Lab 2 ME 550

Analysis of a bistable MEMS device using beam3 elements in Ansys

1. Open ansys				
ANSIS University Advanced Unity Menu (Elisable)	and the			
	Drive Det			
AVEYS TOODER			\$	
Add he have 8				
Preisences			ANSYS	
Di Beard Type 27/14/ 2014			14:26:33	
C Platerial Prage Sections			3	
D Posteme D Create D Create			8	
20 20 VED Size Flave			3	
,70 contines ,70 contines m/Eatler			6	
P to between the				
B KP at center E Hard PF on Inc	L.	4	9	
E Hard PE on area El Lexce	-		<u>4</u>	
E Arman			9	
E flerends E Contact Por			-	
El Poline Hockes El Circuit				
Bacchrack Coll 8) Freesduczy				
Si Mover / Modify Si Cepy			8	
E Reflect E Check Geom				
© Detet # © Cycle: Sector © Cycle: Sector			10	
D Update Genn (0 Meshing			<i>Q</i> •	
G Checking Chie G Pambering/Chie				
Poli, a mena litera ar MARIS Commani (MIRY)	ories Index Index	1		
2 Change working	Directory and	Jobname		•
File>> Change Γ	Directory	oonunie		
File>> Change L				
File>> Change J	Joname			
3. From the Ansys	Main Menu>>	Preferences>	>Structural	
4. Preprocessor>>E	lement			
Type>>add//Edit	/Delete>>add>	>>Beam>>2D	elastic 3>>	•OK>>Close
Library of Element Types				X
ary of Element Types		Structural Mass	🔨 2D elastic	3 🔨
		Link	plastic :	23
		Pipe	3D finite st	rain 📃
		Solid	2 node	188
		Shel Constraint	3 node	189 👱
		Hyperelastic	2D elastic	3
ent type reference number		1	,	
ок	Apply	Cancel		Help

5. Preprocessor>>Real-Constants>>Add/Edit/Delete>>Add>>OK
 Enter Parameters as shown below
 (The second Real Constant Set No needs to be 2 not 1)

Real Constants for BEAM3	X	Real Constants for BEAM3	
Element Type Reference No. 1		Element Type Reference No. 1	
Real Constant Set No.	1	Real Constant Set No.	1
Cross-sectional area AREA	3	Cross-sectional area AREA	20
Area moment of inertia IZZ	.5625	Area moment of inertia IZZ	167
Total beam height HEIGHT	1.5	Total beam height HEIGHT	10
Shear deflection constant SHEARZ	1.2	Shear deflection constant SHEARZ	1.2
Initial strain ISTRN	0	Initial strain ISTRN	0
Added mass/unit length ADDMAS	0	Added mass/unit length ADDMAS	0
OK Apply Cancel	Help	OK Apply Cancel	Help

6. Preprocessor>>Material Props>> Material Models >>Structural>>Linear>>Elastic>>Isotropic

Enter Parameters as shown below (Remember Ex needs to be in



∧ Define Material Model Behavior			
Material Edit Favorite Help			
Material Models Defined	Material Models Available		
Material Model Number 1	Structural Structural Elastic Structural Solution Control Solution S	× 	Add Temperature Delete Temperature Graph OK Cancel Help

- 7. Preprocessor>>Modeling>> Create>>Keypoints>>In Active CS Enter 4 key points. (Coordinates come out of Lab1) Point 1 is 0,0,0 Find the other 3
- Preprocessor>>Modeling>> Create>>Lines>>Lines>>Straight Line Line 1 between point 1 and 2 Line 2 between point 2 and 3



9. Preprocessor>>Meshing>> Mesh Attributes>>Picked Lines

Pick line 1 and 3 and assign the correct attributes (default values) Pick line 2 and change "Real constant set number" to 2 >> OK

10.Preprocessor>>Meshing>> Size Cntrls>>ManualSize>>Lines>>All Lines

▲ Element Sizes on Picked Lines	
[LESIZE] Element sizes on picked lines	
SIZE Element edge length	
NDIV No. of element divisions	50
(NDIV is used only if SIZE is blank or zero)	
KYNDIV SIZE,NDIV can be changed	🔽 Yes
SPACE Spacing ratio	
ANG5IZ Division arc (degrees)	
(use ANGSIZ only if number of divisions (NDIV) and	
element edge length (SIZE) are blank or zero)	
Clear attached areas and volumes	∏ No
OK Apply	Cancel Help

Specify 50 for the "No of element divisions

- 11.Preprocessor >> Meshing >> Mesh >> Lines >> Pick All
- 12. Solution >> Analysis Type >> New Analysis >> Static >> OK
- 13. Solution >> Analysis Type >> Sol'n Controls Change Small Displacement to Large Displacement Static Number of substeps 15 Max no. of substeps 30 Min no. of substeps 5

Frequency: Write every substep

14. Solution >> Define Loads >> Apply >> Structural >> Displacement >> On Keypoints

With mouse, select key point 1 and highlight "ALL DOF" and OK Repeat for point 4 but only highlight UX and ROTZ and OK

Apply U,ROT on KPs	
[DK] Apply Displacements (U,ROT) on Keypoints	
Lab2 DOFs to be constrained	All DOF UX UY ROTZ
Apply as	Constant value
If Constant value then:	
VALUE Displacement value	0
KEXPND Expand disp to nodes?	∫ No
OK Apply	Cancel Help

Repeat for point 4 but only highlight UY and Specify a "VALUE Displacement value" of "-15"

- 15. Solution >> Solve >> Current LS
- 16. General Postproc
- 17. Once the solution is done, ouput the Reaction Forces to verify Bistability
- 18.List>> Results>> Reaction Solution...

▲ List Reaction Solution	
[PRRSOL] List Reaction Solution	
Lab Item to be listed	All items Struct force FX FZ All struc forc F Struct moment MX MY MZ All struc mome M
	FY
OK Apply C	Cancel Help

19.Compare with force results from Lab 1 (realize that the reaction forces given by Ansys represent the force response of the mechanism at that position, not the max force)

20.General Postproc >> Plot Results >> Deformed Shape

