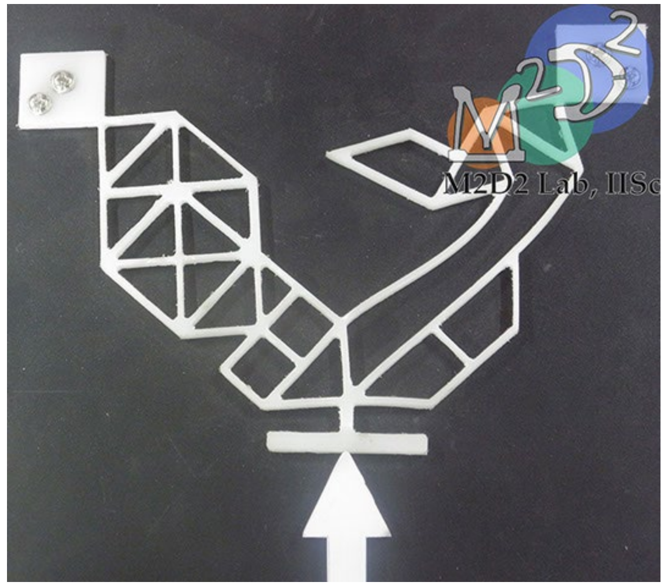
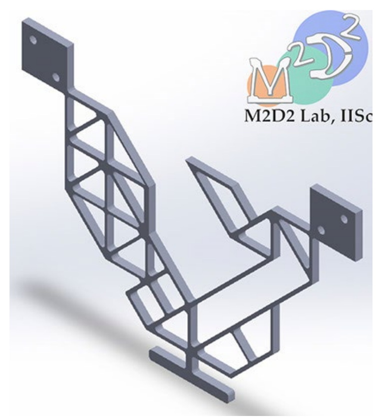
ME 254: Compliant Mechanisms		
Assigned: Jan. 9, 2024	Homework 1: Solution for CM9	Due: Jan. 16, 2024

Question 1 (10 points)

Choose any compliant mechanism from the M2D2 collection available at https://mecheng.iisc.ac.in/m2d2/CMcollection.



CM9 is chosen for outlining what is expected in this homework. CM9 is a Displacement-amplifying Compliant Mechanism (DaCM). It is a result from a topology optimization algorithm. It is a compact mechanism fixed at two corners. An upward vertical force is applied at the bottom in the middle due to which a point at the top in the middle moves downwards, amplified. It is an inverter mechanism in the sense that the output Degree of Freedom (DoF) moves opposite to the direction of the input force.

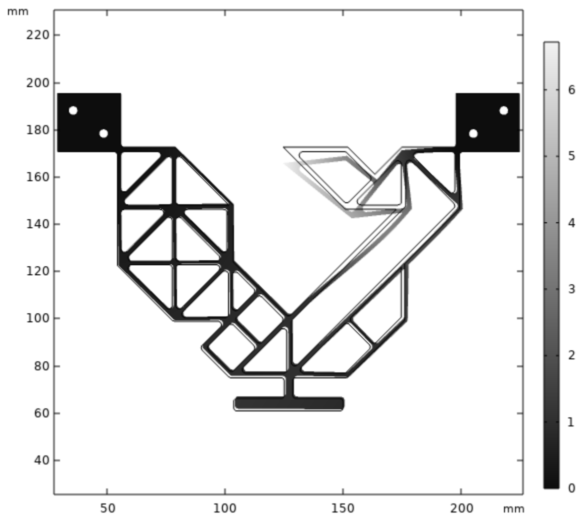


The CAD file was downloaded into COMSOL software. Its dimensions were noted by also consulting the prototype in the M2D2 laboratory. The mechanism is shown in a perspective view. The two squares at the two corners at the top are to be fixed.

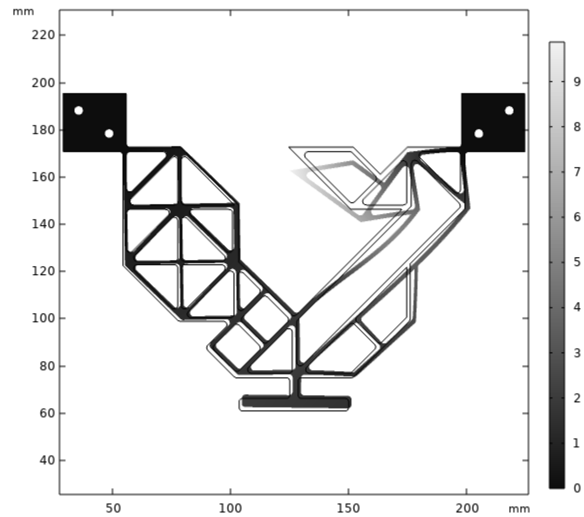
Material properties used are: Young's modulus = 1.2 GPa and Poisson's ratio = 0.45; they correspond to polypropylene.

a. Compare the COMSOL simulation results by hand-testing the real compliant mechanism available in the M2D2 laboratory (take TA's help for this). Does it work the way simulation results show? Record your observations.

Input force was varied from 0 to 50 N in steps of 1 N. The output displacement was recorded. Both linear and geometrically nonlinear analysis runs were done. The deformed plots for both are shown next for 50 N force. It is difficult to tell them apart. But in general, linear analysis (which does not subtract out rigid-body motion in kinematic analysis and assumes linear strain), the deformed structure appears "bloated" (i.e., increase in length (1D), area (2D), and volume (3D)). Here, that is not discernible. If you chose a mechanism that has bulky portions, you may be able to discern the difference between linear and geometrically nonlinear analyses.



Linear analysis (50 N)

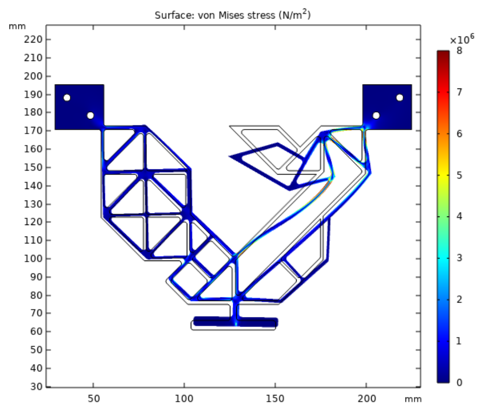


Geometrically nonlinear analysis (50 N)

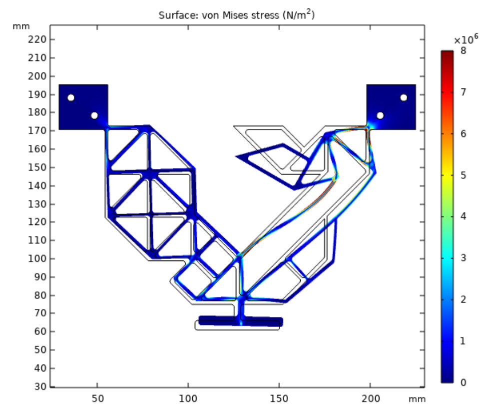
It can be seen in the videos at https://mecheng.iisc.ac.in/m2d2/CMcollection/9.html that the deformation pattern of the real device and the simulation are the same. Output displacement is slightly more than that in the linear analysis. So, this mechanism shows softening behaviour in the sense that the stiffness decreases with increasing load. Pertinent to this is that the long beam has buckled and does so increasingly and thus decreasing the stiffness. You will see this more clearly in the plot in answer to Question 2.

b. Extra 10 points if you 3D-print the mechanism (in small size so that you do not waste material of the 3D printer) (take TA's help for this) and see where it might break if you apply excessive load. Again, compare with COMSOL simulation results to see where the stress is excessive.

The stress plots for both linear and geometrically nonlinear analysis are shown next. Both cases have about 8 MPa of maximum von Mises stress. But the distribution of stress is slightly different. We see the peak stress (red color) in more places in the nonlinear case. This mechanism is likely to fail at the right support or in the long beam that is excessively deforming.



