

---

## COMSOL Multiphysics: design of a consumer MEMS accelerometer

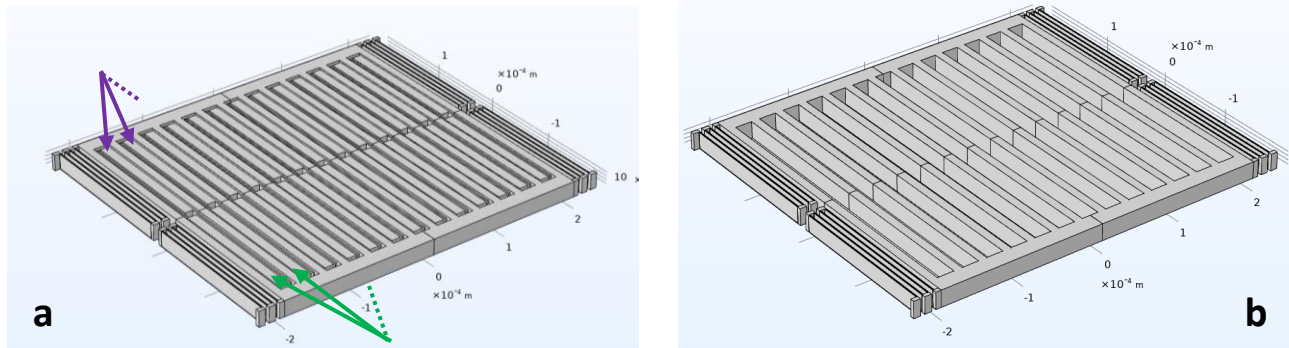
---

Giacomo Langfelder

25/10/2023

### Introduction

The class will discuss the micromechanical modeling of a MEMS accelerometer. The structure is formed by a mass, suspended through four folded springs at the corners, and accommodating nested holes for parallel plates. Stators (shown in the left picture below, green and purple) will not be simulated as they do not take part in the eigenfrequency calculation of structural modes. The goal is (i) to confirm our theoretical predictions on the value of the stiffness and of the resonance frequency of the first mode, considering the full body instead of the lumped spring-mass damper system approximation, and (ii) to explore the presence of possible high-order modes.



The exercise is split into two parts: the first one presents a *stationary study* for the evaluation of the overall elastic stiffness  $k$ , the latter is the study of the device *resonance modes* ( $f_0$ ).

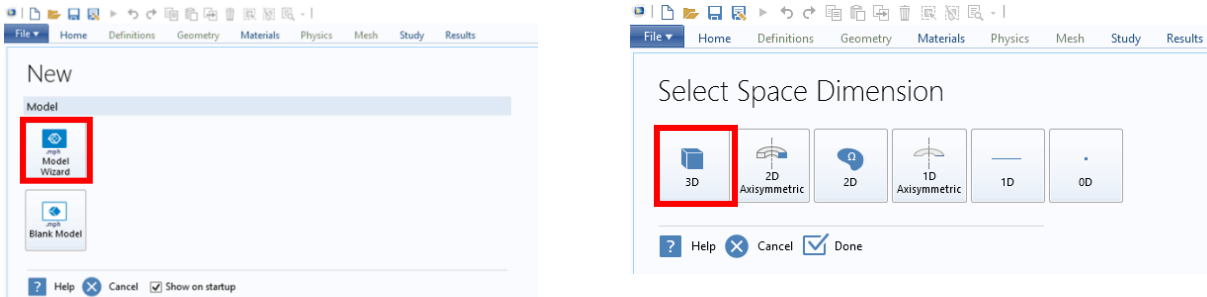
We assume the following parameters as fixed by the design specifications: side length of the mass  $l_{mass} = 400 \mu\text{m}$ ; process thickness  $thickness = 20 \mu\text{m}$ ; resonant frequency  $f_0 = 4 \text{ kHz}$ ; use of 4 springs, 4-times folded with fold length  $l_{fold} = 185 \mu\text{m}$ .

The goal will be to find the correct width of the folds that matches the required stiffness in order to have the target resonance frequency. Before proceeding to that step, an evaluation of the effective mass value will be required. The geometry is relatively simple, but this will be used as a paradigmatic example for next classes with more complex geometries.

The simulation is split into the usual steps: choice of the physical problem to solve, definition of materials and geometry (this time in full 3D mode), definition of boundary conditions, solution of the problem and analysis of the results.

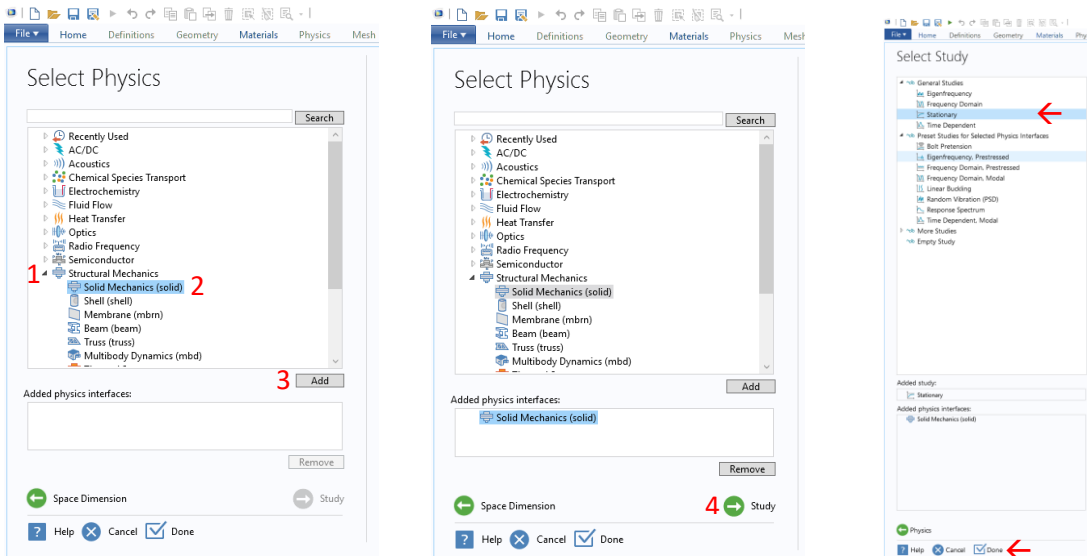
# 1. Design of the structure

Open Comsol Multiphysics 5.5 → choose “Model Wizard” → in Select Space Dimension choose “3D”

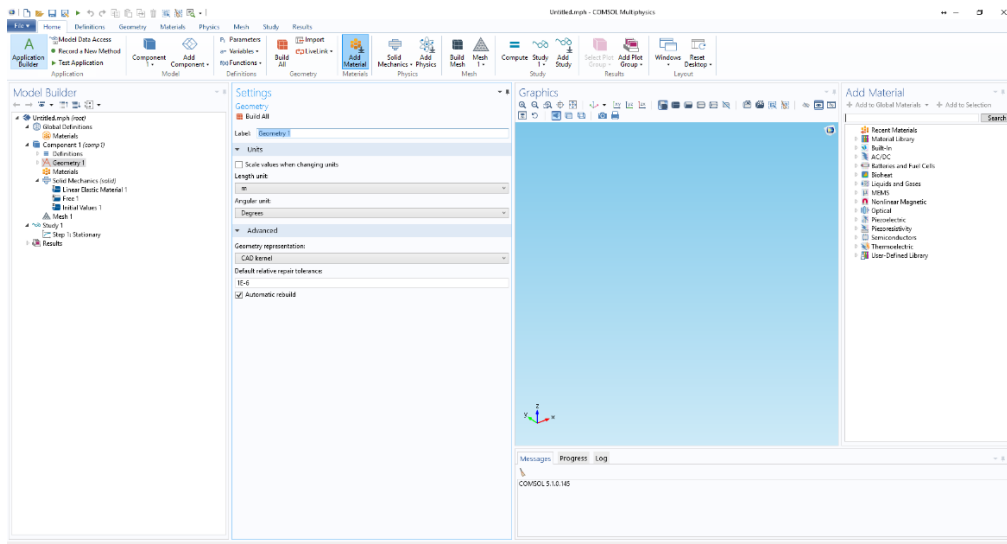


In Select Physics: 1) open the selection window “Structural Mechanics” → 2) choose “Solid Mechanics (solid)” → 3) click “Add”, the selected physic appears inside the box “Added physics interfaces” → 4) click “Study”

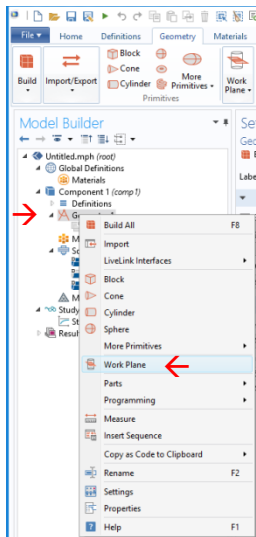
In “Select Study”, click on “Stationary” → click on “Done”



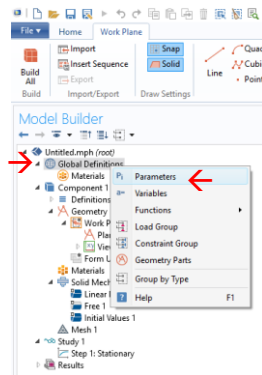
The well-known main COMSOL home interface is now open.



For the sake of simplicity, we are going to design our structure first on a **2D plane**; then we will extrude it in the out-of-plane direction by a distance corresponding to the epitaxial layer thickness (indeed, almost all the MEMS fabrication processes are “planar” process, where the design occurs in 2D and the third dimension is fixed).



In the “Model Builder” window, *right click* on “Geometry 1” → in the pop up window select “Work plane” (this puts you on a 2D plane).

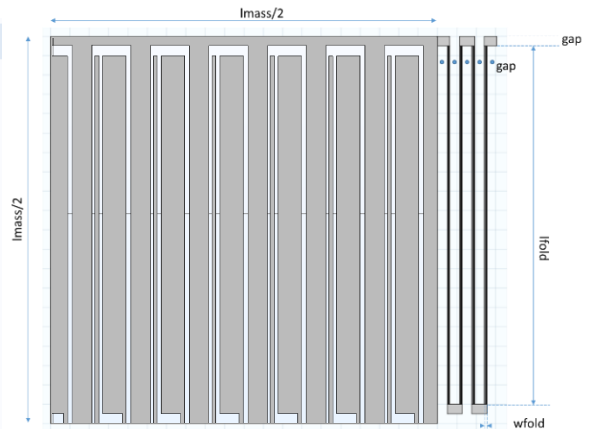


Now, we can design on this 2D plane. We have to design manually all the shapes: if you right click on “Plane Geometry”, a pop-up window appears and you can select various shapes, for which you have to insert the dimensions. As we saw in the last CAD class, it is very useful to first define a set of **parameters**:

To do this, in the “Model Builder” window, *right click* on “Global Definition” → in the pop-up window select “Parameters”.

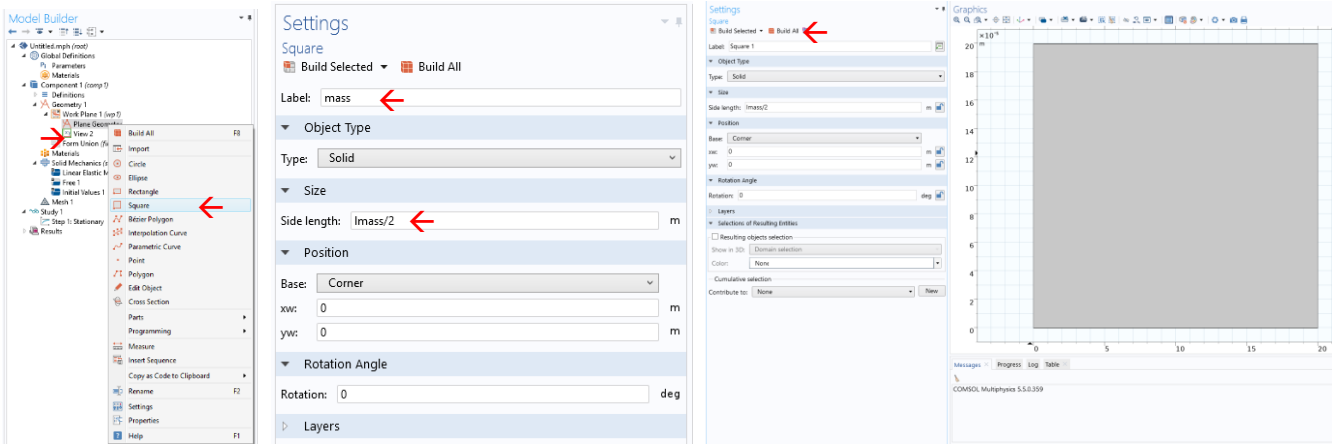
We can add all the dimensions of our geometry (the column “Description” is facultative).

| Parameters                      |            |         |                           |
|---------------------------------|------------|---------|---------------------------|
| Name                            | Expression | Value   | Description               |
| E                               | 160e9      | 1.6E11  | Young's modulus           |
| thickness                       | 20e-6      | 2E-5    | epitaxial layer thickness |
| l <sub>mass</sub>               | 400e-6     | 4E-4    | mass side                 |
| l <sub>fold</sub>               | 185e-6     | 1.85E-4 | spring fold length        |
| n <sub>fold</sub>               | 4          | 4       | number of folds           |
| n <sub>spring</sub>             | 4          | 4       | number of springs         |
| gap <sub>fold</sub>             | 5e-6       | 5E-6    | gap between folds         |
| w <sub>min<sub>mass</sub></sub> | 10e-6      | 1E-5    | minimum mass width        |
| w <sub>hole<sub>PP</sub></sub>  | 20e-6      | 2E-5    | width of holes for PP     |
|                                 |            |         |                           |



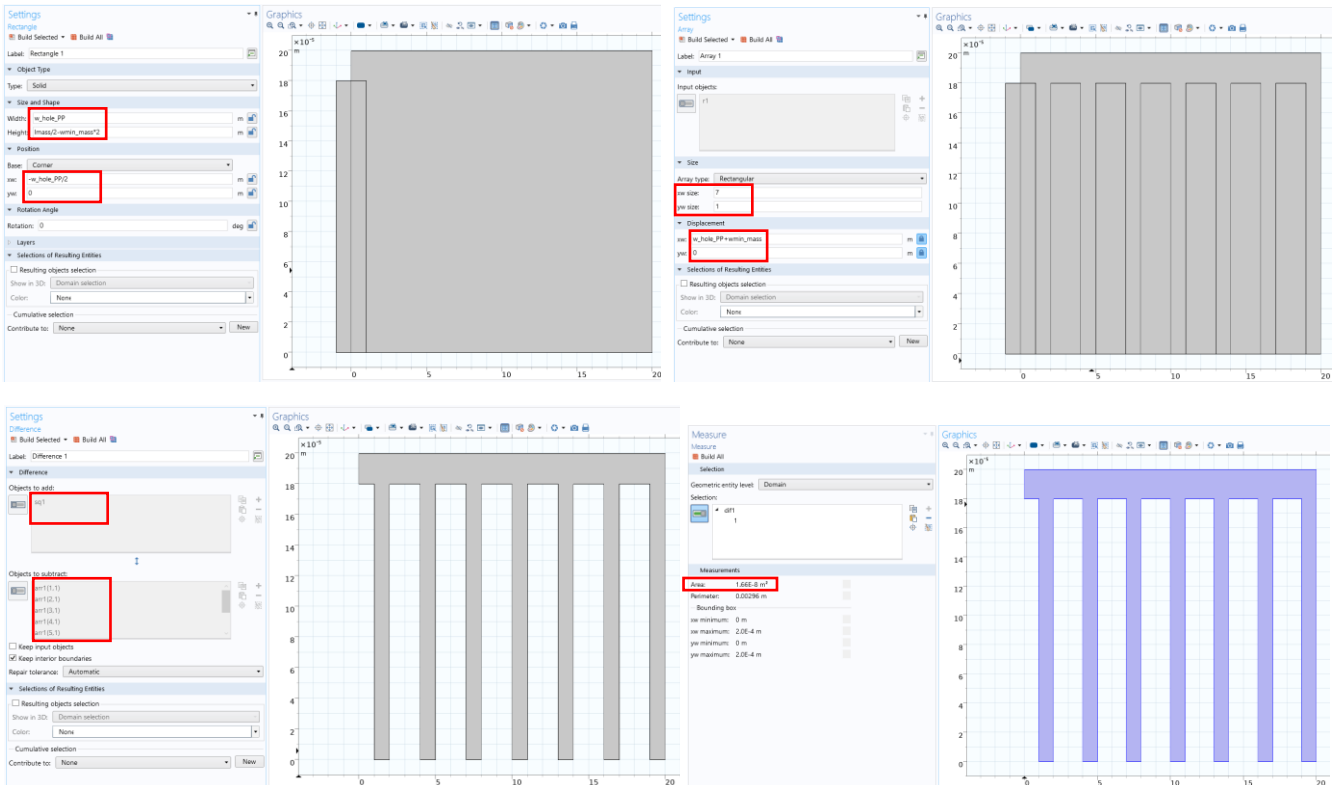
Then, we can design the first “quarter” of the geometry: as MEMS structures are often symmetric along one or two axes, designing halves or quarters of a structure, and then mirroring them, minimizes the design efforts and the risks of symmetry errors (critical e.g. for quadrature in gyroscopes, as we will see...).

In the “Model Builder” window → Right click on “Plane Geometry” → in the pop-up window select “**Square**” → in the “Setting” window of the created square, type “mass” in the “Label”. As the overall length of the mass is “l<sub>mass</sub>”, and as we are designing the upper right quarter, we have to type “l<sub>mass</sub>/2” in “Side length”. Then, press “Build Selected”: the square, with the proper dimension appears in the “Graphic” window.



At this point, we need to draw the slots used to accommodate nested parallel plates. To do so, we use the “subtraction” between different shapes. In other words, we design the rectangles corresponding to the holes, and then we make a subtraction between the square shape and all the rectangles.

This can be obtained by (i) designing a first rectangle (right click on “plane geometry” → “rectangle”), (ii) creating an array of such rectangles (right click on “plane geometry” → “Transform” → “Array”), and (iii) making the subtraction (right click on “plane geometry” → “Boolean” → “Difference”). The figures below highlight the 3-step procedure. Note: a similar approach could be used to design the release holes of the suspended mass. We avoid doing this for easier computational complexity.



At this point, we can evaluate the area of a quarter of the suspended mass, in order to find out the value of the mass  $m$  itself. You can get the area of a shape by right clicking on “Plane geometry” → “Measure” → choose “Domain” in “Geometric entity level”. The area of  $\frac{1}{4}$  of the mass is given as  $1.66 \cdot 10^{-8} m^2$ .

Now that we know the area, we can easily calculate the mass, and thus, given the resonance frequency, the required stiffness. Given the required stiffness, we can calculate the width of the folds. The equations to find these values are reported in the following. The found dimensions can be added in the parameters list.

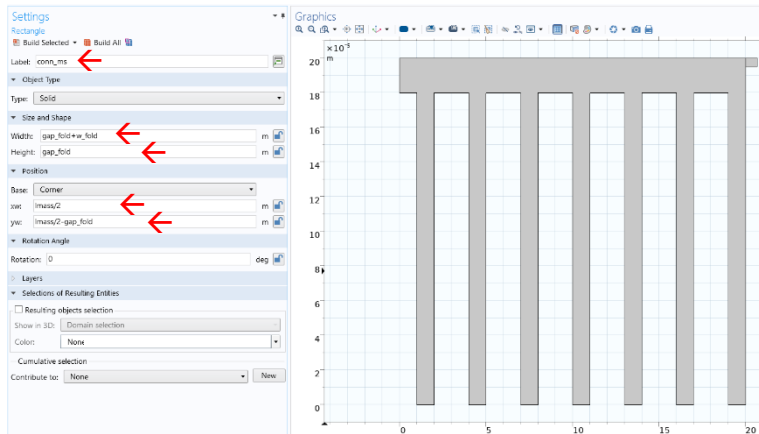
$$m = A \cdot \rho \cdot h = 1.66 \cdot 10^{-8} \text{ m}^2 \cdot 2350 \frac{\text{kg}}{\text{m}^3} \cdot 20 \mu\text{m} = 3.12 \text{ nk g}$$

$$k = \omega_0^2 m = (2\pi f_0)^2 m = 2 \frac{N}{m}$$

$$E \left( \frac{w_{sp}}{L_{sp}} \right)^3 h \frac{N_{sp}}{N_{fold}} = 2 \frac{N}{m} \rightarrow w_{sp} = L_{sp} \sqrt[3]{2 \frac{N}{m} \frac{1}{E h}} = 1.58 \mu\text{m}$$

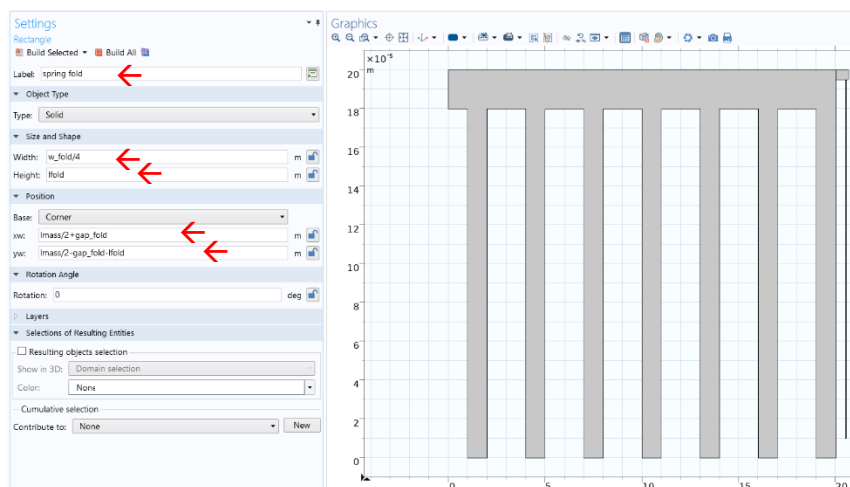
| Name      | Expression                                | Value     | Description               |
|-----------|---|-----------|---------------------------|
| E         | 160e9                                     | 1.6E11    | Young's modulus           |
| thickness | 20e-6                                     | 2E-5      | epitaxial layer thickness |
| lmass     | 400e-6                                    | 4E-4      | mass side                 |
| lfold     | 185e-6                                    | 1.85E-4   | spring fold length        |
| nfold     | 4   | 4         | number of folds           |
| nspring   | 4   | 4         | number of springs         |
| gap_fold  | 5e-6                                      | 5E-6      | gap between folds         |
| wmin_mass | 10e-6                                     | 1E-5      | minimum mass width        |
| w_hole_PP | 20e-6                                     | 2E-5      | width of holes for PP     |
| A_quarter | 1.66E-8                                   | 1.66E-8   | area of a quarter         |
| rho       | 2350                                      | 2350      | silicon density           |
| m         | A_quarter*4*thickness*rho                 | 3.1208E-9 | proof mass                |
| l0        | 4000                                      | 4000      |                           |
| k         | (2*pi*l0)^2*m                             | 1.9713    | stiffness                 |
| w_sp      | lfold*(k/E/thickness*nfold/nspring)^(1/3) | 1.5741E-6 | spring width              |

Then, we have to design the first spring. As you know, length, width, thickness, and number of folds define the geometry of a spring. However, real springs are formed also by rigid (short) connections (i) to the suspended mass, (ii) to the anchor points and (iii) between folds. We start by designing the first of such short connections.



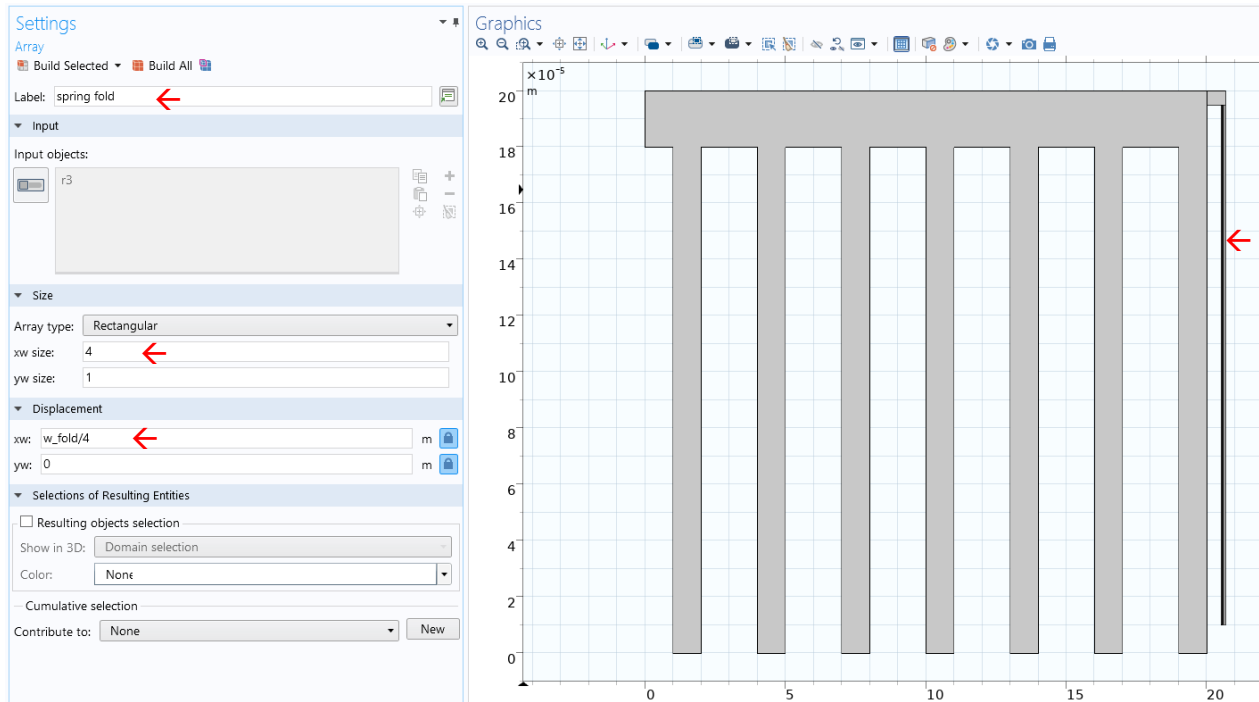
In the “Model Builder” window → Right click on “Plane Geometry” → in the pop up window select “Rectangle” → in the “Setting” window of the created square type “conn\_ms” (connector mass-spring) in the “Label”. Type “gap\_fold+w\_fold” in “Width” → type “gap\_fold” in “Height” → type “lmass/2” as xw coordinate in “Position” → type “lmass/2-gap\_fold” as yw coordinate in “Position”. Then, click on “Build Selected” (or press “F7” on keyboard)

For a better simulation of parts that are subject to consistent deformations (e.g. the springs), it is recommended to have at least three points of Mesh (the software uses these points for the calculation) in the direction of the largest displacement. For this reason, it is useful to split each spring into an **array** of 4 identical sub-elements.



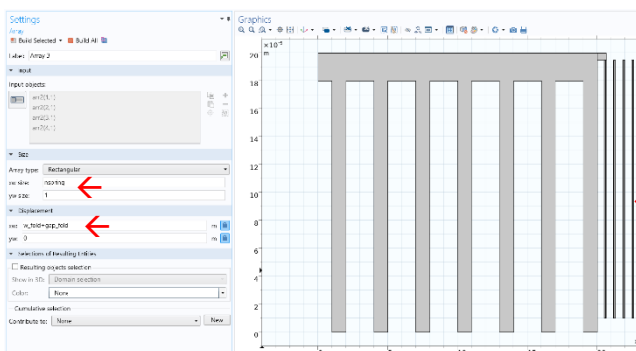
In the “Model Builder” window → Right click on “Plane Geometry” → in the pop-up window select “Rectangle” → in the “Setting” window type “spring fold” in the “Label” → type “w\_fold/4” in “Width” → type “lfold” in “Height” → type “lmass/2+gap\_fold” as xw coordinate in “Position” → type “lmass/2-gap\_fold-lfold” as yw coordinate in “Position”. Then, click on “Build Selected”.

In the “Model Builder” window → Right click on “Plane Geometry”, → in the pop up window select “Transforms”, → in the pop up window select “Array”. The “Setting” window for the “Array” is now open. In the “Graphic” window, *left click* on the first quarter of the spring (it will become colored in light blue and “r3” will appear in the “Input object” window) → type “4” as *xw* size in the “Size” tab → type “w\_fold/4” as *xw* in the “Displacement”. Press F7 (or “Build all”). Four adjacent rectangles will now form our first spring fold.

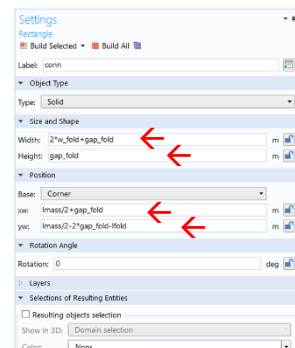


Now, we can create the other three folds, again with the command “Array”:

in the “Model Builder” window → Right click on “Plane Geometry” → in the pop up window select “Transforms” → in the pop up window select “Array” → In the “Setting” window change the “Input objects” into “fold1” → type “nspring” in *xw* size → type “w\_fold+gap\_fold” as *xw* in the “Displacement”. Press F7.



We now need the three connectors between the folds: in the “Model Builder” window → Right click on “Plane Geometry” → in the pop up window select “Rectangle” → in the “Setting” window type “conn” in the “Label”. Type “2\*w\_fold+gap\_fold” in “Width” → type “gap\_fold” in “Height” → type “lmass/2+gap\_fold” as *xw* coordinate in “Position” → type “lmass/2-2\*gap\_fold” as *yw* coordinate in “Position”. Press F7.

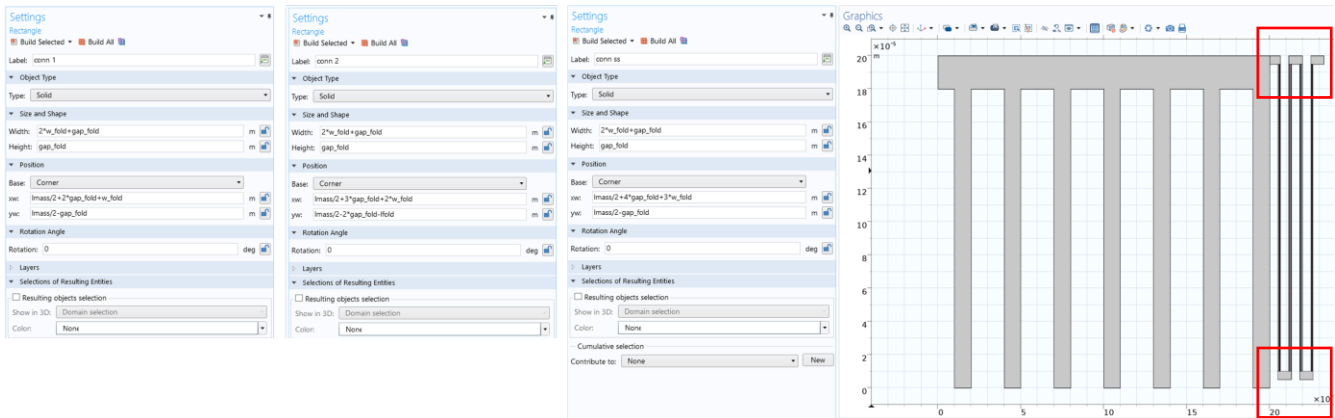


Now we create the other connectors (between the spring folds and from the spring to the substrate) with the command “duplicate”, as indicated below.

Right click on “conn” → select “Duplicate” in the pop up → change the xw “Position” in “lmass/2+3\*gap\_fold+2\*w\_fold”. Press F7.

Right click on “conn” → select “Duplicate” in the pop up → change the xw “Position” in “lmass/2+2\*gap\_fold+w\_fold” → change the yw “Position” in “lmass/2-gap\_fold”. Press F7.

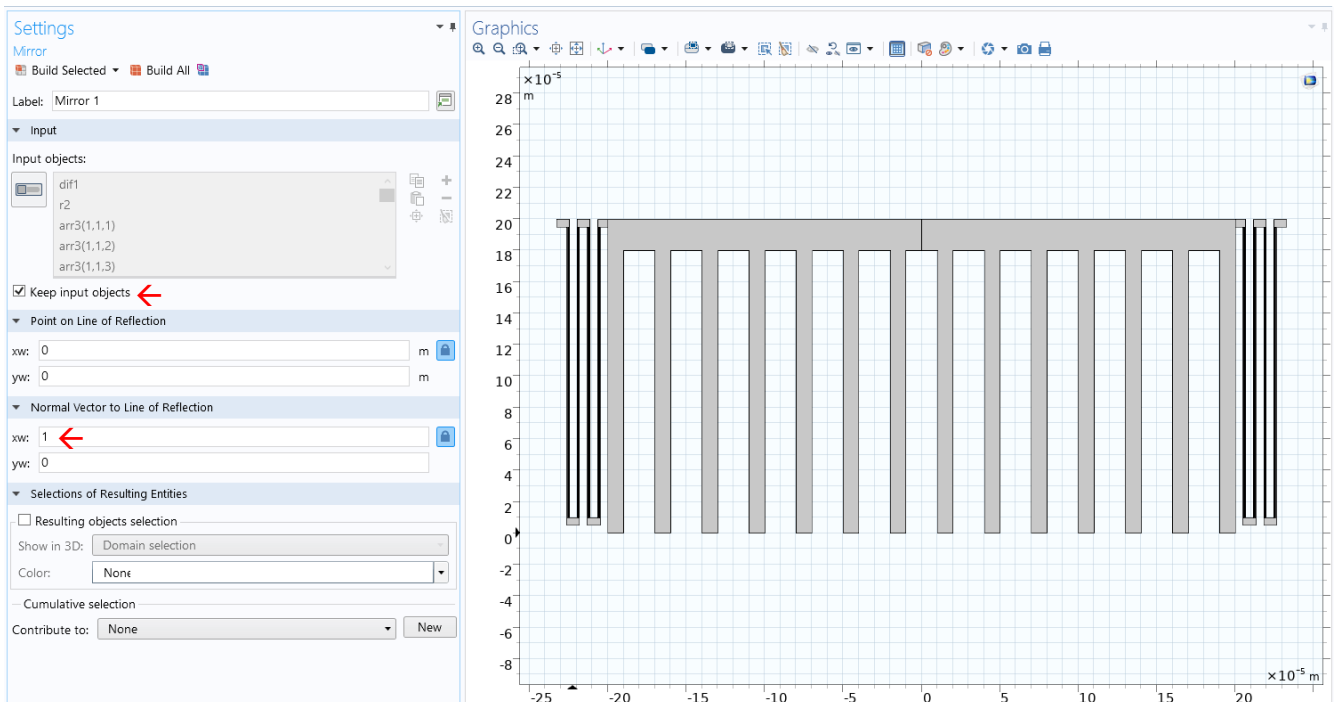
Right click on “conn\_ms” → select “Duplicate” in the pop up → type “conn\_ss” in “Label” (connector spring-substrate) → change the xw “Position” in “lmass/2+4\*gap\_fold+3\*w\_fold”. Press F7.



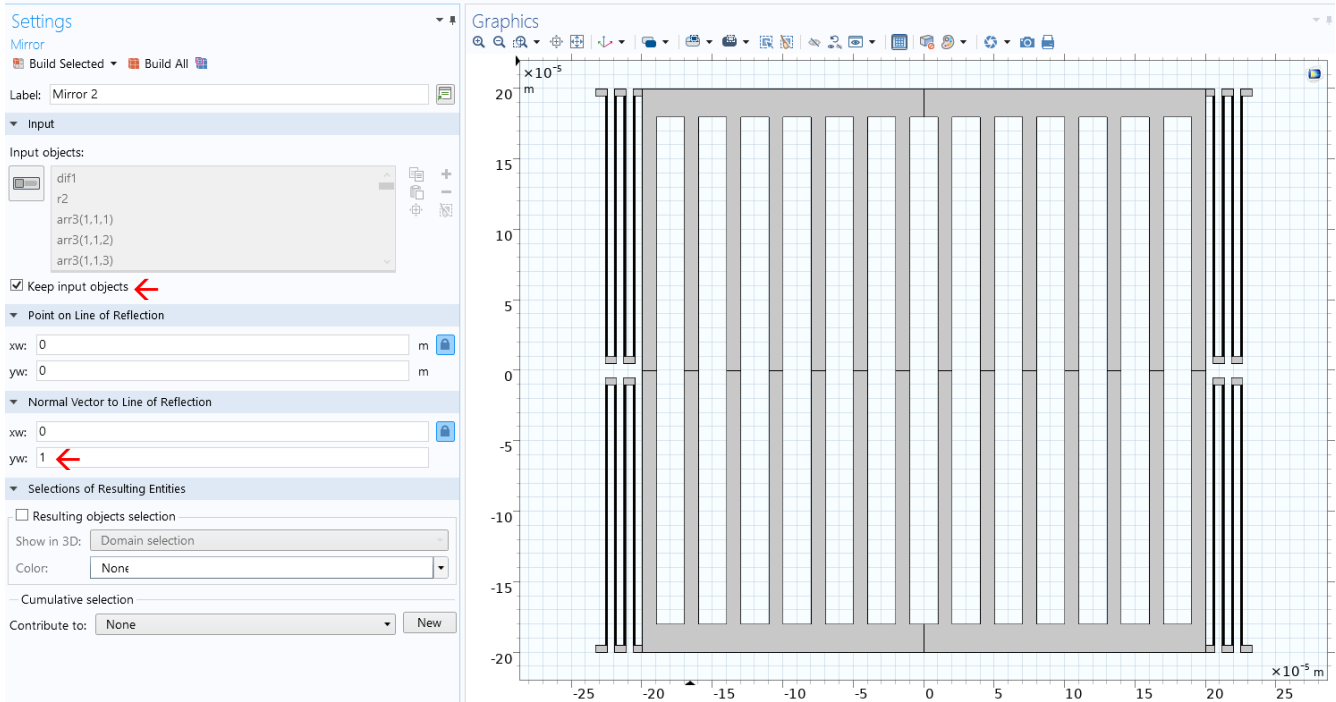
Now we can **mirror** the geometry (twice), so to finally obtain the complete 2D device.

Right click on “Plane Geometry” → “Transforms” → “Mirror”. Left click on “Graphic” sheet → press “CTRL+A” to select all the domains → select the box “Keep input objects”. Press F7.

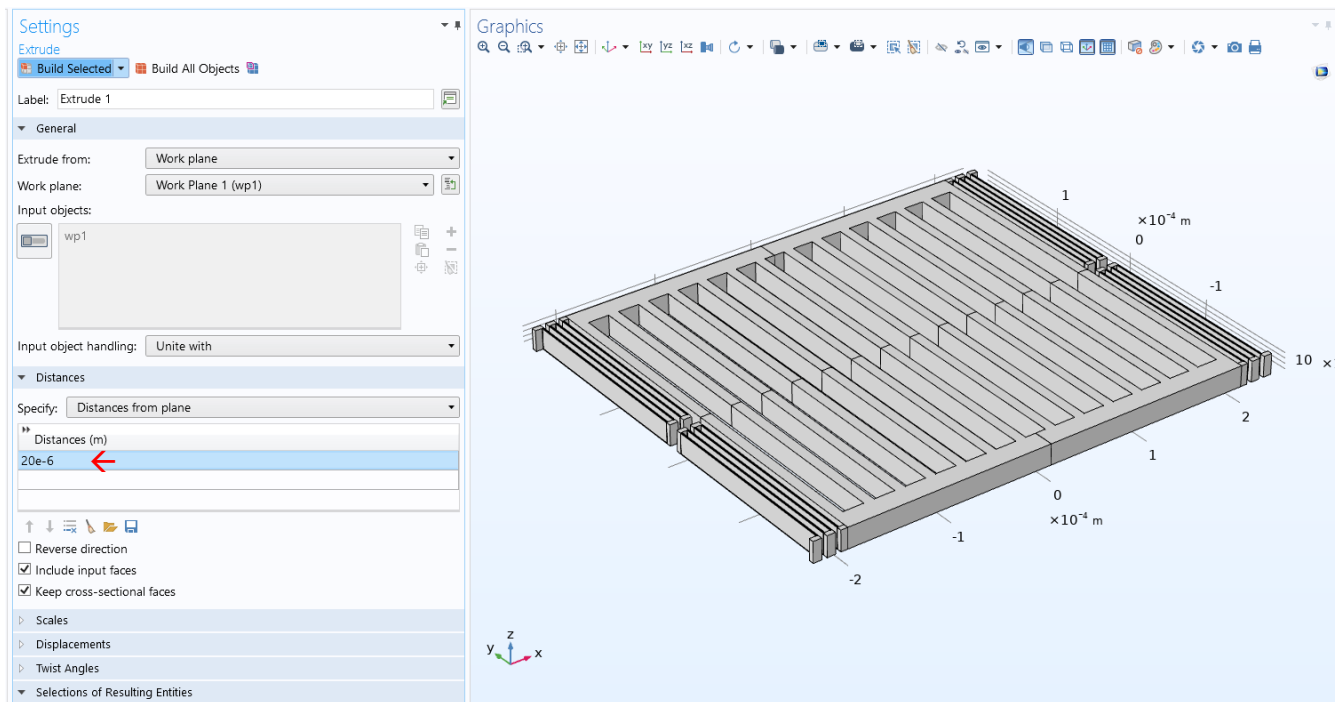
To fit the designed geometry into the “Graphic” window we can press the “Zoom Extents” (shown by the green arrow in the figure).



Right click on “Plane Geometry” → “Transforms” → “Mirror”. Left click on “Graphic” sheet → press “CTRL+A” to select all the domain → type “0” as  $x_w$  and “1” as  $y_w$  in the “Normal Vector to Line of Reflection” → select the box “Keep input objects”. Press F7. This concludes the definition of the in-plane geometry. Please save the file.



Then, we have to **extrude** it in the out-of-plane direction, by a dimension corresponding to the epitaxial layer thickness. Right click on “Work Plane” → select “Extrude”. In the “Settings” window type “thickness” as “Distances” (instead of the 20  $\mu\text{m}$  shown here). Press F7. We obtain the final 3D geometry as reported below.

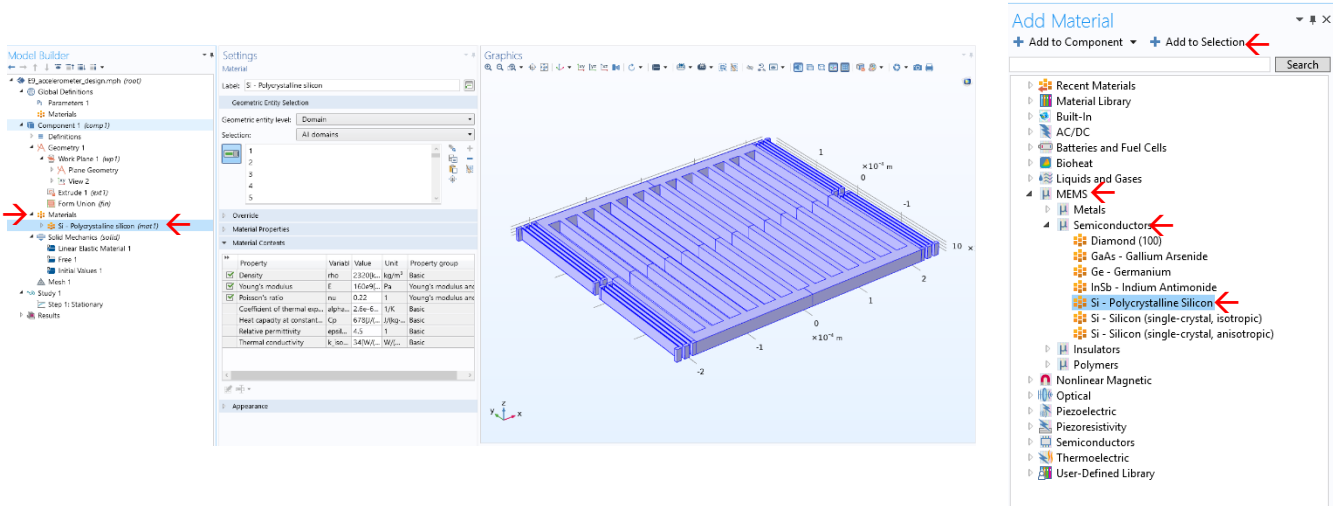




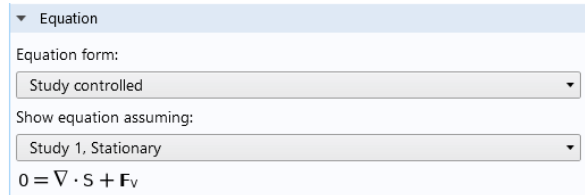
## 2. Boundary Settings

The next step consists in defining the **material** and the **boundary conditions** for the solution of our equations. The procedure is similar to what we saw in the last CAD class.

Right click on “Materials” → click on “Add Material”. A new window appears. In the “Graphic” window, left click on the four quarter of the mass (they become blue). Then, expand “MEMS” → expand “Semiconductors” → click on “Si – Polycrystalline Silicon” → click on “Add Selection”.



Then, we can add the **boundary conditions**. Keep in mind that the solver always manages partial differential equations for the entire number of finite elements. If you want to have an insight into the equations, you have just to expand the corresponding section as shown aside: the equation indicates that the divergence of stress (S) equals the volume force ( $F_V$ ).

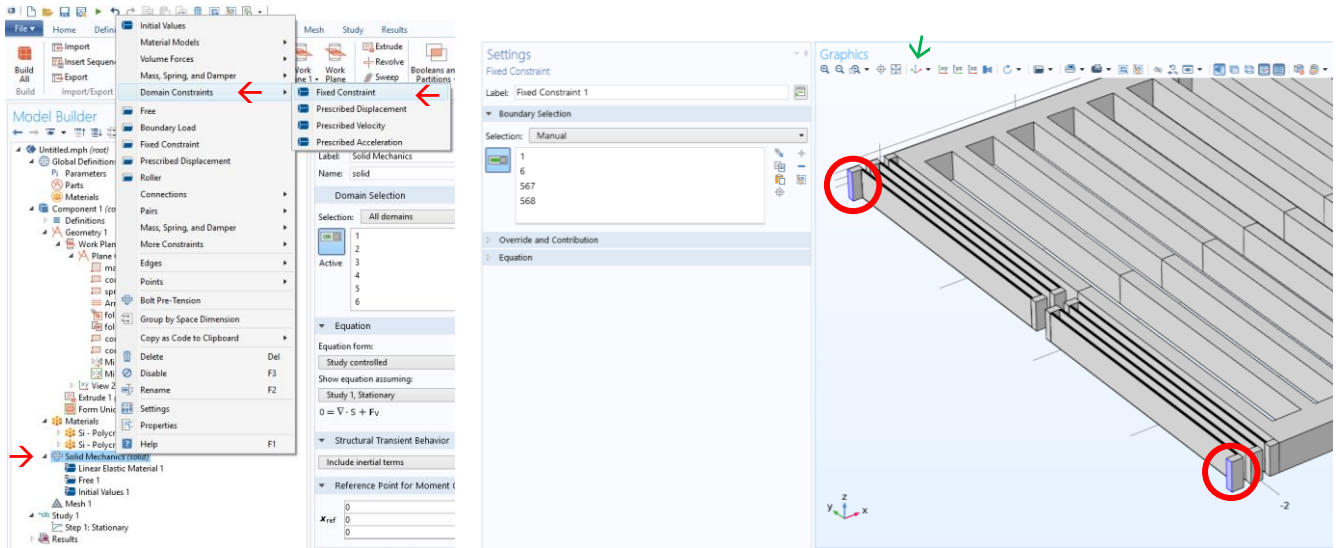


This is a vectorial view of the generalized Hooke's law which relates stress and strain (see Lecture n. 6).

To add the boundary conditions, right click on “solid Mechanics” → choose “Domain Constraints” → click on “**fixed Constraint**”. This imposes a condition of no motion for the elements that we select. We choose the four domains at the spring edges, by clicking on them in the “Graphics” window. Note that this corresponds to a Dirichlet boundary condition as we are, in turn, setting no stress S for the selected domains of our geometry in the equation above.

To help you easily selecting the various subdomains, here are some tips:

- 1) you can rotate the geometry by holding the “left click” of your mouse;
- 2) you can move the geometry by holding the “right click” of your mouse;
- 3) you can zoom in and zoom out by holding the “scroll click” of your mouse and moving left and right
- 4) you can reset the default geometry view by clicking the “Go to Default 3D View” button (green arrow in the figure below).

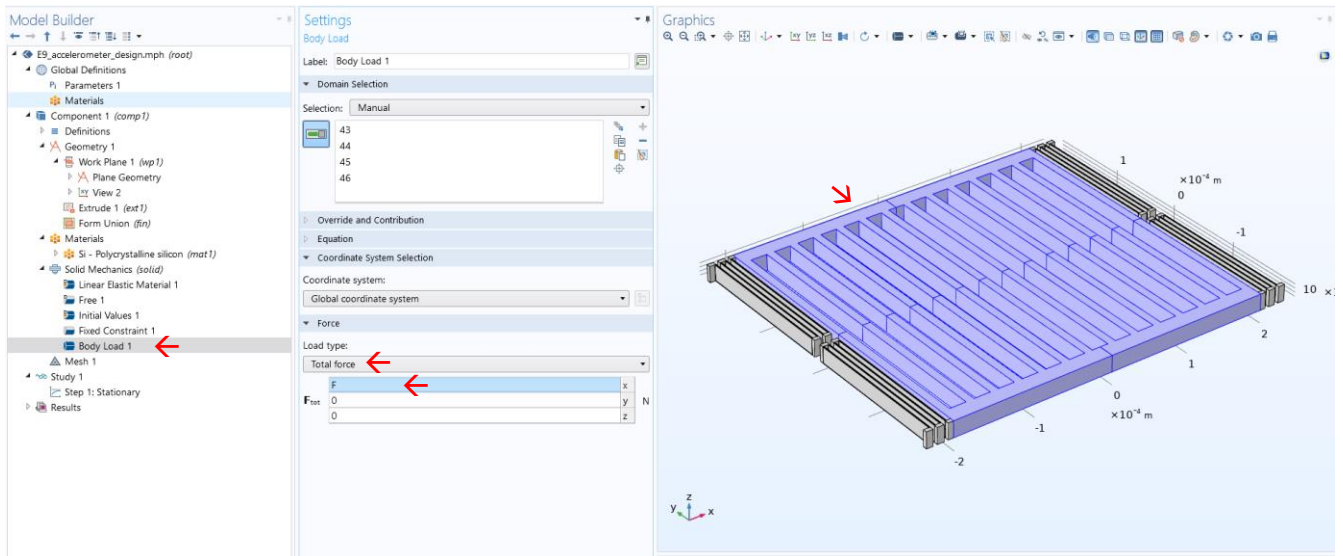


In order to evaluate the stiffness, we now **apply a force** ( $F = 1 \mu\text{N}$ , corresponding roughly to  $32 \text{ g}$  of acceleration, a typical FSR) to the mass of the accelerometer, and then we evaluate the corresponding simulated displacement. The application of a fixed volume force corresponds to another Dirichlet boundary condition.

Click on “Parameters” → type “F” in “Name” and “ $1\text{e-}6 \text{ [N]}$ ” in “Value” (you can also define an acceleration parameter ( $acc = 32 \text{ g}$ ), and then define the force as  $m*acc$ ).

| Name      | Expression            | Value     | Description               |
|-----------|-----------------------|-----------|---------------------------|
| E         | 160e9                 | 1.6E11    | Young's modulus           |
| thickness | 20e-6                 | 2E-5      | epitaxial layer thickness |
| lmass     | 400e-6                | 4E-4      | mass side                 |
| lfold     | 185e-6                | 1.85E-4   | spring fold length        |
| nfold     | 4                     | 4         | number of folds           |
| npring    | 4                     | 4         | number of springs         |
| gap_fold  | 5e-6                  | 5E-6      | gap between folds         |
| wmin_mass | 10e-6                 | 1E-5      | minimum mass width        |
| w_hole_PP | 20e-6                 | 2E-5      | width of holes for PP     |
| A_quarter | 1.66E-8               | 1.66E-8   | area of a quarter         |
| rho       | 2350                  | 2350      | silicon density           |
| m         | A_quarter*4*thickness | 3.1208E-9 | proof mass                |
| fd        | 4000                  | 4000      |                           |
| k         | (2*pi*fd)^2*m         | 1.9713    | stiffness                 |
| w_fold    | lfold*nk/f/thickness  | 1.5741E-6 | spring width              |
| F         | 1e-6                  | 1E-6      | applied force             |

Right click on “Solid Mechanics” → select “Volume Forces” → click on “Body Load”. Change “Load type” into “Total force” → type “F” in the x component of  $F_{tot}$  → in the “Graphic” window click on the four quarters.

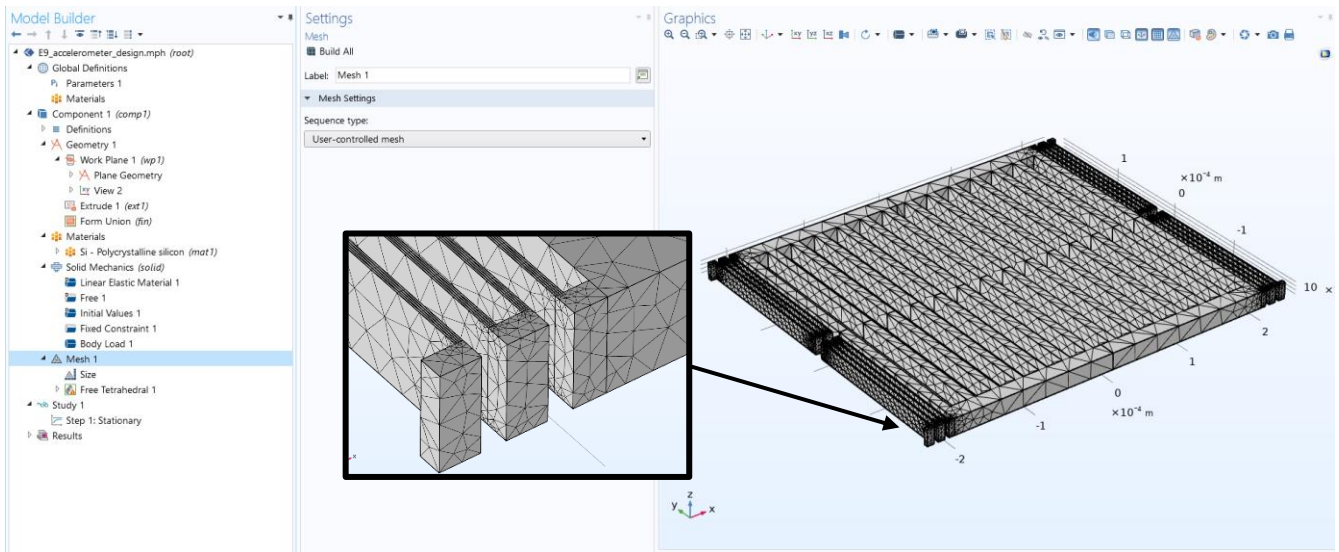


### 3. Mesh

Now, we have to define the points where the software calculates the solutions of the mechanical problem, i.e. the mesh (or grid).

