Lab 3 ME/ECE 550

Modal Analysis of a Folded Beam Suspension

1 Introduction

Folded beam suspensions are useful because they allow for linear motion of a shuttle with little parasitic motion. They are used in a variety of MEMS applications; including comb drives and inertial sensors. Any mass-spring system responds to vibrational excitation. And each system has certain natural frequencies.
Resonance is when the frequency of the vibrational excitation matches a natural frequency. Resonance can cause larger amplitudes in oscillations and, in some cases, can destabilize the system. It is important to understand the natural frequency when designing a system (mechanical or electrical) in order to avoid any unwanted response.

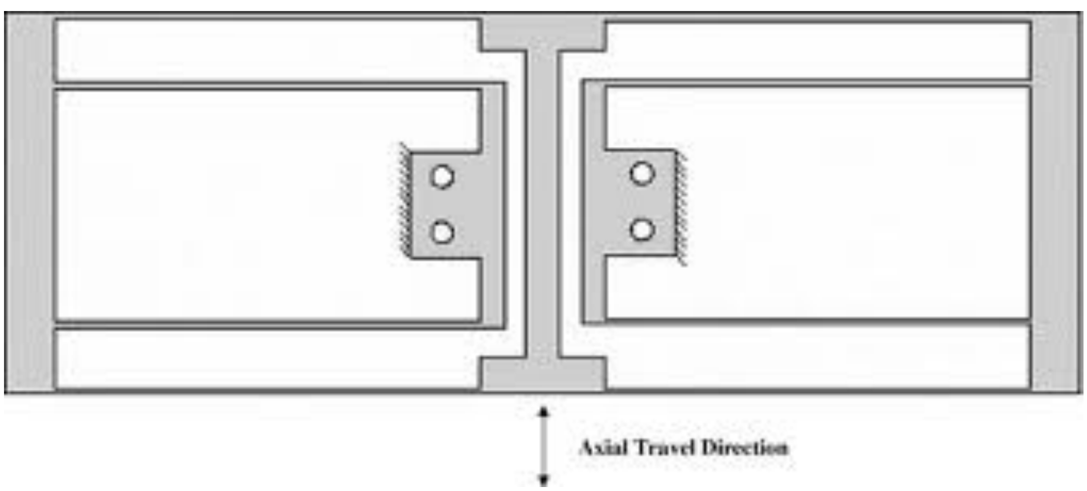
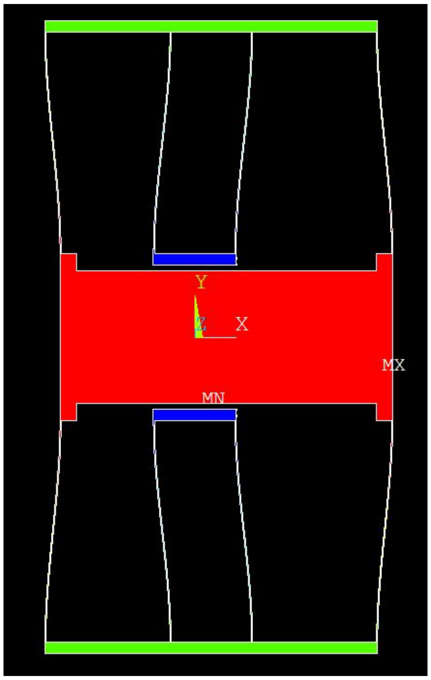


Figure 1. Folded Beam Suspension
Cannon, Bennion R., "A compliant end-effector for microscribing"

2 The Problem

Your assignment is to determine two of the natural frequencies of a folded beam suspension similar to the one pictured in Fig. 1. The first natural frequency you will be looking for is if the center mass is oscillating. A picture of this mode is shown in Fig. 2a. The second natural frequency you will calculate is when the thicker external beams are oscillating (Fig. 2b). You will use the lumped-mass assumption to calculate the natural frequencies in MATLAB, and you will also model the system in ANSYS and perform a modal analysis.



