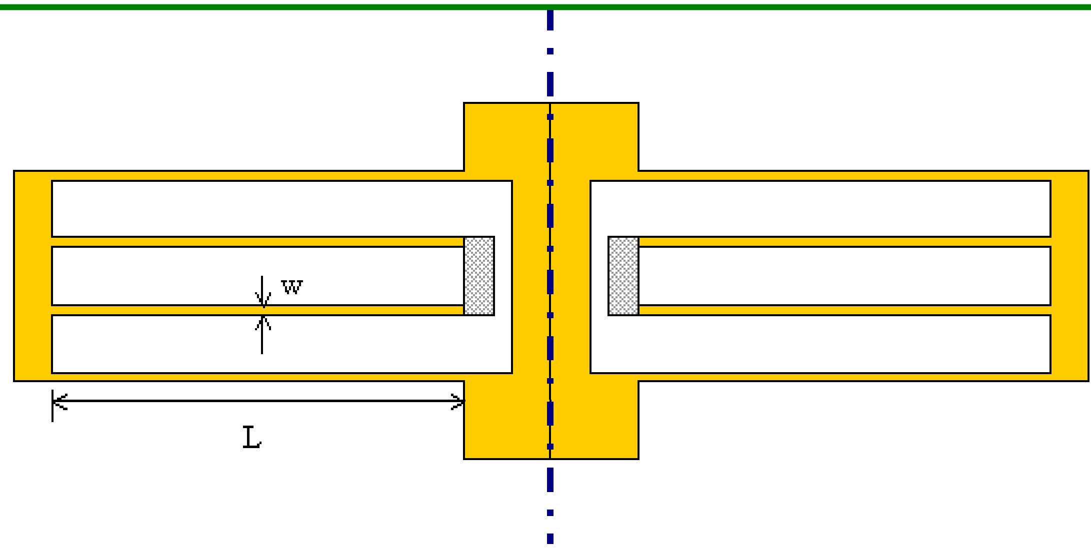
Practice problems #5

Problem 1



The above figure shows the suspension of the electrostatic comb-drive. Except the shaded gray region, the rest of the structure is free to move above the substrate separated by a gap. Obtain the linear stiffness of the suspension in the vertical direction (i.e., along the axis of symmetry shown in the figure) in terms of the length L, width w, and the Young's modulus E, and the layer thickness t. Obtain the expression for the maximum stress as well.

Use E = 150 GPA, ν = 0.25, L = 200 um, w = 5 um, and t = 3 um.

A fixed-guided beam (length = l, width = b, depth = h) with a transverse tip load of F has the following formulae for maximum transverse deflection δ, and maximum axial stress σ:

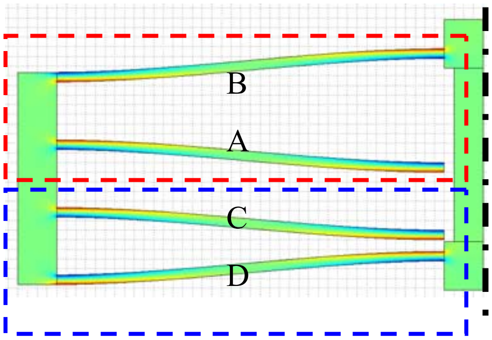
$$\delta = \frac{Fl^3}{12EI} = \frac{Fl^3}{12E\left(\frac{bh^3}{12}\right)} = \frac{Fl^3}{Ebh^3}$$

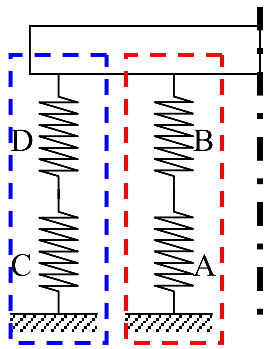
So, the stiffness of one fixed-guided beam is, $k = \frac{F}{\delta} = \frac{Ebh^3}{l^3}$.

In our notation and values, $k = \frac{Ebh^3}{l^3} = \frac{Etw^3}{l^3} = \frac{150E9 * 3E-6(5E-6)^3}{(200E-6)^3} = 7.0313 \quad N/m$

If we denote each fixed-guided beam as a spring of stiffness k, in the left half of the comb-drive suspension, there will be four such springs. They are to be arranged as shown in the figure below. This is because beams A and B (and also C and D) have the same force. So, they are springs in series. The A-B spring pair and C-D spring pair have the same displacement and hence are in parallel. Looking at the spring-schematic, we can observe that the stiffness of the left half of the suspension is k (note that A-B and C-D spring pairs each have $k/2$, together they have $k = k/2 + k/2$).

