MNIS – Physical models for micro and nanosystems

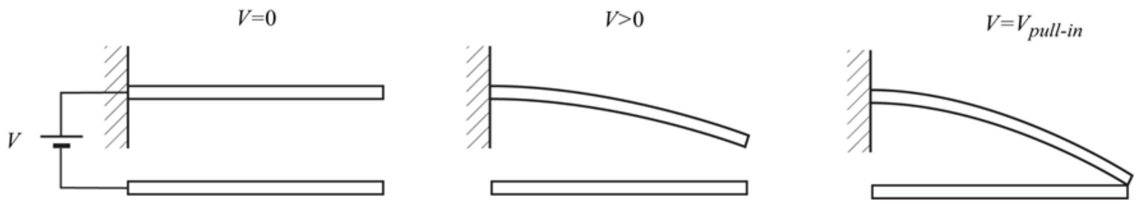
Exercise 5: Electrical and Mechanical Coupling

WHAT ARE WE GOING TO LEARN:

-how to use electrostatic force to deform materials
-use of moving mesh

In this model, we will cover a simple example of a cantilevered beam with electrostatic force applied on its bottom face. The goal is to introduce the approach to modeling electrical and mechanical coupling.

We will consider a simple cantilever presented on the next figure. The application of a voltage difference between the cantilever and the substrate, causes the cantilever to bend until it snaps into contact with the substrate. Our task is to identify the voltage at which this event occurs ($V_{pull\text{-}in}$).



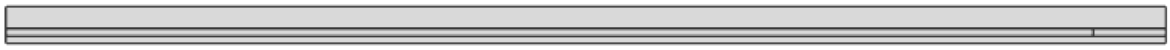
1. BUILDING THE MODEL

We will first start Comsol and define the type of geometry as **2D** and choose the physics package **Structural mechanics ► Solid mechanics.** Select **study type ► stationary.** Choose **done.**

We should now build the model. It will consist of three rectangles, defining the bending cantilever and the empty space surrounding it. Go to **Component 1 ► Geometry 1** and create three rectangles with the following sizes and coordinates:

Name	**width**	**height**	**x position**	**y position**
Rectangle 1	3e-4	2e-6	0	2e-6
Rectangle 2	3.2e-4	1e-5	0	0
Rectangle 3	2e-5	2e-6	3e-4	2e-6

The model should look like this:



Also, create a parameter:

Name	**Expression**	**Description**
V_in	1[V]	Input voltage

We can now proceed to the next steps. First, we have to define material properties.

2. DEFINING MATERIAL PROPERTIES

The cantilever will be made of polysilicon, a common material used in the fabrication of MEMS. Being polycrystalline, polysilicon can be approximated as an isotropic material. Load the corresponding material definition into the model and assign it to the cantilever.

The surrounding area will be made of air. Please also load it and assign it to the area surrounding the cantilever.

We can now move to the next step and define the boundary conditions.

3. LOADING PHYSICS PACKAGES

Right-click on **Component 1** in the model builder and load the following packages from add physics (if not already loaded): **mathematics ► deformed mesh ► Legacy Deformed Mesh ► Moving Mesh (ale), AC/DC ► Electric Fields and Currents ► electrostatics**. Select **Add to Component 1** for all of them.

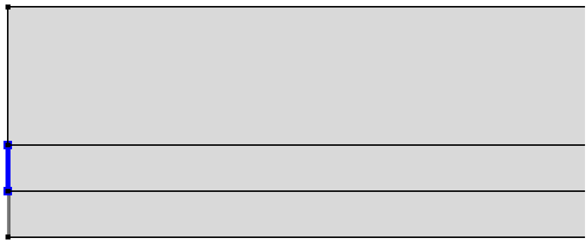
Under solid mechanics, keep only the cantilever domain (domain 2 in my case) and remove all others. Under **thickness** enter 20e-6.

4. DEFINING BOUNDARY CONDITIONS

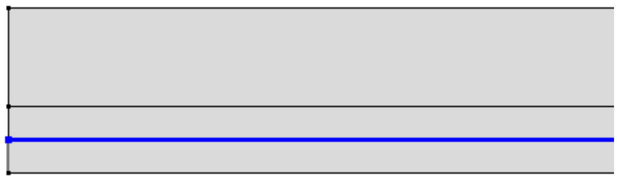
Here, we have to define the boundary conditions for all three physics packages. Let us start with solid mechanics.