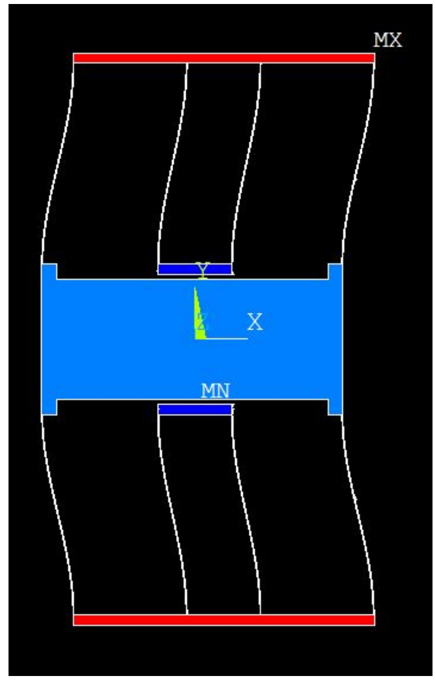


Fig 2a: Case 1
First Mode to calculate. Center shuttle is the oscillating mass

Fig 2a: Case 2
Second Mode to calculate. Top and bottom stiff beams are oscillating.

3 Procedure

You will write an ANSYS batch file to run your analysis from. The basic form you are encouraged to follow, along with some hints, are included below. Once your you have solved the problem in ANSYS, you will then calculate the natural frequency of the system using lumped-mass assumption. The natural frequency equation is of the form:

$$f_n = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$$

It will be your responsibility to determine the appropriate spring constant and mass for each case.
HINT: Remember to keep units consistent. Think about what units the natural frequency will be in if you use the micron-based unit set.

ANSYS Batch File Help
The following covers the general steps you must take to complete your batch file. You should use the Help file in ANSYS to determine what commands do and how to use them. Also, it isn't a bad idea to use the GUI and then pull the command you need from the log file. In creating and debugging your batch file you will find the command '/eof' to be extremely valuable. This command will tell ANSYS that it has reached the end of the batch file, meaning that anything after the '/eof' will be ignored. Much of the value of this lab is in learning how to break down

problems to make them solvable using finite element analysis software.
/BEGIN

Useful COMMANDS: =, *SET, FINISH

In the begin processor you will define all the variables you expect to use. A list of dimensions and properties is given later.

Also define these parameters: numModes = 10, freqLo = 0, freqHi = 0.1

/Prep7

Useful COMMANDS, ET, MP, /INPUT, ASBA, VEXT, TYPE, MSHAPE, MSHKEY, ESIZE, VMESH, NSEL, D, F, ALLSEL, AL, L

*Define the element type (Use SOLID187).

*Assign material properties (MP command)

HINT: remember to include a density

*Define keypoints and lines

-The keypoints are provided in a separate text file (ME550_Lab3FEA_Keypoints.txt)

-Most of the lines are also included in a separate text file (ME550_Lab3FEA_partial_Lines.txt)

-Use the /INPUT command in your batch file to include the keypoints and the lines. Just as your batch file is called by ANSYS using the /INPUT command, you can call additional files using /INPUT. You can nest quite a few batch files.

-Edit the line text file to complete the lines you need.

*Define areas

-A separate text file is included to get you started on defining areas (ME550_Lab3FEA_partial_Areas.txt)

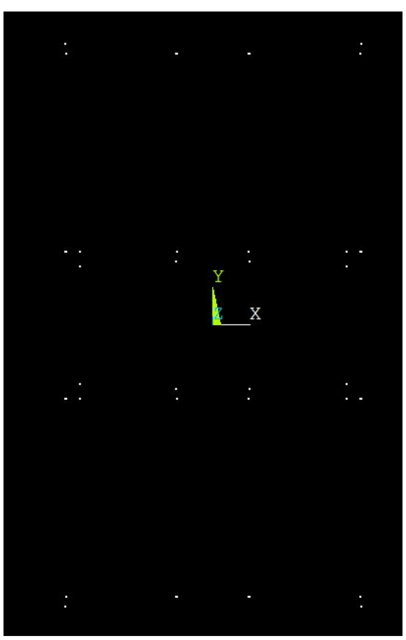


Fig 3a: Keypoints

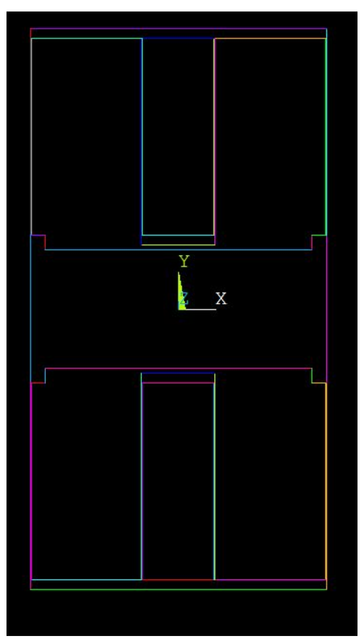


Fig 3a: Lines

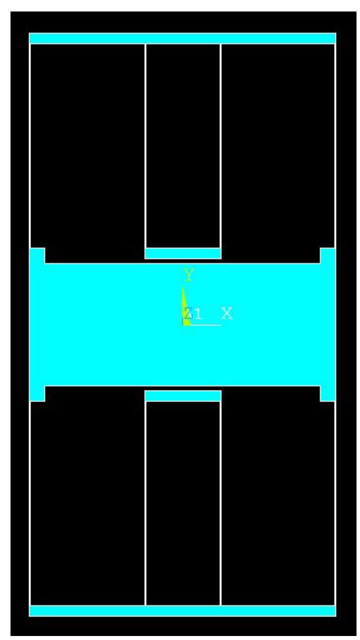
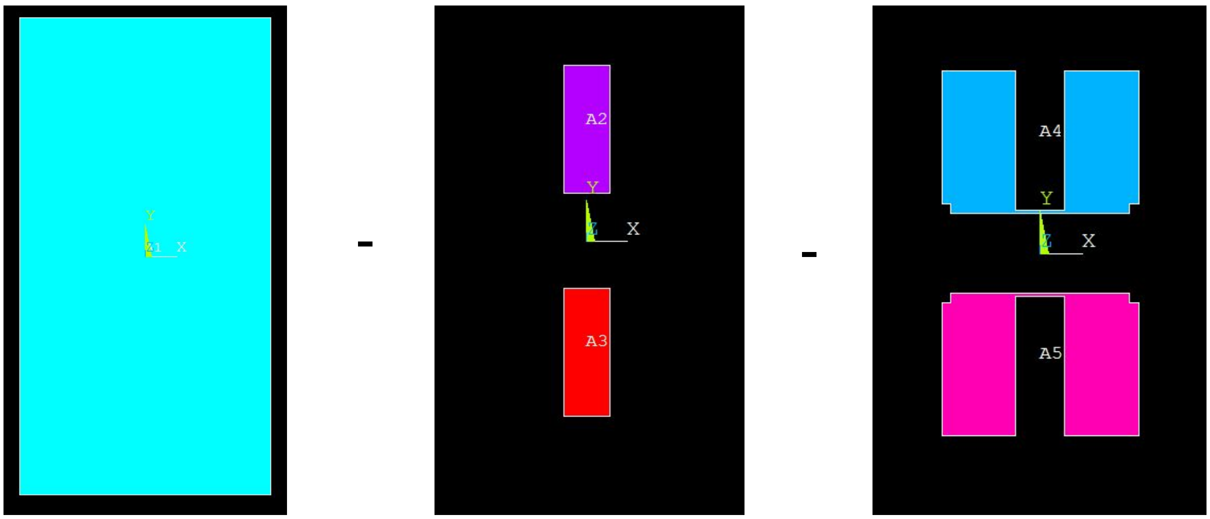


Fig 3a: Final Area

-Recommended that you complete the text file and use the /INPUT command

*Boolean operation to subtract some areas from others. (ASBA)

-Subtract smaller areas from the larger one



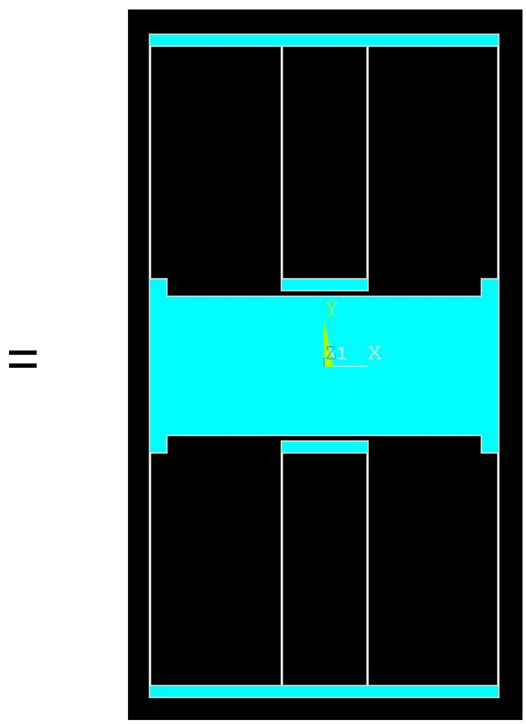


Fig. 4: Boolean Operation to obtain desired shapes

*Extrude the resulting area into a volume. (VEXT)

*Mesh the volume using the VMESH

-Use: MSHAPE, MSHKEY, ESIZE to define meshing characteristics

*Define loads

-Use "D" to define DOF constraints on nodes.

-Any place considered to be “ground” should have all DOF constraints set to zero.
-Ground is along the surface that faces the largest area. (See Fig. 5)

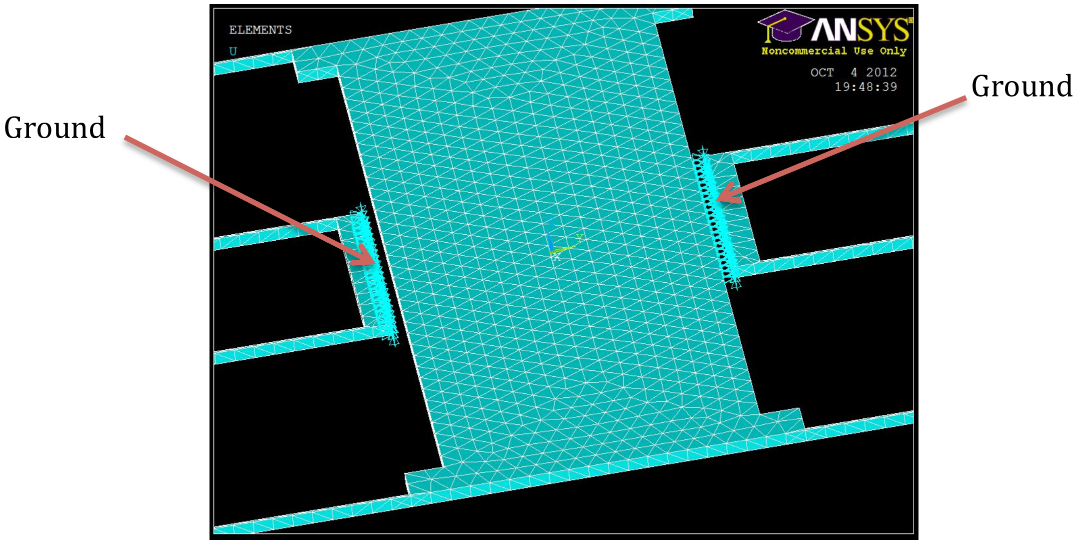


Fig. 5: Ground Points
This indicates where ground is in the system

-Use “F” to apply a force in the x-direction on the left side of the system. (see Fig. 6).
HINT: Use NSEL, and select nodes by location

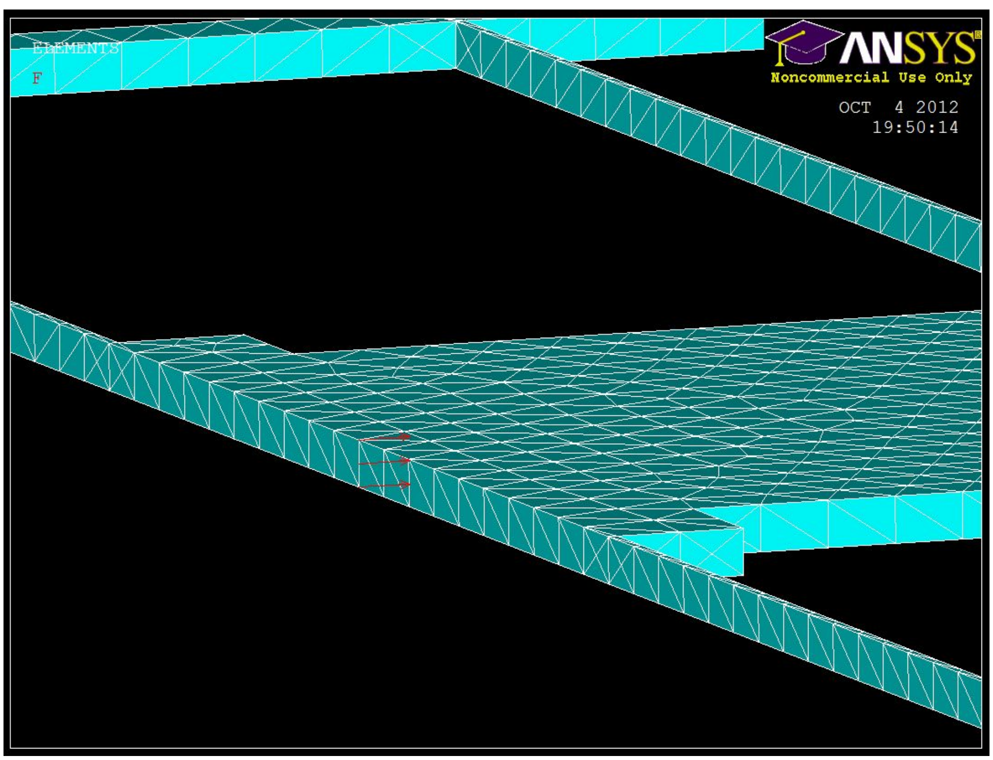
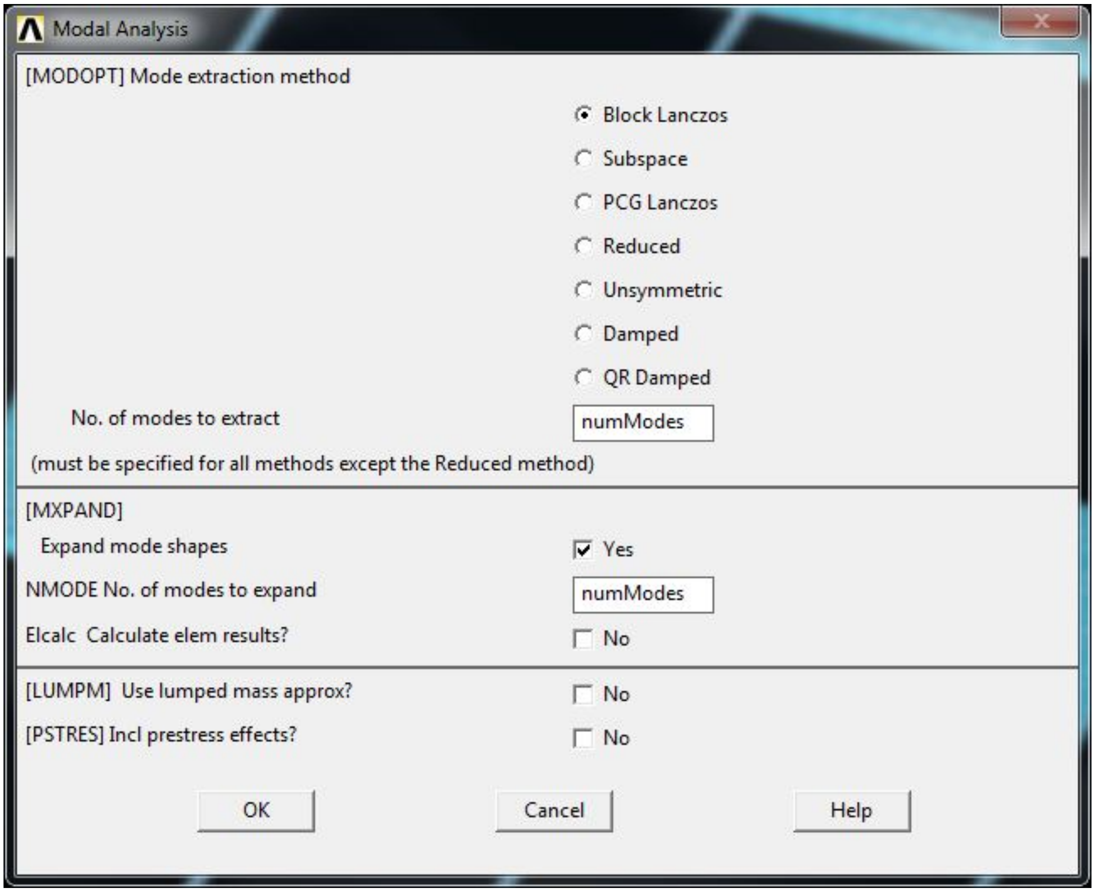


Fig 6.
Shows where the force load should be applied

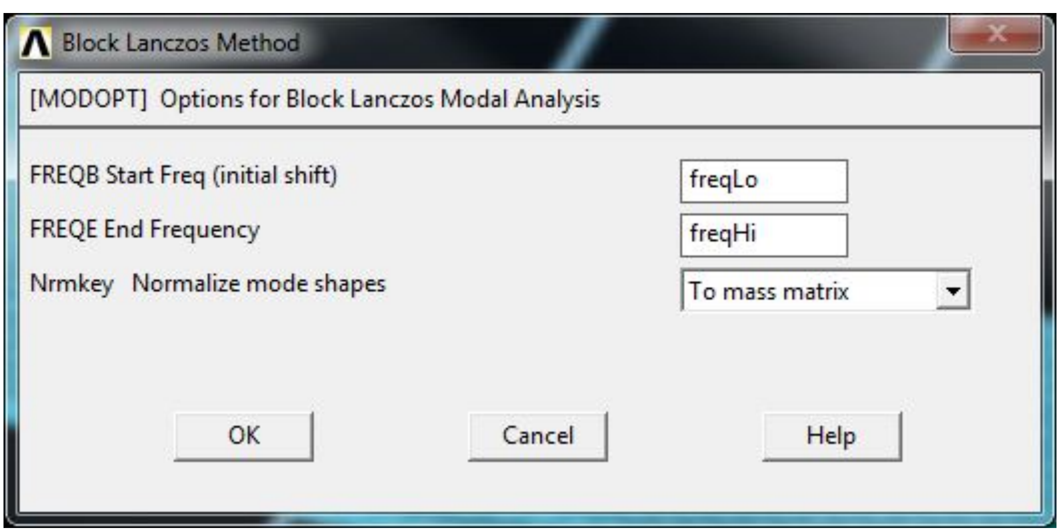
The force load is necessary for ANSYS to perform the modal analysis

/SOL

Once you reach this stage you should place an /eof and run the file. Then use the GUI to set up the modal solution. Once you complete these steps, copy the commands from the log file.
Solution >> Analysis Type >> New Analysis >> Modal
Solution >> Analysis Type >> Analysis Options



Click "OK"



Now copy the commands that you generated in the log file and place them in your batch file.
    Then solve the analysis.

/Post1
    Finally, review the results
    General Postproc >> Results Viewer
    In the Results Viewer:
    Nodal Solution >> DOF solution >> Displacement Vector Sum

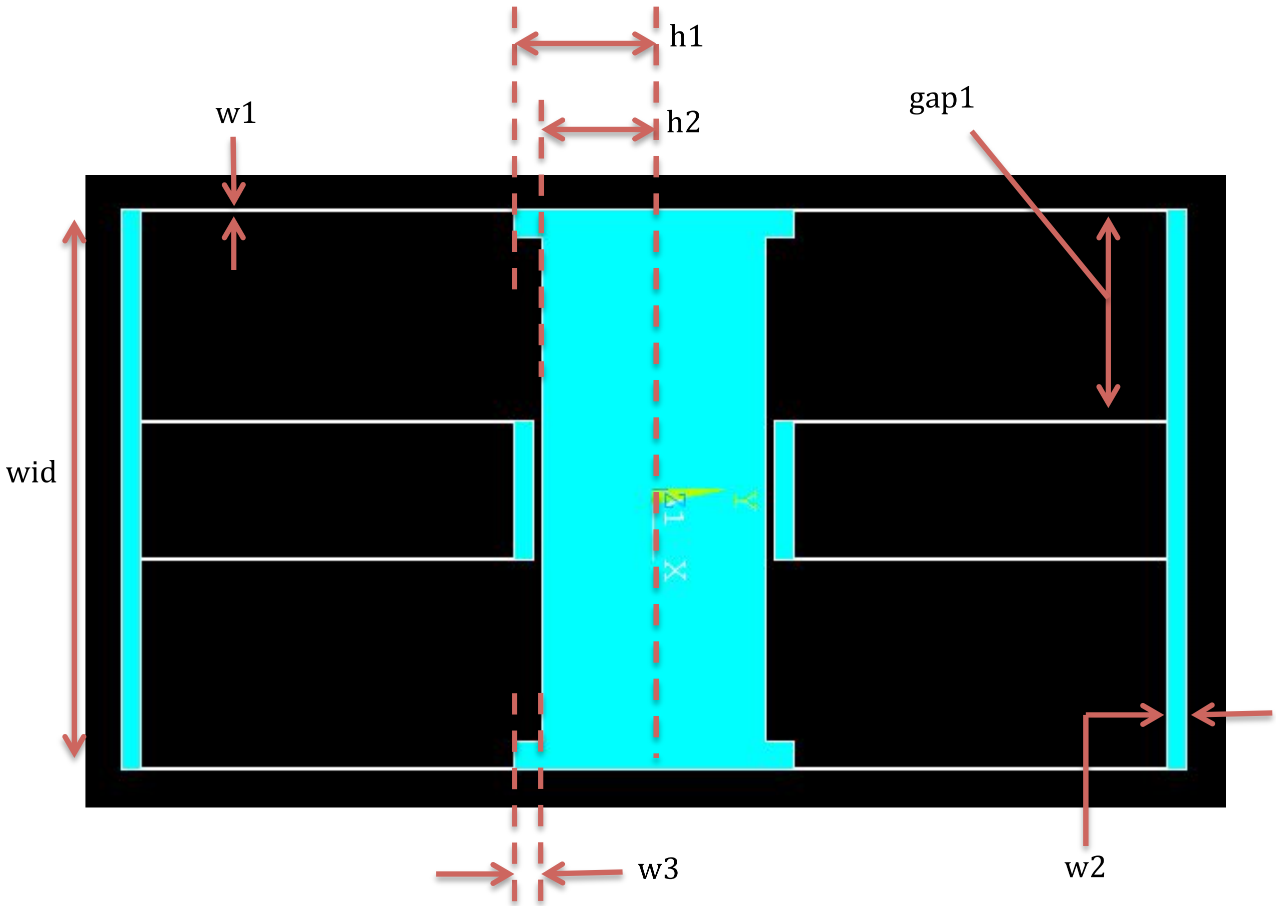
Now use the slider bar to scan through the modes that ANSYS solved for. Find the modes that correspond to the cases you have been asked to solve and record the frequency for those modes.
SAVE a picture of the Case 1 mode shape to include in your Memo.

Lumped-Mass Calculations
Use MATLAB to set up the natural frequency equations for both cases. Both cases can be included in the same m-file.

4 Deliverables

Turn in a professional memo detailing and comparing the natural frequency results of the ANSYS modal analysis and the lumped-mass equations. Discussion in the memo can include, but is not limited to, the uses of folded spring systems in MEMS and when simplifying equations can be used in place of finite element analysis.

Attached to the memo you should include:
-ANSYS batch file (main file only, not the keypoint, line, or area text files)
    -Batch file should be commented to indicate what each step is doing
-MATLAB m-file
-Picture of your Case 1 ANSYS solution



The following are parameters that are needed to make the keypoints input work and to calculate the lumped-mass natural frequency			
Parameters	**Symbol**	**Value**	**Units**
Beam Length	l	2000	μm
Beam Width	w1	10	μm
Width of stiffer segments	w2	100	μm
Width of flanges	w3	w2 + 50	μm
Structure width	wid	3000	μm
Half Structure Width	wid2	wid/2	μm
	h2	600	μm
	h1	h2 + w3	μm
Thickness	t	50	μm
	gap1	(3/4)*(wid2-w1)	μm
Density	p	2.65*10^-3	ng/(μm^3)
Young's Modulus	E	165000	MPa
Poisson's Ratio	nu	0.33	
# of Modes	numModes	10	
Lower frequency bound	freqLo	0	MHz
Upper frequency bound	freqHi	0.1	MHz