Also for 3D geometries, there are a lot of different possibilities to create a “**Mesh**” (which we will not deepen as this goes beyond a course of microsensors).

The easiest 3D mesh type is the “Free Tetrahedral” (the domain is split into tetrahedral objects): right click on “Mesh” → click on “**Free Tetrahedral**”. Press F7 (note that some warnings appear: Comsol advises us that there are regions with few points of mesh. For our purpose, the number of mesh points is anyway enough. These warnings, or sometimes errors, are often the nightmare of a young MEMS designer).

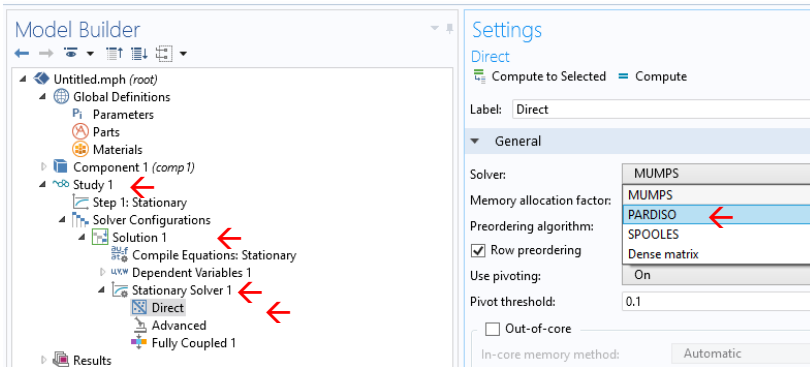


Note also that the number of points to be solved grows very quickly when designing complex 3D structures. Therefore, you may want to lower the number of mesh elements. To do so, click on “Mesh” → “Size” → in the “predefined” selection change from “Normal” to “Coarse” or “Coarser”. You will see that the quality of your mesh decreases, which will also lower the accuracy of your solution... but the solution time itself will be faster.

To avoid that the solution accuracy drops too much, one strategy is to force at least a minimum number of points along those portions of the device that will be deformed by the force. That is why we built the springs with four sub-elements: as you see in the zoom, even with a relatively coarse mesh, we keep a large number of points along the springs.

## 4. Study and results (stationary analysis)

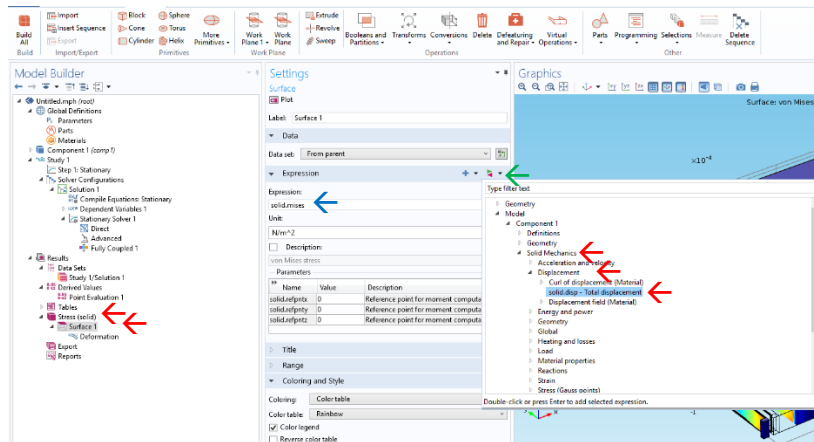
The procedure begins exactly in the same manner as already done in the past class. This is quickly summarized by the figure below.



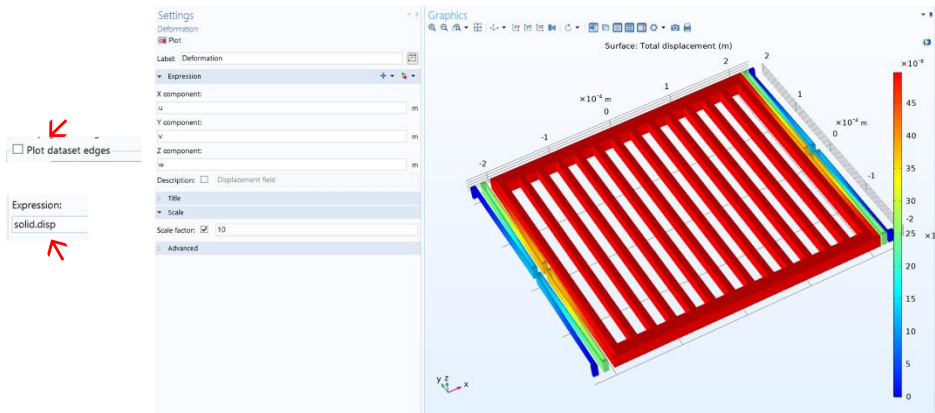
Right click on “Study”  
 → click on “Show Default Solver”  
 → expand “Solution 1”  
 → expand “Stationary Solver 1”  
 → click on “Direct”  
 → in “Setting” window  
 → change the “Solver” into PARDISO.  
 This solver uses the symmetry of the problem to decrease the computational time. Click on “Compute” (or press F8).

Comsol starts with the iterative solution of the equation for all the finite elements until the simulation converges, and the result appears in the “Graphic” window. The plot of the surface stress appears by default. As we are interested in the displacement, we can change it as follows:

- 1) expand “Stress” → click on “Surface 1” → in the “Settings” window click on “Replace Expression” button (green arrow in the figure) → expand “Solid Mechanics” → expand “Displacement” → double click on “solid. disp” → click on “Plot”...
- 2) or simply type “solid.disp” instead of “solid.mises” in the field “Expression” (blue arrow) → click on “Plot”



Note: to improve the visual quality of the results, you may want to unflag “plot data set edges” in the main “Stress” window, and change the deformation factor in the “Deformation” window (setting a value of 1 to the latter would put shown displacements to scale, but would also make the motion hard to understand...).



To evaluate the overall stiffness, either we can manually find the displacement (click on a point of the visual solution and look at the table below), and then calculate the stiffness, or we can use Comsol for this purpose. Let us follow the second path.

Under “Results” right click on Derived Values → click on “Point evaluation” → in the “Graphic” window click on one point of the mass (e.g. close to the center) → type “F/solid.disp” in the “Expression” field → click on “Evaluate” (or press F8).

The screenshot displays the COMSOL Multiphysics interface. On the left, the 'Point Evaluation 1' dialog is open, showing the expression 'F/solid.disp' and the unit '1/m'. The main window shows a 3D model of a spring-mass system with a red arrow indicating the point of evaluation. The bottom pane shows a table with the result '2.0081'.

| Expression   | Unit | Description |
|--------------|------|-------------|
| F/solid.disp | 1/m  |             |

| Name           | Value | Unit | Description                                    |
|----------------|-------|------|--|
| solid.refpn... | 0     | m    | Reference point for moment computation, x c... |
| solid.refpn... | 0     | m    | Reference point for moment computation, y c... |
| solid.refpn... | 0     | m    | Reference point for moment computation, z c... |

| Message                        |
|--------------------------------|
| F/solid.disp (1/m), Point: 286 |
| 2.0081                         |

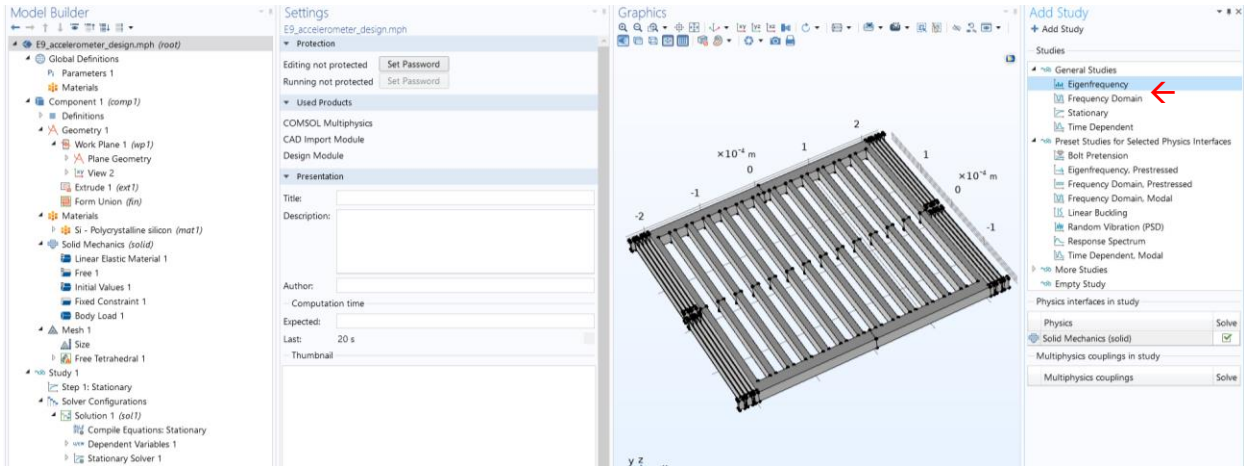
A new **Table** appears with the result, indicating a stiffness of 2 N/m.

The result very well matches our theoretical prediction, as the geometry of this problem is relatively simple. However, in other devices with more complex geometries, a FEM analysis is typically more reliable than simple theoretical calculations, and thus mandatory.

## 5. Study and results (eigenvalue analysis)

Now we want to evaluate the characteristic resonance modes of this device. We remind here that the expected resonance frequency of the first mode is about 4 kHz. We are interested in verifying this number, and in checking the presence of high-order modes and whether they are close or far from the mode of interest.

There is no need to create a new file, but rather we can add different studies in the same Comsol file.

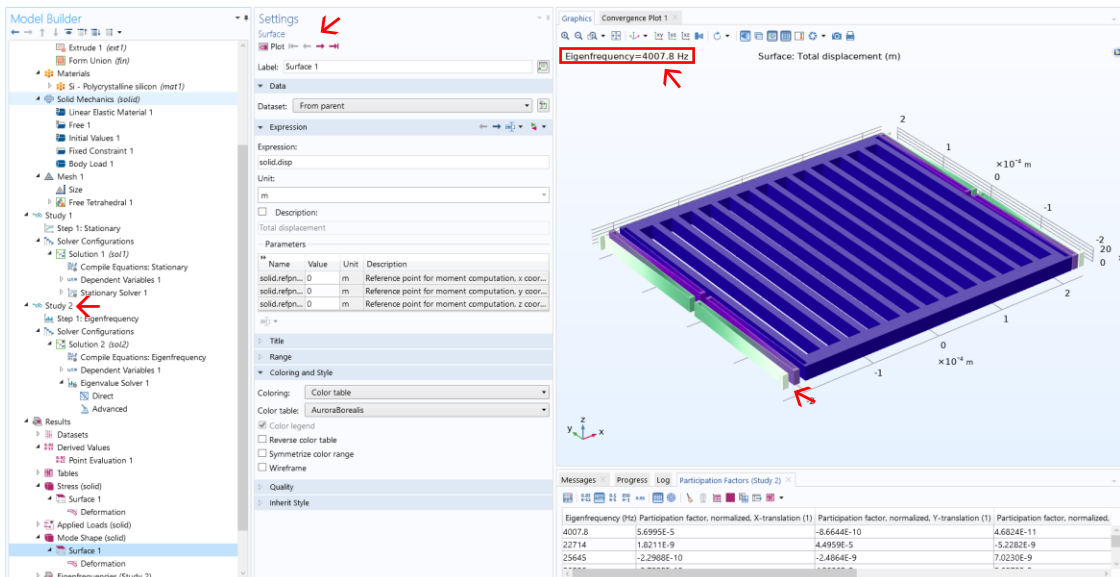


Right click on “Untitled.mph” (file name) → click on “Add Study” → in the new window click “Eigenfrequency” → click “Add study” → “Study 2” appears. Note that the change of the study also changes the equation that the solver considers. When solving for eigenvalues, external forces are ignored, and the 3D generalized spring mass-damper system is considered in absence of damping (eigenvalue solution, see lecture n. 3).

Study 2, Eigenfrequency

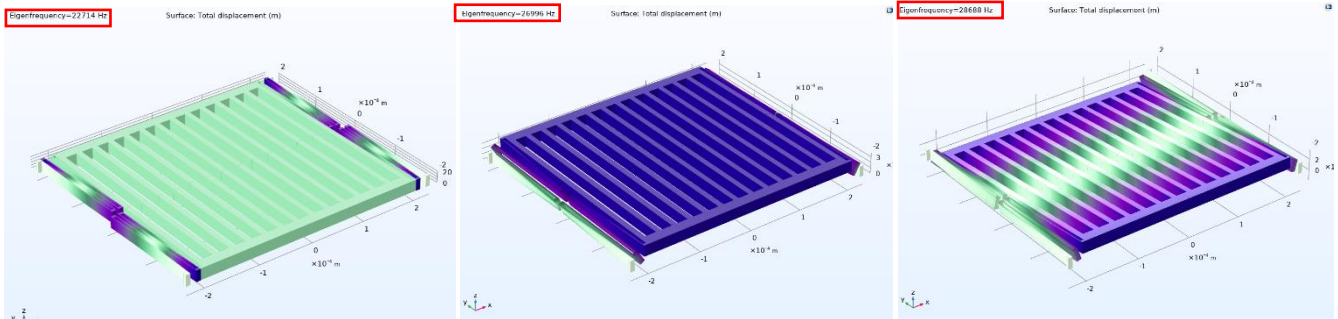
$$-\rho \omega^2 \mathbf{u} = \nabla \cdot \mathbf{S}, \quad -i\omega = \lambda$$

To start the solution, we can set the “PARDISO” solver as already done for the stationary solution: right click on “Study 2” → click on “Show Default Solver” → expand “Study Configurations” → expand “Eigenvalue solver” → change the solver in “PARDISO”. Press F8 (or the “Compute” button) to start the simulation.



The simulation calculates the first six modes of the device. The first mode, as expected, represents a translation along the in-plane direction, and has a resonance frequency of 4.007 kHz, very close to the theoretical value of 4 kHz. You can also appreciate how the springs behave as guided-end folds.

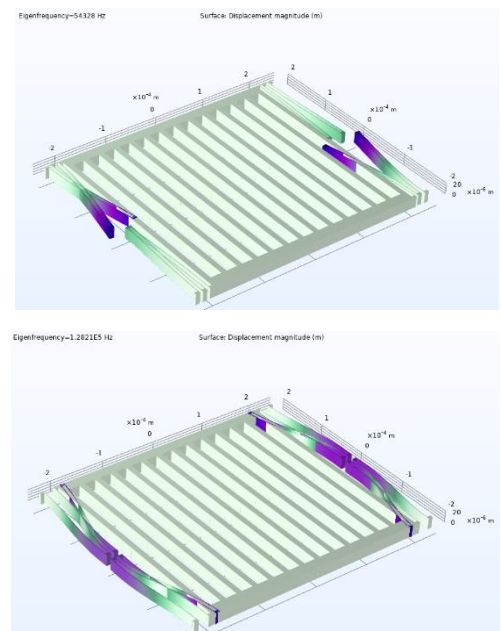
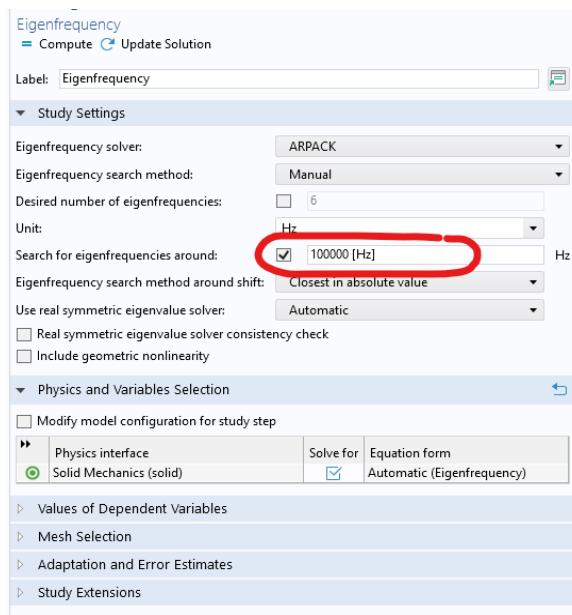
On top of the setting window, by moving through the purple left/right arrows, you can go on to the next modes. In this specific case, you will find some resonance modes of the springs at frequencies around 22 kHz, the out-of-plane translational mode and the out-of-plane torsional mode at frequencies of 27 kHz and 29 kHz, respectively. The conclusion is that these modes are far enough from the mode of interest and will not perturb the expected behavior of our in-plane accelerometer.



Note: as a rule of thumb, we consider high-order modes to be far enough from the mode of interest if they lie at least 50% far from the frequency  $f_0$ : for a value of 4 kHz, all high-order modes shall be at least beyond 6 kHz, and this condition is obtained with a very large margin in this example.

Finally, assuming that you are reading out your accelerometer with a 100 kHz modulated signal, you would like to check whether there are specific modes around that frequency. To do so, go to the Eigenfrequency tab, and tell the simulator to look for frequencies around that value.

Interestingly, we find a mode around 54 kHz, and the following one lies at 128 kHz. This means that we have a wide window where we can position our sinewave frequency without risks of exciting spurious modes.



## 6. Try by yourself now...

Try to design an out-of-plane accelerometer in order to sense z-axis accelerations with the same resonant frequency (for the mode related to the acceleration to be sense). Try to occupy a reasonable area and to keep spring critical dimension ( $w_{\text{fold}}$ ) roughly similar to the in-plane accelerometer that you have just analyzed.

Once you have done it, try to solve for both the accelerometers (in-plane and out-of-plane) a parametric analysis of the resonance frequency as a function of the process thickness, and compare the results to the numerical exercise n. 3!