The left half and right half of the suspension are in parallel (since they have the same displacement). So, the total stiff of the suspension is $k + k = 2k = 14.0626\ N/m$.





<u>Stress</u>

$$\sigma = \frac{Mc}{I} = \frac{(Fl/2)(w/2)}{(w^3 t/12)} = \frac{3Fl}{w^2 t}$$

By substituting for F in terms of δ, we get $\sigma = \dfrac{3Ew\delta}{l^2}$.

Note that the δ of one fixed-guided beam is half of the displacement of the central shuttle $\delta_{shuttle}$. That is, $\delta = \dfrac{\delta_{shuttle}}{2}$.

So, maximum stress = $\sigma = \dfrac{3Ew\delta_{shuttle}}{2l^2}$.

If $\sigma = 900\ MPa$ is the maximum permitted stress before failure occurs, the maximum permitted shuttle displacement is given by

$$\delta_{shuttle}^{\max} = \frac{2l^2\sigma_{\max}}{3Ew} = \frac{2(200E-6)^2 900E6}{3(150E9)(5E-6)} = 32E-6\ m = 32\ \mu m.$$

Then, per beam, the maximum deflection is 16 μm, which is well above the limit for which the linear deflection analysis is valid. So, this should be thought of as only an estimate for the maximum deflection. When a beam undergoes large deflections, geometrically nonlinear analysis needs to be done. A beam such as this will experience increased stiffness due to the axial stretching (recall that linear analysis assumes that the length of the beam along the neutral axis does not change).

Problem 2

Use a finite element analysis (FEA) software to check the validity of the above formula via linear FEA. (FEMLAB is used below with plane stress elements but you will be using a Matlab program that uses beam elements.)

A note about units

In FEMLAB, we use micron units for all dimensions. That is, we are multiplying the length units by a factor of 10^6 when we enter into the FEMLAB program. This must be done with all quantities appropriately. For example, if we want to apply a force of 1 microN, we would enter into FEMLAB as 1 because we need to multiply by 10^6 as force has a dimension of [L] in it. On the same token, the Young's modulus of 150E9 Pa should be entered as 150E3. Here, we multiplied by 10^{-6} because Young's modulus has [L^{-1}] in it. Poisson's ration, being dimensionless, is entered as is (i.e., 0.3).

Likewise, when we read something out of FEMLAB, we should make sure that we multiply by the reverse factor. That is, if FEMLAB shows a stress of 1, it means that it is actually 10^6 Pa or 1 MPa (note that stress has [L^{-1}] in it ➔ so, if we are reading out of FEMLAB, we should multiply by 10^6).

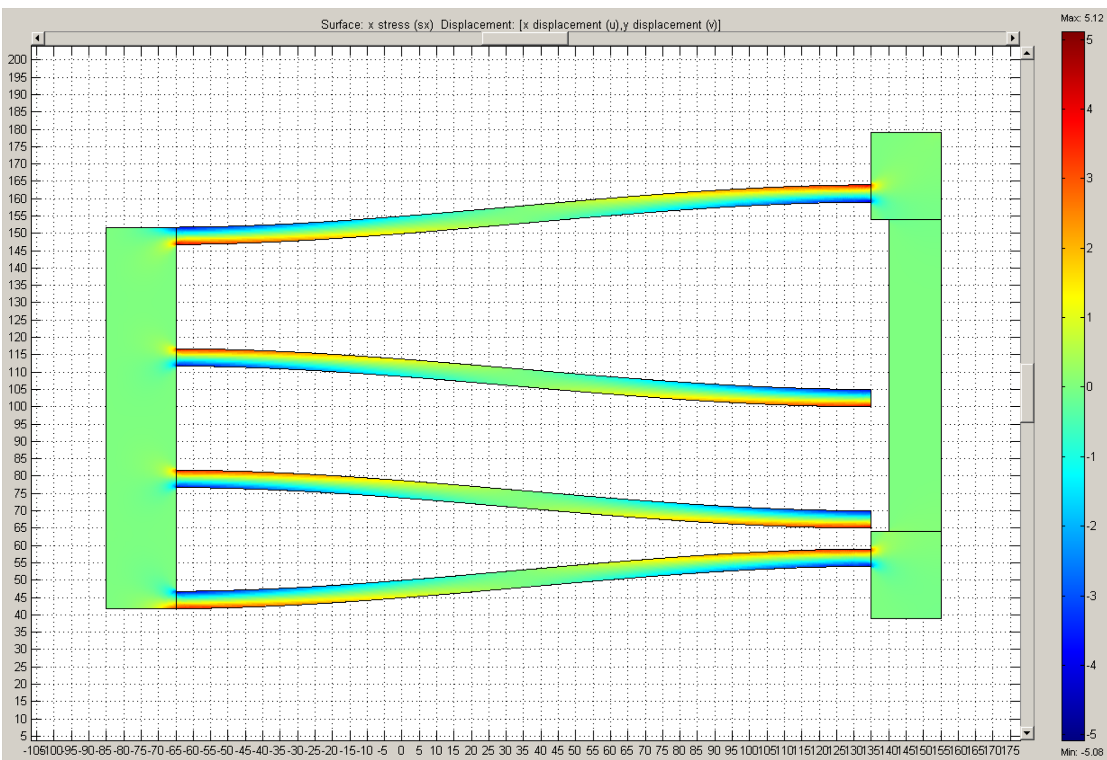
Only the left half of the suspension is modeled in FEMLAB. This means that the symmetry boundary condition is imposed along the axis of symmetry by restricting only the *x*-displacement to zero.