Solid mechanics

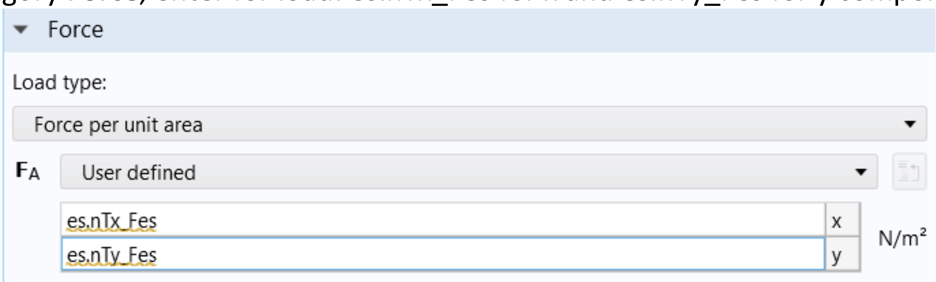
Add a **fixed constraint** boundary condition and assign it to the left, clamped edge of the cantilever.



Add a **boundary load** boundary condition and assign it to the bottom edge of the cantilever.



In the category **Force**, enter for load: es.nTx_Fes for x and es.nTy_Fes for y components:



This will use the electrostatic force, calculated in the electrostatic part of the simulation to be used as the distributed load acting on the bottom face of the cantilever. You should also remove the domains corresponding to air from the solid mechanics **Domain Selection** so that Comsol would not try to treat air as a solid that deforms in this simulation.

Moving mesh

Add a **Prescribed deformation** domain condition. Assign it to the cantilever. Under **prescribed mesh displacement** enter **u** under **dx** and **v** under **dy**.

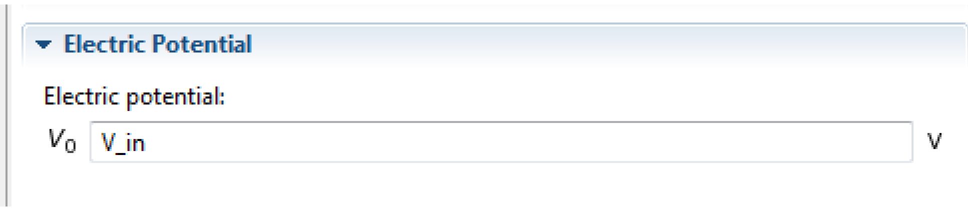
Add a free deformation domain condition and assign it to all the other areas, corresponding to air.

Add a prescribed mesh displacement boundary condition and assign it to the edge of the entire region to be modeled. Keep all other values default (0 displacement).

Electrostatics

Add a **ground** boundary condition and assign it to the bottom of the edge of the region to be modeled, representing the substrate.

Add an **electric potential** boundary condition and assign it to the bottom of the cantilever. Enter V_in under V0 in the electric potential setting.



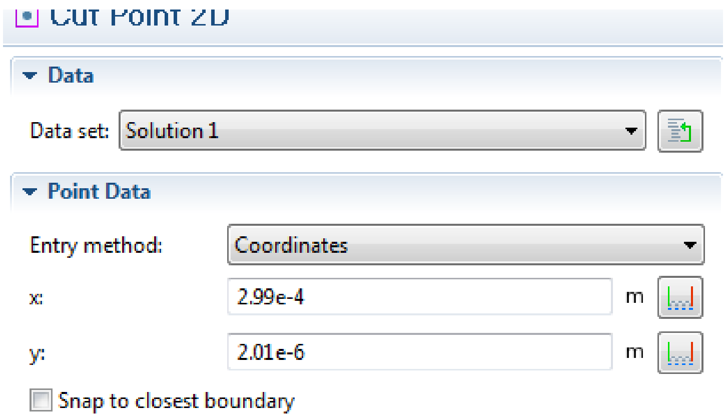
Add a **force calculation** domain condition and assign it to the cantilever. Enter Fes under **force name.** This is where we calculate the force used in the mechanical part.

5. PARAMETRIC SWEEP

Let us now look at the simple deformation-input voltage relationship in this system. To do this, we will define a new parametric sweep under study. The parameter will be V_in, ranging from 0 to 8 V, in 0.25V steps (range(0,0.25,8)).

6. PLOTTING RESULTS

In order to plot the deflection of the end point of the cantilever, we should define a new **cut point 2D** data set under results. Under coordinates, enter 2.99e-4 for **x** and 2.01e-6 for **y.** Let's rename it **right end** for bookkeeping purposes.



To plot the displacement, we should create a new point graph under a new 1D plot group. For y-Axis data expression under the point graph, choose **solid mechanics –> total displacement** or enter solid.disp.

If everything is correct, you should get the following graph: