MNIS – Physical models for micro and nanosystems

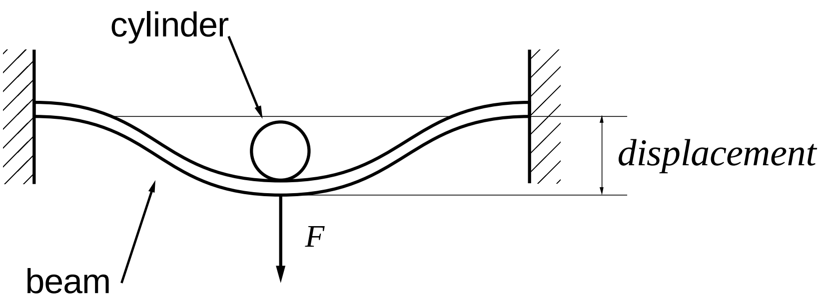
Exercise 4: Beam getting deformed by a cylinder

WHAT ARE WE GOING TO LEARN:

-how to mechanically deform one object using another, moving object

In this model, we will cover a simple example of a beam that is being deformed in contact with another object. The goal is to introduce the approach to modeling general problems in nanomechanics where one object (for example an AFM tip) is used to deform another object.

We will consider a simple beam presented on the next figure. The beam is in contact with a cylinder, pressing on it in the middle. The displacement of the cylinder causes the beam to bend. Our task is to find the resulting mechanical force and the relationship between force and displacement.



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 **finish.**

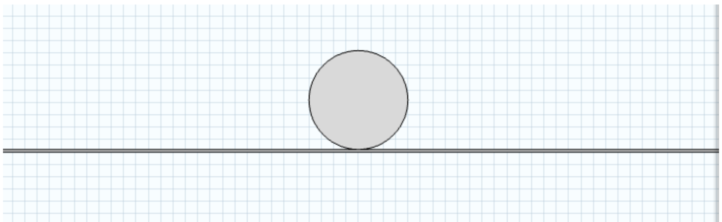
Let us first create the parameters:

Name	Expression	Description
displacement	0.1 [m]	Cylinder displacement
length	20 [m]	Beam length
radius	0.2 [m]	cylinder radius
height	0.01 [m]	beam thickness

We should now build the model. It will consist of a rectangle, corresponding to the bending beam and the cylinder pressing on it. Go to **Component 1 ▸ Geometry 1** and create these objects with the following sizes and coordinates:

Name	**width**	**height**	**x position**	**y position**
Rectangle 1	length	height	-length/2	0
	Radius			
Circle1	radius		0	radius+height

The model should look like this:



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

2. DEFINING MATERIAL PROPERTIES

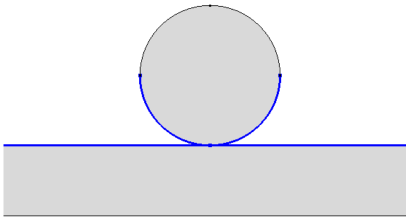
The beam will be made of steel. Load the corresponding material definition (material **structural steel**) into the model and assign it to the beam.

The cylinder will be made of polysilicon. Please also load it and assign it.

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

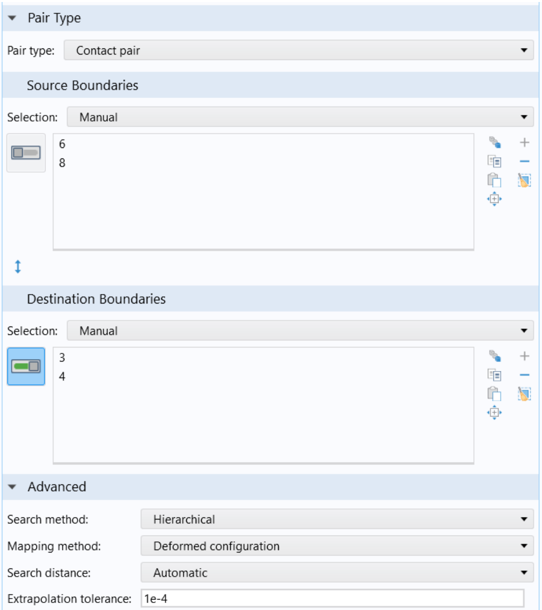
3. DEFINING THE CONTACT

We should now define the contact region. This in principle defines boundaries that cannot penetrate through each other during mechanical deformation. To do this, right click on **Component1▸Definitions,** in the model builder, select menu option **pairs**, create **contact pair** involving the following 4 boundaries (2 bottom boundaries on the cylinder + 2 top boundaries on the beam):



Boundaries on the circle should be the **source,** while those on the beam should be the **destination boundaries.**

This is what the settings should look like:



4. DEFINING BOUNDARY CONDITIONS

Now we have to define the boundary conditions. The left and right edges of the beam will be fixed in place. You can define it by adding a **fixed constraint** boundary condition and applying it to the two edges of the beam.

You should now create a domain-level **prescribed displacement** condition and apply it to the cylinder. Under **prescribed displacement**, you should put **-displacement** for the displacement in the y direction (u_{0y}) and 0 for the x direction.

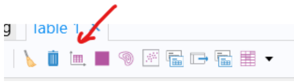
Finally, you should add a contact boundary condition. You can find it in the boundary condition menu, under the section pairs. Do not forget to select the contact pair 1 under **Pair selection**.

5. PARAMETRIC SWEEP

Let us now look at the simple deformation-force relationship in this system. To do this, we will define a new parametric sweep under study. The parameter will be displacement, ranging from 0, to 0.1, in steps of 0.005. Tick the **plot** check box under **results while solving.** Start the sweep.

6. PLOTTING RESULTS

In order to plot the force vs. displacement curve we first need to calculate the force. To do this, right click on **derived values** under **results** and select **integration ▸ surface integration**. The expression should be: **-solid.RFy**. Under **description**, enter **Force_y.** Select the cylinder under **selection.** Click on the equal sign on the top of the settings window to carry out the calculation. This will generate a new table in the section **tables** with the pairs of displacement and force.

Go to the table and click on the plot icon: . It should be in the section **results,** bellow the graphics window. This should be the result: