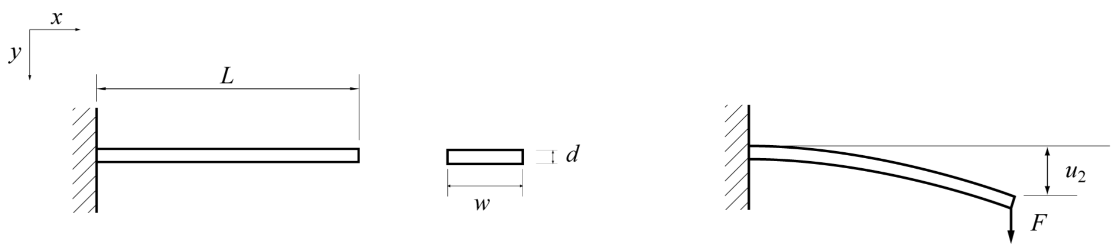
MNIS – Physical models for micro and nanosystems

Exercise 3: Simple Cantilever Bending

WHAT ARE WE GOING TO LEARN:

-basics of modeling mechanical deformation
-typical boundary conditions encountered during modeling of mechanical deformation

In this model, we will cover a simple example of a cantilevered beam with force applied to its end. The goal is to introduce the basic boundary conditions related to mechanical modeling.



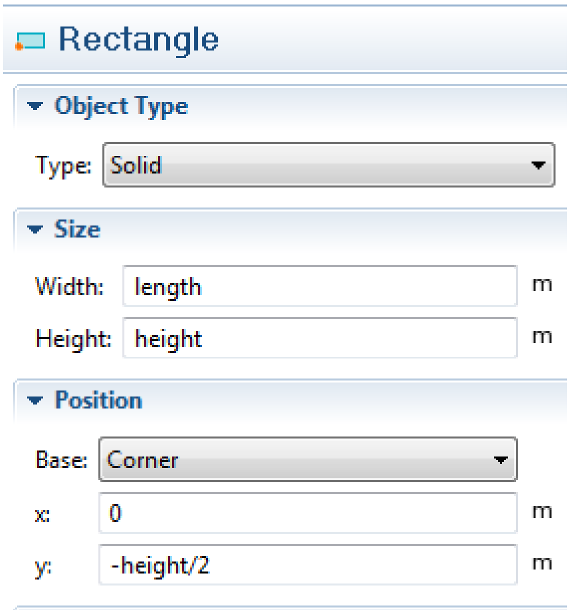
1. BUILDING THE MODEL

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

We should now build the model. It will consist of a simple rectangle with a parameter-driven geometry. First let's define the following parameters:

Name	**Expression**	**Description**
Fy	1000 [nN]	Force - end
length	300e-6 [m]	Cantilever length
height	2e-6 [m]	cantilever height
thickness	20e-6 [m]	cantilever thickness

We can build the model which will consist of a simple rectangle by right clicking on **Component 1**► **Geometry 1** and choosing **rectangle.** Under width add **length,** height should be **height.** Under position choose **corner**, x: 0 and y: **-height/2.**



You will also need to enter the cantilever thickness. The fact that this is a 2D model means that is possesses a translational symmetry in the direction perpendicular to the plane in which you define the model. However, in order to get numbers that can relate to “real” physical quantities, a finite thickness is assumed, by default equal to 1 m. You can modify this thickness by right-clicking on **solid mechanics** in the **model builder** tree. Enter **thickness** under **thickness ▸ d**.

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

2. DEFINING MATERIAL PROPERTIES

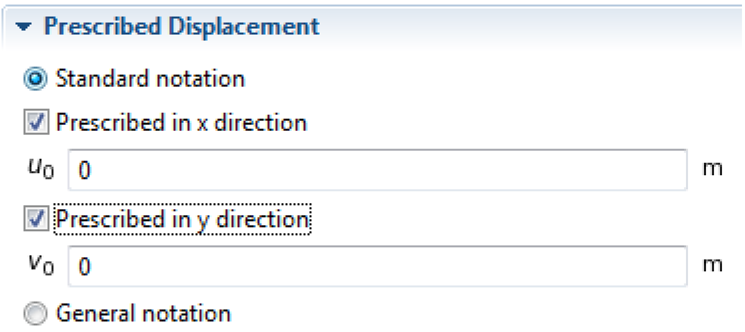
The cantilever will be made of polysilicon, a common material used in the fabrication of MEMS. Being polycrystalline, polysilicon can be approximated as an isotropic material. Load the corresponding material definition into the model and assign it to the cantilever.