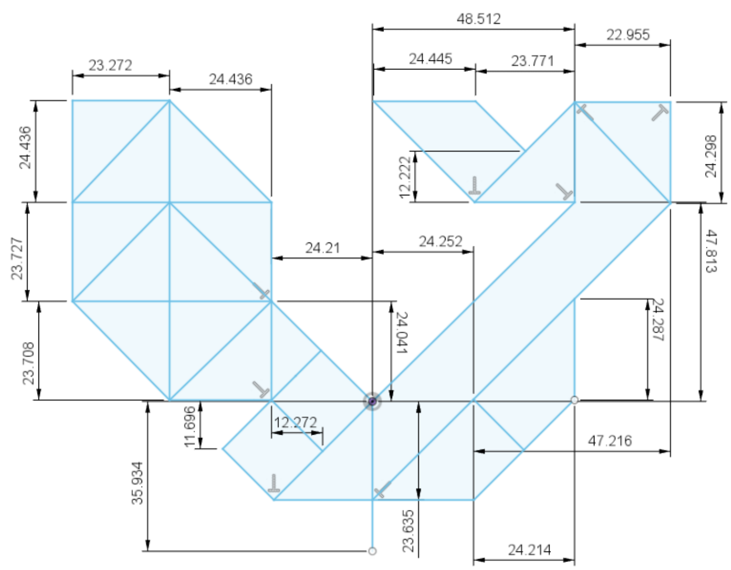
Linear analysis (50 N)



Geometrically nonlinear analysis (50 N)

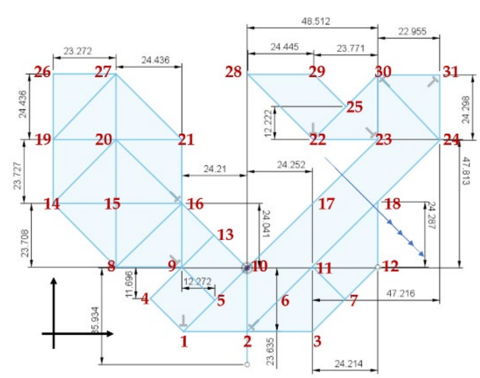
Question 2 (10 points)

For beam modeling in Matlab, the dimensions were added to the model as shown. All dimensions are in mm. Even though the side of the square in all squares should have been the same, it is not so. This is because of the junctions at which two or more beam segments meet. In any case, it is safe to use 24 mm as the side of the square for all. The four input files were created by assuming this.



Create a beam model of the compliant mechanism you have shown. For this, you need to create four data files (node.dat, elem.dat, forces.dat, and dispbc.dat) as per the Matlab files downloadable at https://mecheng.iisc.ac.in/suresh/me254/notes.html in the Week 2 row in the table.

The diagram here shows the node numbers in red. There are 31 nodes. Each node has three degrees of freedom (DoFs), namely, displacements in x and y directions and rotation about the z axis. The four data files were created as per this model.



The cross-section is rectangular for all beams with bread (b, out-of-plane thickness of 10 mm) and depth (*d*, in-plane width of 1.5 mm) were used. Note that *d* is the cross-section dimension in the plane of deformation of a beam. Thus, the second moment of area, $I = bd^3/12$.

A note about units is relevant here. The nodal coordinates are entered in mm in node.dat file. In elem.dat file, the cross-section dimensions are also in mm. Thus, there is a factor of 1000 for length units. To be consistent, the Young's modulus of 1.2 GPa is entered as 1.2E6 mN/mm². So, in forces.dat file, force is entered in mN.

node.dat			elem.dat					
1	48	0	1	1	2	10.0	1.5	1.2E6
2	72	0	2	2	3	10.0	1.5	1.2E6
3	96	0	3	1	4	10.0	1.5	1.2E6
4	36	12	4	1	5	10.0	1.5	1.2E6
5	60	12	5	2	6	10.0	1.5	1.2E6
6	84	12	6	2	10	10.0	1.5	1.2E6
7	108	12	7	3	7	10.0	1.5	1.2E6
8	24	24	8	4	9	10.0	1.5	1.2E6
9	48	24	9	5	9	10.0	1.5	1.2E6
10	72	24	10	5	10	10.0	1.5	1.2E6
11	96	24	11	6	11	10.0	1.5	1.2E6
12	120	24	12	7	11	10.0	1.5	1.2E6
13	60	36	13	7	12	10.0	1.5	1.2E6
14	0	48	14	9	13	10.0	1.5	1.2E6
15	24	48	15	10	13	10.0	1.5	1.2E6
16	48	48	16	13	16	10.0	1.5	1.2E6
17	96	48	17	8	9	10.0	1.5	1.2E6
18	120	48	18	8	14	10.0	1.5	1.2E6
19	0	72	19	8	15	10.0	1.5	1.2E6
20	24	72	20	9	16	10.0	1.5	1.2E6
21	48	72	21	8	16	10.0	1.5	1.2E6
22	96	72	22	10	17	10.0	1.5	1.2E6

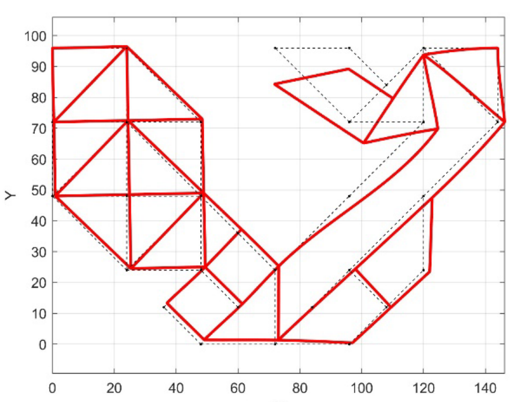
23	120	72	23	11	18	10.0	1.5	1.2E6
24	144	72	24	12	18	10.0	1.5	1.2E6
25	108	84	25	14	15	10.0	1.5	1.2E6
26	0	96	26	15	16	10.0	1.5	1.2E6
27	24	96	27	14	19	10.0	1.5	1.2E6
28	72	96	28	14	20	10.0	1.5	1.2E6
29	96	96	29	15	20	10.0	1.5	1.2E6
30	120	96	30	16	20	10.0	1.5	1.2E6
31	144	96	31	16	21	10.0	1.5	1.2E6
			32	17	23	10.0	1.5	1.2E6
			33	18	24	10.0	1.5	1.2E6
			34	22	25	10.0	1.5	1.2E6
			35	19	20	10.0	1.5	1.2E6
			36	20	21	10.0	1.5	1.2E6
			37	22	23	10.0	1.5	1.2E6
			38	19	26	10.0	1.5	1.2E6
			49	19	27	10.0	1.5	1.2E6
			40	20	27	10.0	1.5	1.2E6
			41	21	27	10.0	1.5	1.2E6
			42	22	28	10.0	1.5	1.2E6
			43	23	30	10.0	1.5	1.2E6
			44	24	30	10.0	1.5	1.2E6
			45	24	31	10.0	1.5	1.2E6
			46	25	29	10.0	1.5	1.2E6
			47	25	30	10.0	1.5	1.2E6
			48	26	27	10.0	1.5	1.2E6
			49	28	29	10.0	1.5	1.2E6
			50	30	31	10.0	1.5	1.2E6

forces.dat				dispbc.dat			
1	2	2	30E3	1	26	1	0
				2	26	2	0
				3	26	3	0
				4	31	1	0
				5	31	2	0
				6	31	3	0

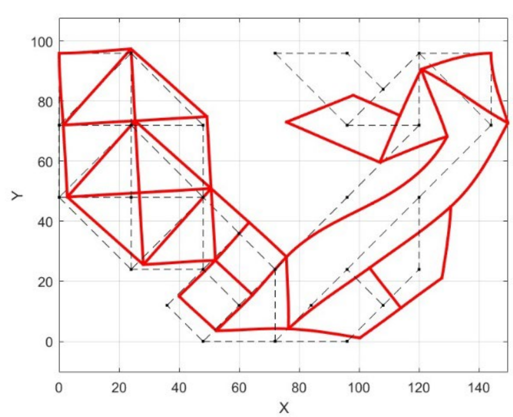
The input files are the same for both linear and nonlinear beam analysis when you use the Matlab codes supplied on the course website in Week 2.

a. Analyze using the linear beam finite elements in Matlab.
b. Analyze using the nonlinear beam finite elements in Matlab.
c. Compare Matlab beam FEA results with COMSOL's continuum-element FEA result, both linear and nonlinear. Record your observations.

The results of deformation analysis are shown next for 30 N force at the input. The deformation patterns are noticeably different in linear and geometrically nonlinear beam analyses. There is more deformation in nonlinear analysis, which confirms that this mechanism shows softening behaviour.



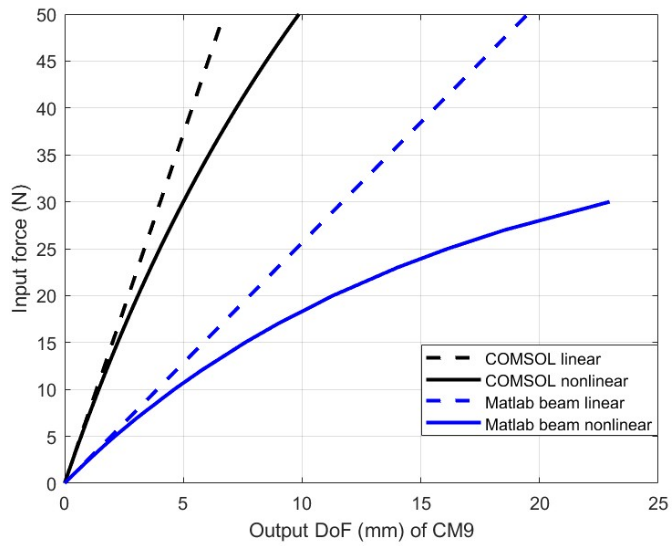
Linear analysis with 30 N



Nonlinear analysis with 30 N

Next, we compare the deformation behaviour of COMSOL (continuum model) and Matlab (beam model) analyses, both with linear and geometrically nonlinear. This can be seen next.

Clearly, finite element analysis in COMSOL (continuum model) analysis shows a stiffer behaviour than Matlab (beam model). This is due to two reasons. The first is that the beam model does not account for junction stiffness at the points where two or more beams meet. The second is that the COMSOL model had beams of slightly different in-plane widths. The first reason is more dominant than the second one. The important point to notice here is that when multiple beams meet at a point, the continuum has a stiff junction. This reduces the effective length of the beam segments. This points to a flaw in beam modeling. We will discuss this later in this course.



An important feature to notice is that the linear force vs. output displacement curve (the line, actually) is tangential to the nonlinear curve in both cases (COMSOL and Matlab). This is how it should be for any nonlinear behaviour when we linearize.

Notice also that in both models (continuum in COMSOL and beam in Matlab), we see softening behaviour as this mechanism's stiffness decreases with increasing input force (or output displacement). This can be attributed to the buckled beam.

Notice also that nonlinear beam result is shown only up to 30 N. Just beyond that, the program fails to converge. This is because the stiffness becomes too low. This can be seen in the curve as it is about to flatten with nearly zero stiffness.

We conclude that the beam modeling is more flexible than the continuum model due to lack of accounting for junction stiffness in the real device and the continuum model. We need to watch out for this flaw when we simulate compliant mechanisms using beam finite elements.

One more thing you should observe:

Trace the path of the output point in linear and geometrically nonlinear analyses by increasing the force form 0 to a large value up to which the program converges. The path should be straight in the linear analysis and curved in nonlinear analysis.