Using the default triangular mesh at first and then refining it three times, we get the maximum displacement and maximum stress as follows.

Results with 1E-6 N applied as shown above.

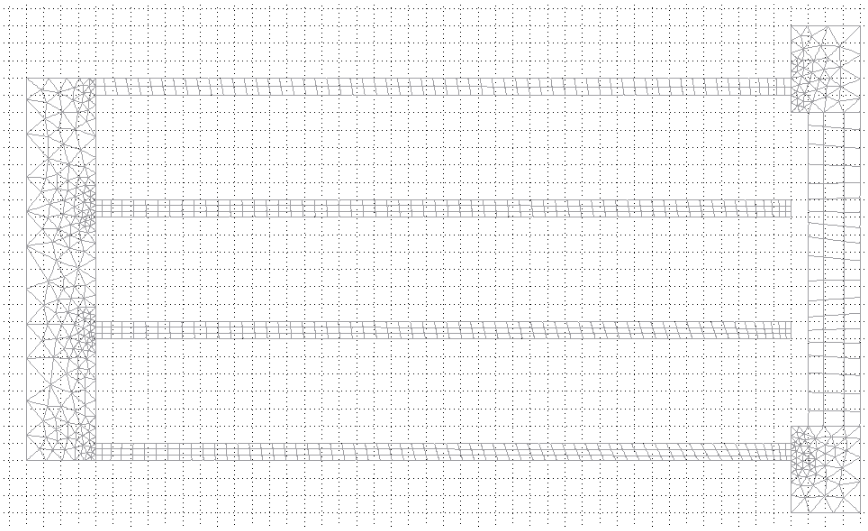
Mesh type	$\delta_{shuttle}\,(\mu m)$	Stiffness = force/displacement = $1E-6/\delta_{shuttle}$ (N/m)	$\sigma_{\max}$ (Mpa)
Default triangular	0.0935	10.6952	1.68
One level finer	0.129	7.7519	2.98
One more level finer	0.144	6.9444	3.97
One more level finer	0.149	6.7114	5.12

We can see that the solution is converging with the refining of the mesh. The stiffness obtained with left half of the suspension can be taken as 6.7114 N/m. For comparison, consider that the analytical formula gave 7.0313 N/m. The difference can be attributed to the shear stresses that are ignored in the analytical calculation. Furthermore, the stress (σ_x) below shows that the beams are not the only places where the stress is non-zero: the portions connected to the beams in the shuttle as well as the side rigid part also experience some stress.



Stress plot with the point force applied.

You can choose quad elements in "Specify Element…" option in FEMLAB if you use the Structural Mechanics Module 1.1. You can click on each subdomain and choose 4-node quadrilateral and "apply" it. For some subdomains, such a mesh is not possible. For those, keep the triangular mesh. The resultant mesh is shown below at the default level.



Results with 1E-6 N applied but with 4-node **quadrilateral mesh**

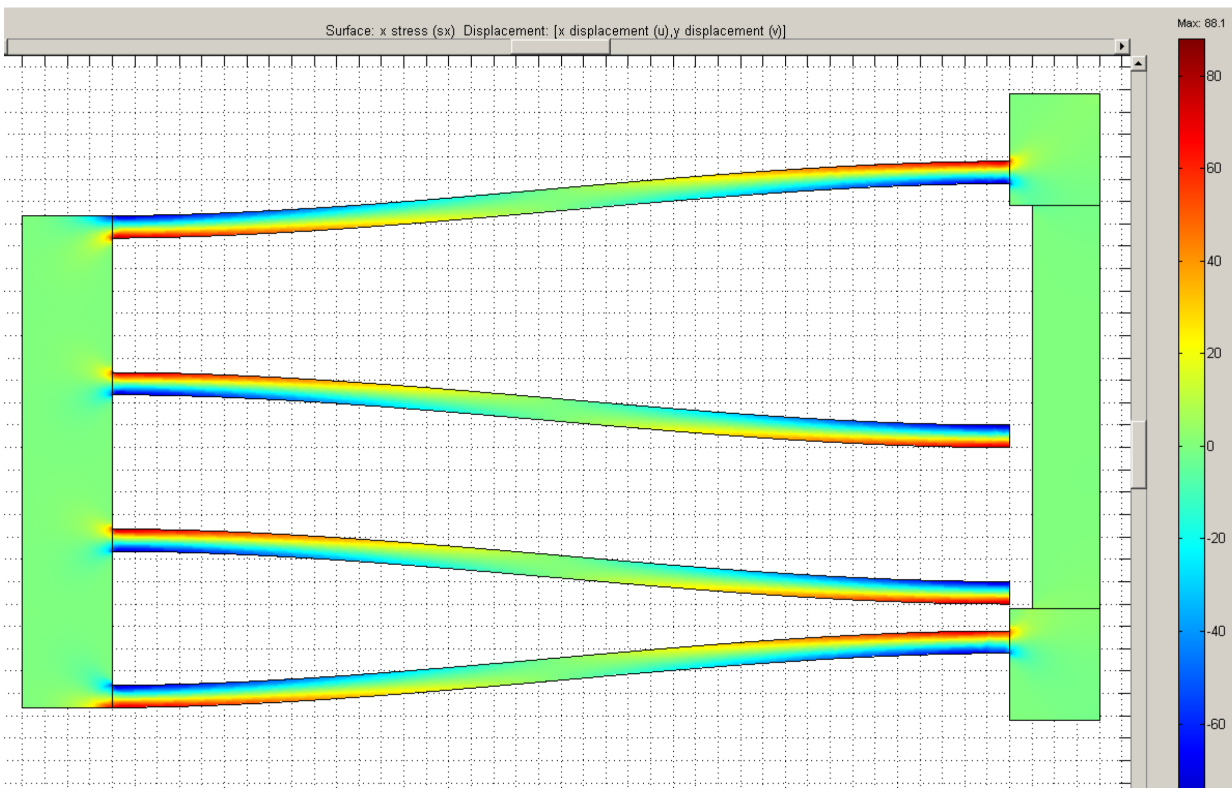
Mesh type	$\delta_{shuttle}\,(\mu m)$	Stiffness = force/displacement = $1E-6/\delta_{shuttle}$ (N/m)	$\sigma_{\max}$ (Mpa)
Default triangular	0.125	8.0000	2.73
One level finer	0.143	6.9930	3.52
One more level finer	0.149	6.7114	4.36
One more level finer	0.150	6.6667	5.36

Now, we see that convergence is faster with refinement. Quad elements are accurate even with one-level coarser mesh than with the triangular elements.

Problem 3

If the failure stress of the material (polysilicon) is 900 MPa, what is the maximum vertical deflection of the central mass before the failre state is reached? Once again, do a calculation and then verify with FEA results.

We know from above that 32 μm is the maximum permissible shuttle displacement before failure stress is reached. This is based on linear analysis. Let us apply, 3.2 μm **displacement boundary condition** instead of the force. This can be done in the same place where we specify the forces. Just put in the displacement value for the upper-right corner of the shuttle. Then, we can see how close the maximum stress is to 90 Ma (900/10 = 90, because we applied only 3.2 μm instead of 32 μm).



The stress (σ_x) plot with 3.2 μm displacement applied.

The maximum stress is 88.1 MPa with second-level finer mesh. It is 90 MPa according to the formula. With third-level finer mesh, it is 110 MPa.

The stress at the sharp corners will be theoretically infinite. In FEM, it will keep increasing with mesh refinement. So, the corners should be rounded and care should be exercised in interpreting the maximum stress in the results of FEM.