency Calculation.

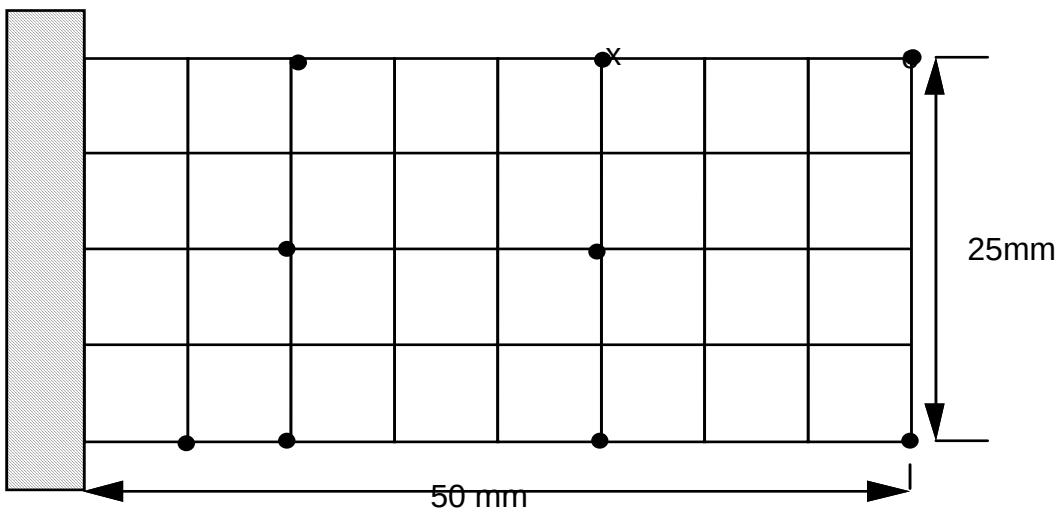
2.1) (30%) For the structure shown in Figure 2 below calculate the first three natural frequencies. Use SHELL63 Elements.

(A) In one case, use 120 dynamic degrees of freedom (not including the wall points) for motion in-and-out of the paper.

(B) In another case, use reduction (matrix condensation) to specify the 9 Master degrees of freedom (in-and-out of the paper direction as marked by the DOTS).

(C) For the final (and third case), let the software automatically select nine master degrees of freedom**. Also include a sketch that shows where these nine DOF are located.

(D) Briefly comment on the results.



Density = 7850 kg/m^3, E=207 GPa, Poisson's Ratio=0.30
Out-of-paper dimension is 1 mm

Figure 2

Natural Frequency Table (Hz)

# dynamic degrees of freedom	**Mode 1**	**Mode 2**	**Mode 3**
120			
9 (user specified at DOTS)			
9 (automatically selected)			

**You can tell ANSYS to automatically select a specified number of Master degrees of freedom. It makes this selection based on ranking the ratio of mass/stiffness for every degree of freedom. The largest ratios are kept as the masters degrees of freedom. See pages 6-7 of this document, and the ANSYS Documentation, Structural Analysis Guide, Section 3.14 “Matrix Reduction” for more information.

2.2) (15%) The stiffness, consistent mass, and optimally lumped mass matrices for the unsupported, uniform, *three-noded quadratic bar element* (shown below in Figure 3) are:

$$\left(\frac{AE}{3L}\right)\begin{bmatrix} 7 & -8 & 1 \\ -8 & 16 & -8 \\ 1 & -8 & 7 \end{bmatrix};\ \left(\frac{\rho AL}{30}\right)\begin{bmatrix} 4 & 2 & -1 \\ 2 & 16 & 2 \\ -1 & 2 & 4 \end{bmatrix};\ \left(\frac{\rho AL}{6}\right)\begin{bmatrix} 1 & 0 & 0 \\ 0 & 4 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

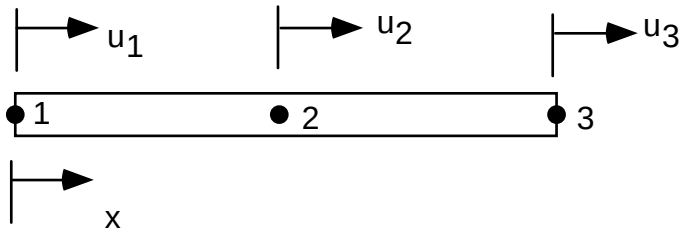


Figure 3

(a) For axial vibration, determine the three natural frequencies and mode shapes using the consistent mass matrix.

(b) What is the physical significance of the lowest frequency and mode shape for part (a)?

(c) The exact frequencies of an unsupported continuous bar of length L and mass density ρ are $\omega=(n\pi/L)(E/\rho)^{0.5}$, n = 0,1,2,... What are the percentage errors of the frequencies computed in part (a)?

3.) Modeling for Structural Evaluation.

3.1) (40%) In this problem you go through all of the basic steps of structural FE modeling. This includes selecting the model appropriate for the problem, choosing your elements, defining the boundary conditions, and interpreting your results.

Figure 4 on the following page illustrates a fatigue specimen called a “modified coach-peel” specimen. It consists of a single resistance spot weld (RSW) of diameter D of 0.25", made in 0.057" thick sheet steel. It is tested on a MTS machine with grips that "grab" the specimen 1" from each end (as shown by the cross-hatched regions in the figure). Consider the case of testing from 0 to tensile load, with the load direction as shown in the figure. Determine the *stiffness* of the specimen using the finite element method. Before actually doing any modeling, think about:

1. what stiffness IS and the extent of model required to determine it for this structure.
2. appropriate boundary conditions (e.g., forces, displacements).

WHAT YOU MUST DO:

A. Describe of two alternative models for the problem, including:

- Element types you will use (complete models using only bar or beam elements are probably too simplified).
- Needed input properties (e.g., material properties, thickness, etc).
- Your boundary conditions.
- The mesh density you believe appropriate (SKETCH this).
- Pros and cons of the given alternative approaches.

B. Choose one of the modeling approaches that you described in (A), build a model, and determine the value of stiffness of the specimen. Clearly describe your definition of stiffness. Various approaches will be compared with experimental data in class.

HINTS:

(1) Feel free to ask questions when tackling this problem, especially if it is becoming an infinite time sink.
(2) See pages 8-10 of this document for notes on gluing.

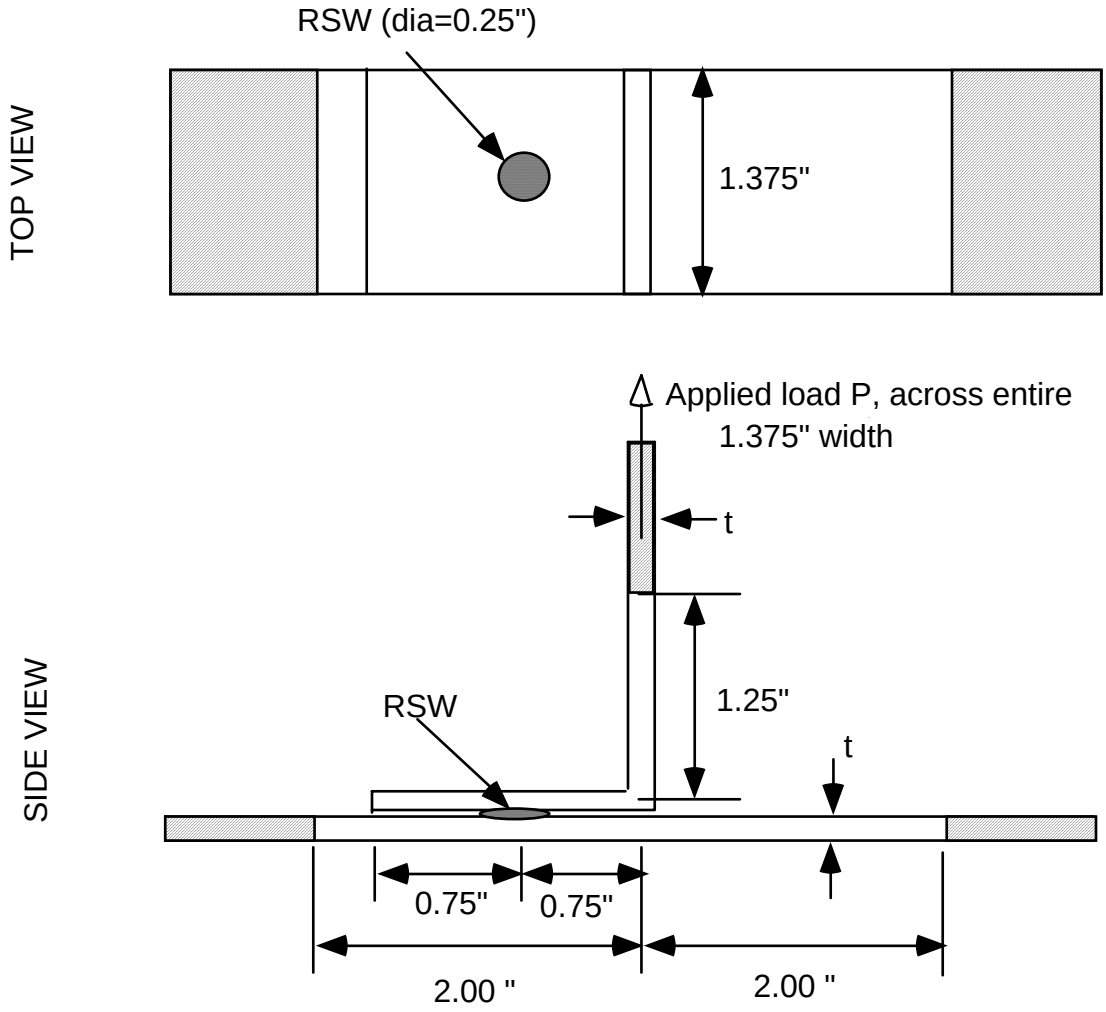
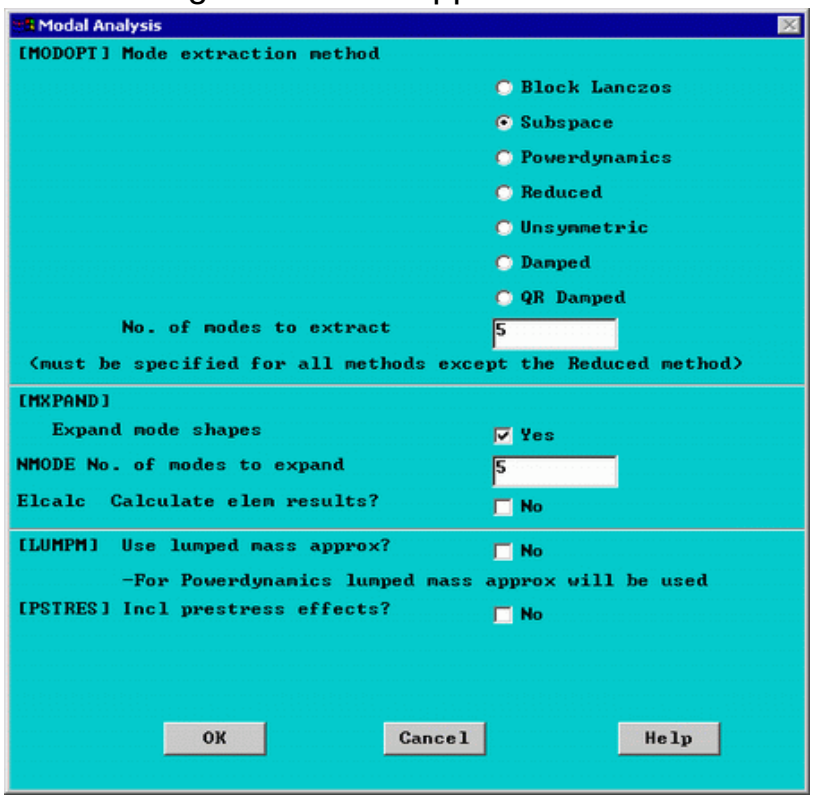

RSW (dia=0.25")
TOP VIEW
1.375"
Applied load P, across entire
1.375" width
t
SIDE VIEW
RSW
1.25"
t
0.75"
0.75"
2.00 "
2.00 "

Figure 4

How to Select Master Degrees of Freedom.

- Define Analysis Type Solution > Analysis Type > New Analysis > Modal
- Set options for analysis type: Select: Solution > Analysis Type > Analysis Options.

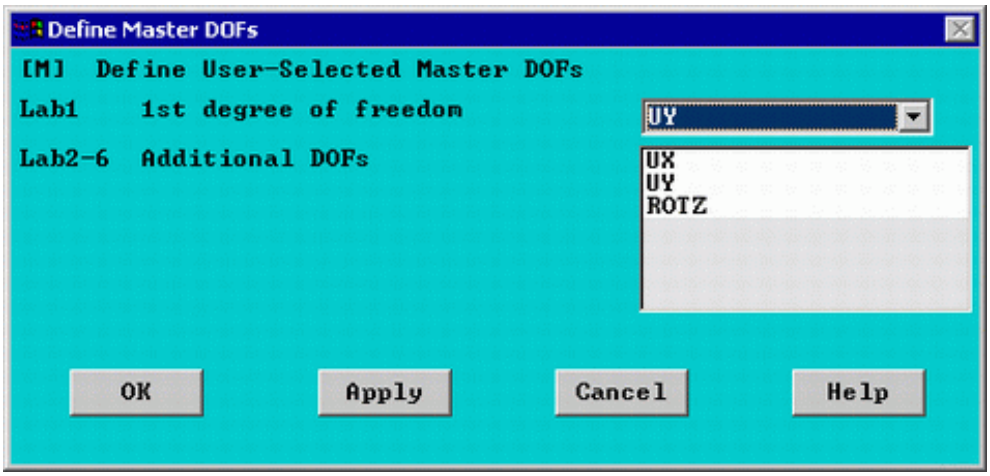
The following window will appear…



- As shown, select the Reduction method and enter the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter in the 'No. of modes to expand'
- Click 'OK'

• Select **Solution > Master DOF > User Selected > Define**
When prompted, select the required nodes.

The following window will appear…



•Select the required degree of freedom
(Decide based on the question).

Note: The default mode extraction method chosen is the **Reduced Method**. This is the fastest method as it reduces the system matrices to only consider the Master Degrees of Freedom (see below). The **Subspace Method** extracts modes for all DOF's. It is therefore more exact but, it also takes longer to compute (especially when the complex geometries).

For problem #3, a “glue” command may be useful. The commands “LGLUE”, “AGLUE” “VGLUE” connect lines, areas and volumes respectively. This can be very useful if you want to create individual entities and connect them in your model. See below for details…

LGLUE, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9*

Generates new lines by “gluing” lines.

PREP7:Booleans

Mp Me St DY LP Th E3 E2 FL PP ED

NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9

Numbers of the lines to be glued. If *NL1* = ALL, all selected lines will be glued (*NL2* to *NL9* will be ignored). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Generates new lines by “gluing” input lines. The glue operation redefines the input lines so that they share keypoints at their common ends. The new lines encompass the same geometry as the original lines. This operation is only valid if the intersections of the input lines are keypoints at the ends of those lines. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu >Preprocessor >Operate >Glue >Lines

AGLUE, *NA1, NA2, NA3, NA4, NA5, NA6, NA7, NA8, NA9*

Generates new areas by "gluing" areas.

PREP7:Booleans

Mp Me St DY LP Th E3 E2 FL PP ED

NA1, NA2, NA3, NA4, NA5, NA6, NA7, NA8, NA9

Numbers of the areas to be glued. If *NA1* = ALL, all selected areas will be glued (*NA2* to *NA9* will be ignored). If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Generates new areas by "gluing" input areas. The glue operation redefines the input areas so that they share lines along their common boundaries. The new areas encompass the same geometry as the original areas. This operation is only valid if the intersection of the input areas are lines along the boundaries of those areas. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to new entities generated.

Menu Paths

Main Menu >Preprocessor >Operate >Glue >Areas

VGLUE, *NV1, NV2, NV3, NV4, NV5, NV6, NV7, NV8, NV9*

Generates new volumes by "gluing" volumes.

PREP7:Booleans

Mp Me St DY LP Th E3 -- FL PP ED

NV1, NV2, NV3, NV4, NV5, NV6, NV7, NV8, NV9

Numbers of the volumes to be glued. If *NV1* = ALL, all selected volumes will be glued (*NV2* to *NV9* will be ignored). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Generates new volumes by "gluing" input volumes. The glue operation redefines the input volumes so that they share areas along their common boundaries. The new volumes encompass the same geometry as the original volumes. This operation is only valid if the intersections of the input volumes are areas along the boundaries of those volumes. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu >Preprocessor >Operate >Glue >Volumes