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

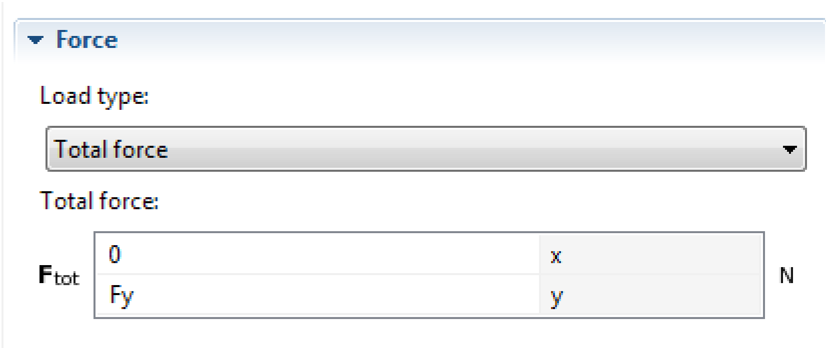
3. DEFINING BOUNDARY CONDITIONS

Here, we have to define the boundary conditions. By default, Comsol assigns the free boundary condition to all boundaries. This means all boundaries are free to move under the application of the force. We have to define the boundaries that correspond to the left (clamped) and right (applied force) side of the cantilever

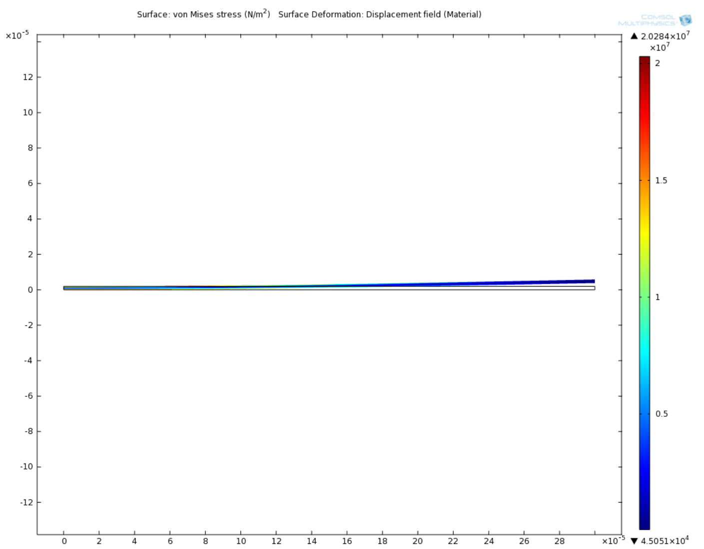
Right click on **Model ► Solid mechanics** and add **prescribed displacement.** Add the left side of the cantilever to boundary selection. Under prescribed displacement select **prescribed in x direction** and **prescribed in y directions.** Both values should be **0.**



The right-hand side will have a defined load. Right click on **Model** ► **Solid mechanics** and add **boundary load.** Assign it to the right-hand side of the cantilever. Select total force under **load type** and Fy under **total force, y** component**.**



We can now start compute and if everything is OK we should get this kind of a stress distribution:



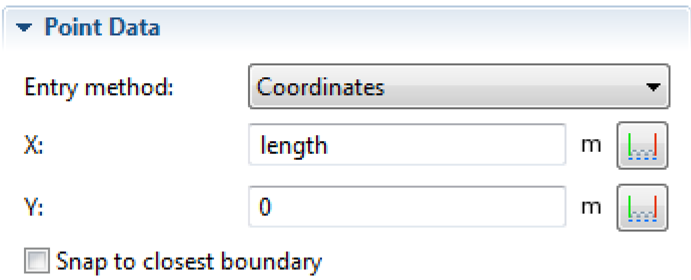
By default, Comsol scales the deformation so it would appear to be significant. Unfortunately, this can obscure the final result, as situations with different absolute deformations can appear to be same. To correct for this behavior, go under **results** ► **stress** ► **surface 1** ► **deformation** and enter 1 under **scale** ► **scale factor.**

4. PARAMETRIC SWEEP

Let us now look at the simple force-deformation relationship in this system. To do this, we will define a new parametric sweep under study. The parameter will be Fy, ranging from 0nN to 1000 nN, in 100nN steps (**range(0,1.0e-7,1.0e-6)**).

5. 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 **length** for **x** and **0** 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: