Spring Semester 2024

# Topic 2 - MICRO 373 COMSOL SIMULATION GUIDEBOOK

MICRO – 373 Advanced microfabrication practicals

CHENXIANG ZHANG, POL TORRES-VILA, DR. ARNAUD BERTSCH

**Contacts:**

chenxiang.zhang@epfl.ch

pol.torresvila@epfl.ch

arnaud.bertsch@epfl.ch

# Finite Element Analysis of beams

## 1 Introduction

### 1.1 TP objectives

In this part, you will get along and discover the FEM (finite element modelling) software COMSOL MULTIPHYSICS® v6.1 by simulating a copper beam and a MEMS (micro-electro-mechanical system) thermal actuator that you will fabricate in the cleanroom.

### 1.2 Introduction of COMSOL MULTIPHYSICS

The basic steps of simulating in COMSOL include:

- Definition of the geometry model. The COMSOL provides 3D, 2D symmetric, 2D, 1D symmetric, 1D, and 0D models. The appropriate choice of dimension can simplify the calculation while remaining accuracy.
  You can draw your model with geometry elements provided in the software, or import your model from other CAD (computer assistant designing) software such as SOLIDWORKS or 3DSMAX. (The importing procedure is not included in this guideline. If you are interested, you can find guidelines in Analyze Your SOLIDWORKS® Designs Using COMSOL Multiphysics®)
- Set parameters and materials for your model. For the ease of parameter sweeping, you can define variables for some of your parameters both in geometry and physics.
  To complete a model, you should also declare the material mostly. You can choose the pre-defined materials with essential properties for the physics in the material library of COMSOL, or define your own material by filling in the blanks of parameters required by physics.
- Definition of physical model. In the COMSOL, you can work with only one or several different physical fields, including mechanical, electrical, thermal, and others that you can imagine. The couple of different physical fields are also done by COMSOL. To well describe the model, you need to set initial physical status, physical models, and excitations. It's the most important but also the trickiest work. You must be experienced to construct a suitable model for calculation.
- Setting of study steps. The COMSOL provide you with not only basic study such as stationary, time dependent, and frequency domain, but also advanced ones such as eigenfrequency. You can choose what you need, and set sweeping parameters if you want.
- Click "Calculate" button to just start the calculation!
- Results plotting and exporting. Basically, after calculation, you can get every parameter that you care about from the results. The COMSOL provide you with all kinds of plots to show results. Choose what you believe is the best way to show your results, and check it in the software. If you want to export your results for further evaluation, it is also easy for the COMSOL.

### 1.3 Steps

In this practical session, you will have 3 models to simulate, and 1 experiment to do. The first simulation tells you the static deflection of a macro copper beam under the effect of a certain force. The second simulation gives you the resonance frequencies. After these two simulations, you will have an experiment of the copper beam the same as your simulation model, to compare the results between the simulation and the experiment.

All the steps above are the preparation for the final step, by getting yourself familiar with the simulation and COMSOL software. The final step is to simulate one of the fabricated MEMS beams. The simulation of your own design is highly recommended. The simulation results then will help you with the designing and the test of the MEMS beam.

# 2 Preparation

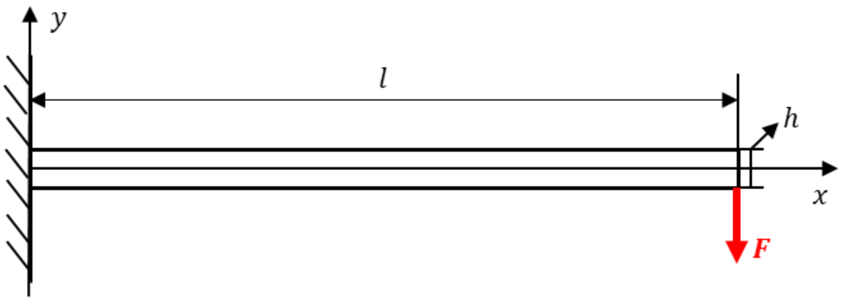
During this preparation, you will perform a theoretical prediction of the behaviour of the macro copper beam that you will study during the session.

The theoretical prediction, is to let you perform calculations on a simplified model, with reasonable assumptions. Making theoretical predictions is essential for assessing the validity and performance of the model used during the simulation. You will also have a rough idea of how the parameters during the designing influence the performance.

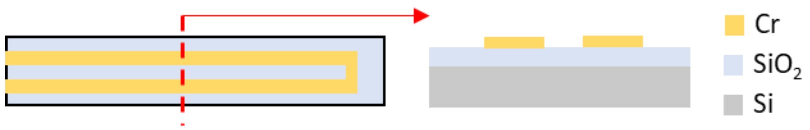
Apart from the theoretical calculation of the copper beam, to perform the simulation on your MEMS cantilever, please bring the calculation about the inertia moment, spring constant, and eigenfrequencies you made previously to the session.

## 2.1 Structure and parameters

Both the copper beam and the MEMS cantilever are embedded and fixed at $x = 0$. For the copper beam, a vertical force of $F = 50mN$ is dynamically applied at $x = l$, which $l$ is the length of the beam.




The MEMS beam is composed of a $SiO_2$ layer and a Cr track on top. The cross section of the MEMS beam is shown in the figure below.




The beam dimensions and characteristics are as follows. These numbers are used only in the preparation. In the TP session, please use the number you measured (for the copper beam) or designed (for the MEMS beam).

| | Copper beam | MEMS beam |
|---|---|---|
| Beam length $l$ | $l = 180mm$ | $l = 500\mu m$ |
| Beam height $h$ | $h = 1mm$ | $h_{SiO_2} = 1.5\mu m$<br>$h_{Cr} = 0.5\mu m$ |
| Beam width $w$ | $w = 20mm$ | $w_{SiO_2} = 80\mu m$<br>$w_{Cr} = 2 \times 20\mu m$ |
| Young's modulus $E$ | $E = 110GPa$ | $E_{SiO_2} = 70GPa$ |

| | | |
|---|---|---|
| | | $E_{Cr} = 140GPa$ |
| Density $\rho$ | $\rho = 8.7g/cm^3$ | $\rho_{SiO_2} = 2.2g/cm^3$<br>$\rho_{Cr} = 7.15g/cm^3$ |

## 2.2 Questions for preparation

Answer these questions before the TP session. The TA will check your answers before the session.

### 2.2.1 Question 1

Give the deflection $y_{max}$ of the copper beam with a force $F = 50mN$ applied to its end (static deflection; neglect gravity). You can use as reference https://mechanicalc.com/reference/beam-analysis.

### 2.2.2 Question 2

Plot (with any software that you are familiar with) the deflection $y(x)$ of the copper beam. You can compare this with the simulation result.

### 2.2.3 Question 3

Calculate inertia moment, spring constant and then the equation for the eigenfrequencies (resonance frequency) and compute them for the copper beam. Explain and justify approximations you did.

# 3 Questions during session

The following are the questions you need to answer in your report. These questions are the ones that you need to consider during your practical course. The other questions related to data processing and analysis can be found in the file Topic2-373_report_guidelines.pdf. Please don't forget to answer all the questions in your report.

## 3.1 Copper beam

### 3.1.1 Static analysis

- Plot the deflection of the beam as simulated in COMSOL.

### 3.1.2 Frequency response

- Plot the measured and simulated frequency response from 10Hz – 100Hz in the same graph. What can you find from the value of the resonance frequency and amplitude at resonance? Note: concentrate the measurements around the resonant frequencies.

## 3.2 MEMS beam

- Sketch the graphs of the mechanical deflection and temperature of the end point in z at frequencies 1Hz and 100Hz. Customize the stepping time accordingly to the frequency. Comment on the results and predict behaviour at 10kHz.
- Determine the eigenfrequency of the system and report corresponding shapes with screenshots.

# 4 Copper beam

## 4.1 Simulations

You may see some minor interface changes because the description corresponds to version 6.1.

### 4.1.1 Static analysis

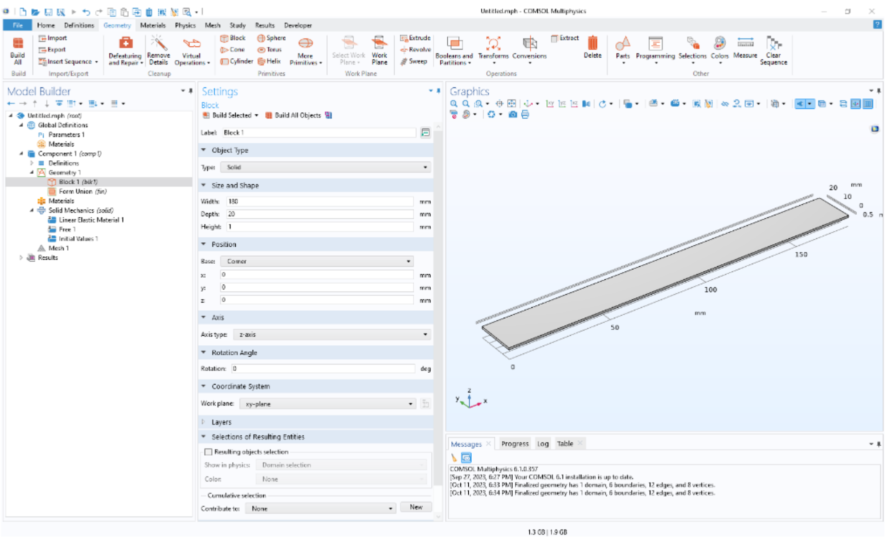
In this part, you will construct a 3D model of a copper beam, and apply a vertical force on its end point. By simulation, you will know the displacement of the copper beam, and research on the relationship between stress and strain.

- ❖ From the **File** menu, choose **New**
- ❖ **NEW**

  In the **New** window, click **Model Wizard**
- ❖ **MODEL WIZARD**
  - ➢ In the **Model Wizard** window, click **3D**
  - ➢ In the **Select Physics** tree, select **Structural Mechanics > Solid Mechanics (solid)**
  - ➢ Click **add**
  - ➢ Click **Study**
  - ➢ In the **Select Study** tree, select **General Studies > Stationary**
  - ➢ Click **Done**
- ❖ **GEOMETRY 1**
  - ➢ Click **Model Builder > Geometry 1**
  - ➢ In the **Settings** window for **Geometry 1**, locate the **Units** section.
  - ➢ Find the **Length unit** sub selection, and choose **mm**.

*Block 1 (blk1)*

- ➢ In the **Geometry** toolbar, click **Primitives > Block**.
- ➢ In the **Settings** window for **Block 1**, locate the **Size and Shape** section.
- ➢ In the **Width**, **Depth**, and **Height** text fields, type the copper beam dimension.
- ➢ In the **Geometry** toolbar, click **Build All**.

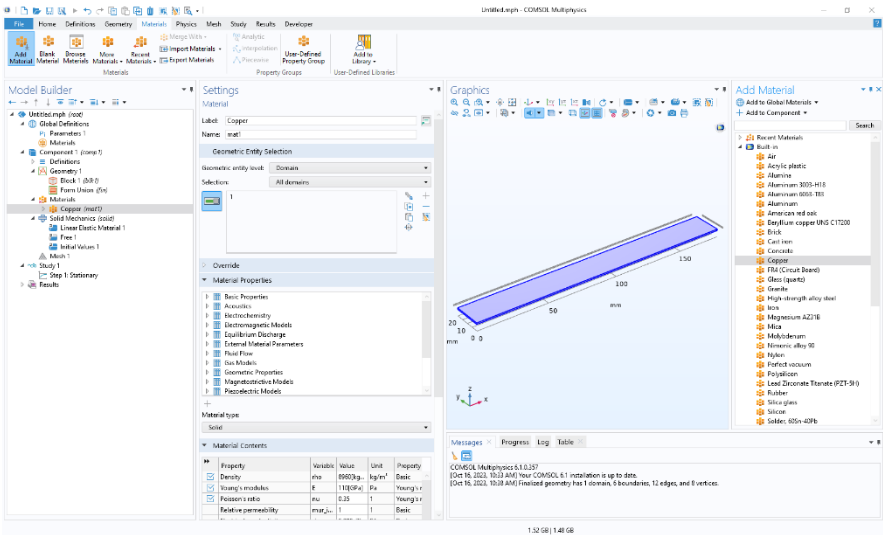


- ❖ **MATERIALS**

*Copper (mat1)*

- ➢ In the **Materials** toolbar, click **Add Material**

- In the **Add Material** window, select **Built-in > Copper**
- Click **Add to Component**
- In the **Settings** window for **Copper**, make sure that the beam is chosen in the **Geometric Entity Selection**
- Close the **Add Material** window

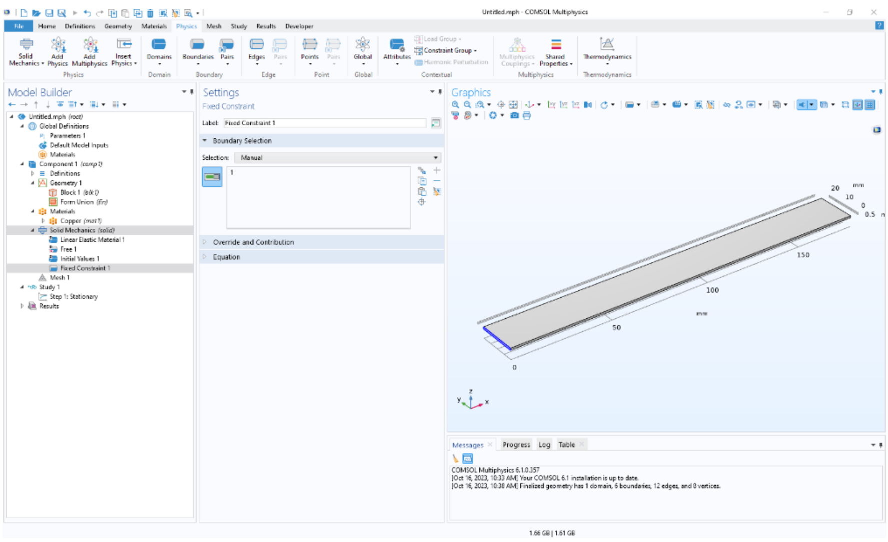


## ❖ SOLID MECHANICS (SOLID)

- Left-click the **Solid Mechanics (solid)** in the **Model Builder**

*Fixed Constraint 1*

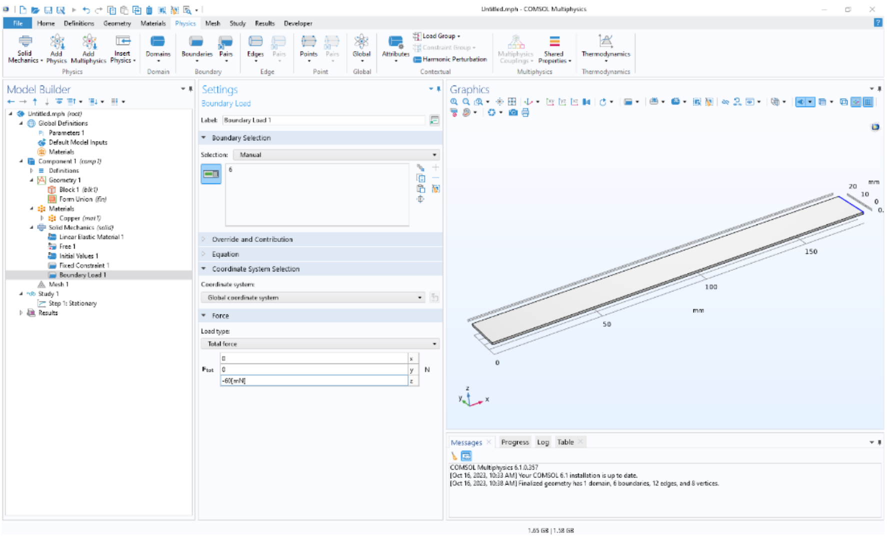
- In the **Physics** toolbar, select **Boundaries > Fixed Constraint**
- In the **Settings** window for **Fixed Constraint 1**, locate **Boundary Selection**
- In the **Graphics** window, choose one surface at the end of the beam by left-click



*Boundary Load 1*

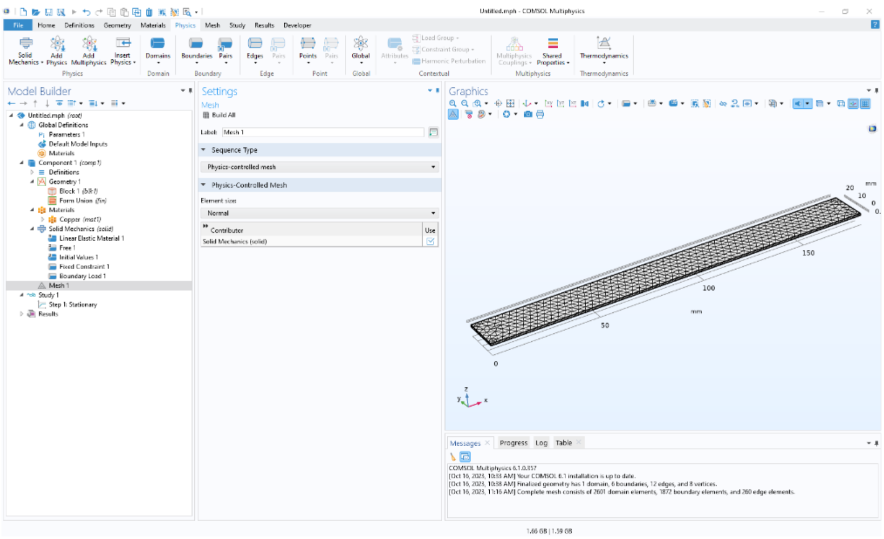
- In the **Physics** toolbar, select **Boundaries > Boundary Load**
- In the **Settings** window for **Boundary Load 1**, locate **Boundary Selection**
- In the **Graphics** window, choose the surface at another end of the beam by left-click
- In the **Load type** selection list of the **Force** tab, choose **Total force**

- In the **z** text field of $\mathbf{F}_{tot}$, enter the value of the force and the unit in between [] (notice the direction of the force)



❖ **MESH 1**

- In the **Settings** window of **Mesh 1**, check that **Physics-controlled mesh** is chosen in **Sequency Type** selection list, and **Normal** is chosen in **Physics-Controlled Mesh > Element size** selection list
- Click **Build All**



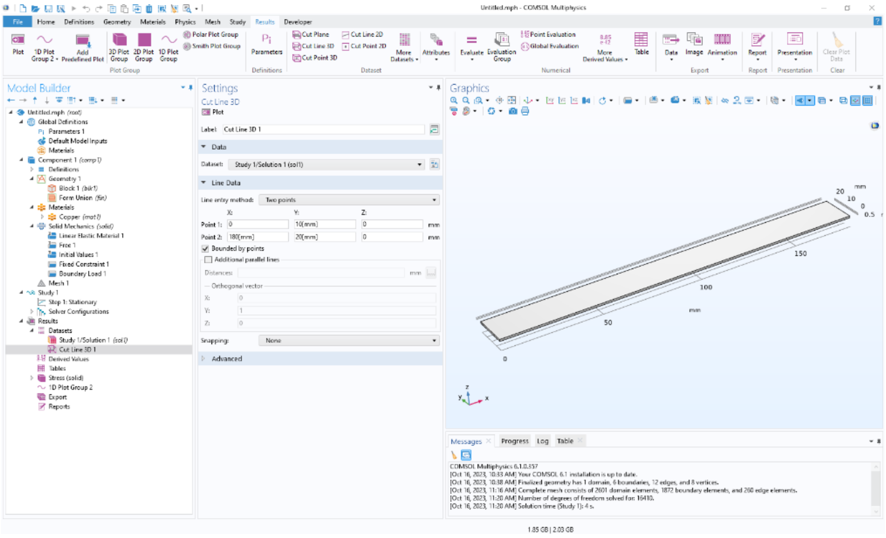
❖ **STUDY 1**

- Choose **Study 1** in the **Model Builder**, click **Compute** in the **Settings** window

❖ **RESULTS**

*Cut Line 3D 1*

- In the **Results** toolbar, click **Dataset > Cut Line 3D**
- In the **Settings** window of **Cut Line 3D 1**, locate the **Line Data** section
- Enter the coordinates of two points to create a cut line along the beam in the **X**, **Y**, and **Z** text fields of **Pont 1** and **Point 2**. For example, **Point 1 (0, Width / 2 [mm], 0)** and **Point 2 (Length [mm], Width / 2 [mm], 0)**, replace **Width** and **Length** by the values



*1D Plot Group 2*

- In the **Results** toolbar, click **Plot Group > 1D Plot Group**
- In the **1D Plot Group 2** toolbar, click **Add Plot > Line Graph**
- In the **Settings** window of **Line Graph 1**, locate the **Data > Dataset** choosing list, and select **Cut Line 3D 1**
- Locate the **y-Axis Data > Expression** text field, enter (or choose from **Replace Expression**) **solid.disp**
- Click **Plot**

*Export*

- In the **Results** toolbar, click **Export > Data > Plot**
- In the **Settings** window of **Plot 1**, locate **Plot > Plot group** choosing list, and choose **1D Plot Group 2**
- Locate **Output > Filename**, select the place and filename with **Browse** for your exported file
- Click **Export**
- Use the exported data to answer the question in section 3.1.1

### 4.1.2 Frequency Response Analysis

In this part, you will construct a 2D model of the same copper beam as we simulated in the static analysis. The using of 2D model is to simplify the calculations. You will apply a harmonic force with frequency excitation between 10Hz and 1kHz to determine the frequency response of this copper beam, to find a rough range of the eigenfrequency of it.

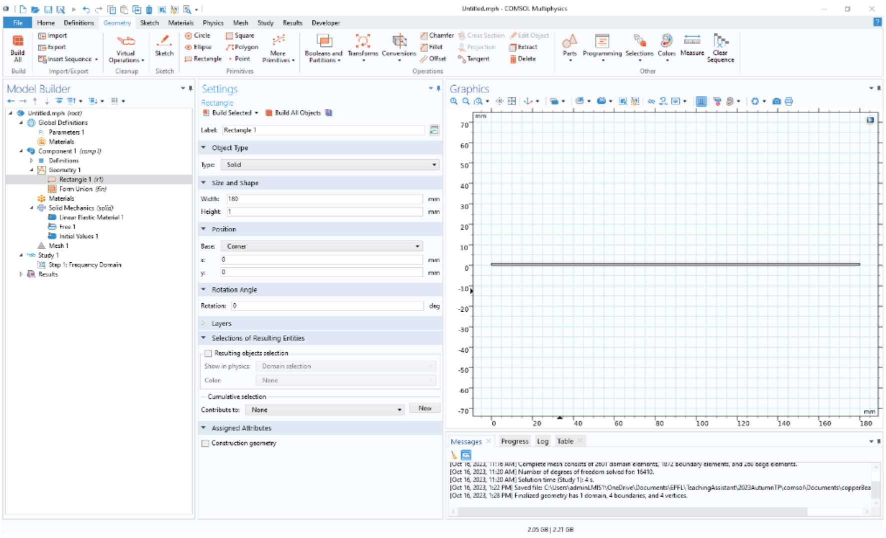
- From the **File** menu, choose **New**
- **NEW**

  In the **New** window, click **Model Wizard**
- **MODEL WIZARD**
  - In the **Model Wizard** window, click **3D**
  - In the **Select Physics** tree, select **Structural Mechanics > Solid Mechanics (solid)**
  - Click **add**
  - Click **Study**
  - In the **Select Study** tree, select **General Studies > Frequency Domain**
  - Click **Done**
- **GEOMETRY 1**

- Click **Model Builder > Geometry 1**
- In the **Settings** window for **Geometry 1**, locate the **Units** section.
- Find the **Length unit** sub selection, and choose **mm**.

*Rectangle 1 (r1)*

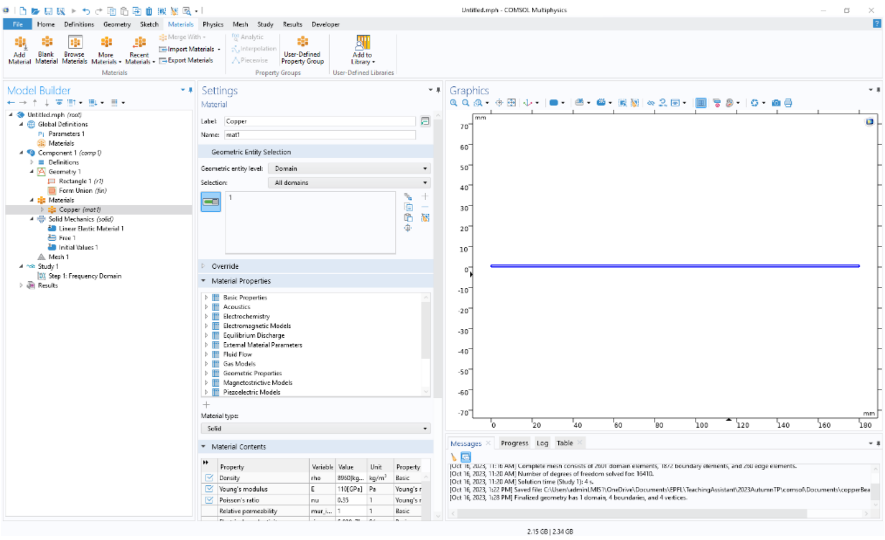
- In the **Geometry** toolbar, click **Primitives > Rectangle**
- In the **Settings** window of **Rectangle 1**, locate the sub-section **Size and Shape**
- In the **Width** and **Height** text fields, enter the length and height of the copper beam
- In the **Geometry** toolbar, click **Build All**



## ❖ MATERIALS

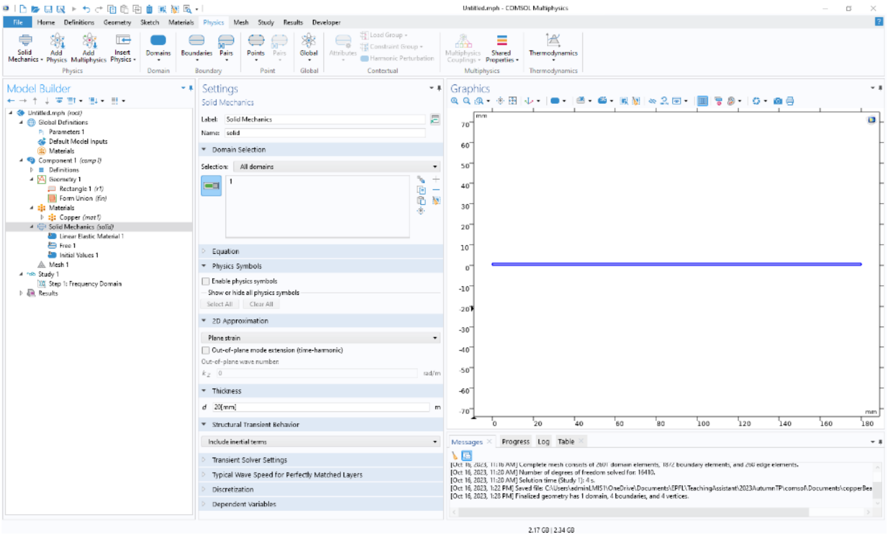
*Copper (mat1)*

- In the **Materials** toolbar, click **Add Material**
- In the **Add Material** window, select **Built-in > Copper**
- Click **Add to Component**
- In the **Settings** window for **Copper**, make sure that the beam is chosen in the **Geometric Entity Selection**
- Close the **Add Material** window



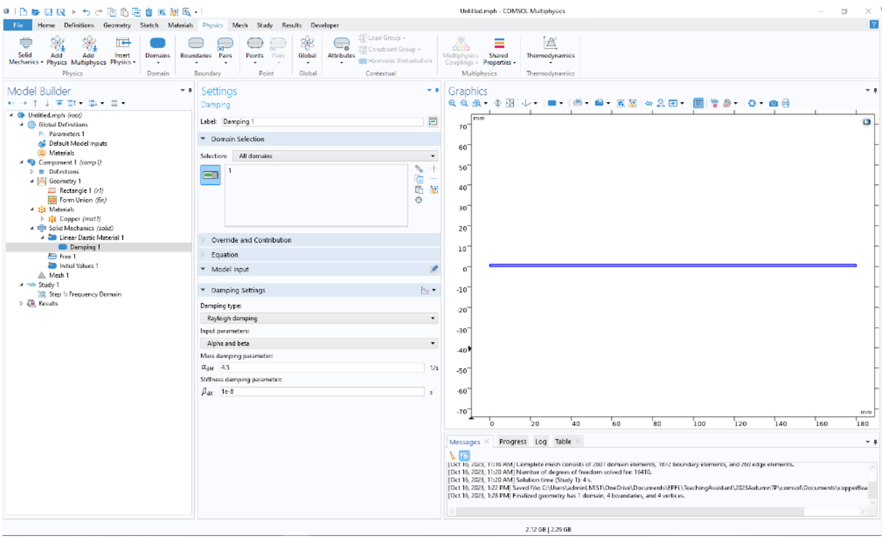
❖ **SOLID MECHANICS (SOLID)**

- Left-click the **Solid Mechanics (solid)** in the **Model Builder**
- In the **Settings** window of **Solid Mechanics**, locate **Thickness**
- In the **d** text field, enter the width of the copper beam (be careful to the unit)



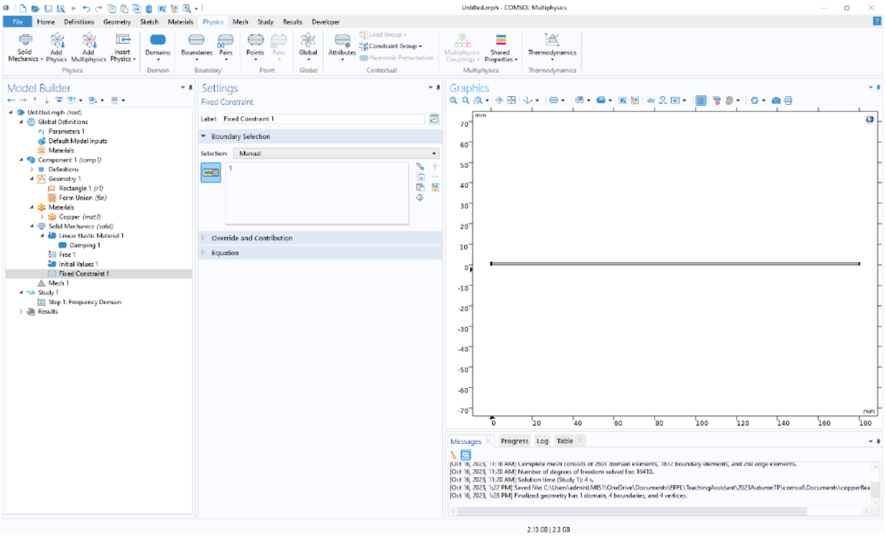
*Damping 1*

- Right click **Linear Elastic Material 1**, choose **Damping**
- In the **Settings** window, locate the **Damping Settings** section
- In the **Mass damping parameter** text field, enter **4.5**
- In the **Stiffness damping parameter** text field, enter **1e-8**



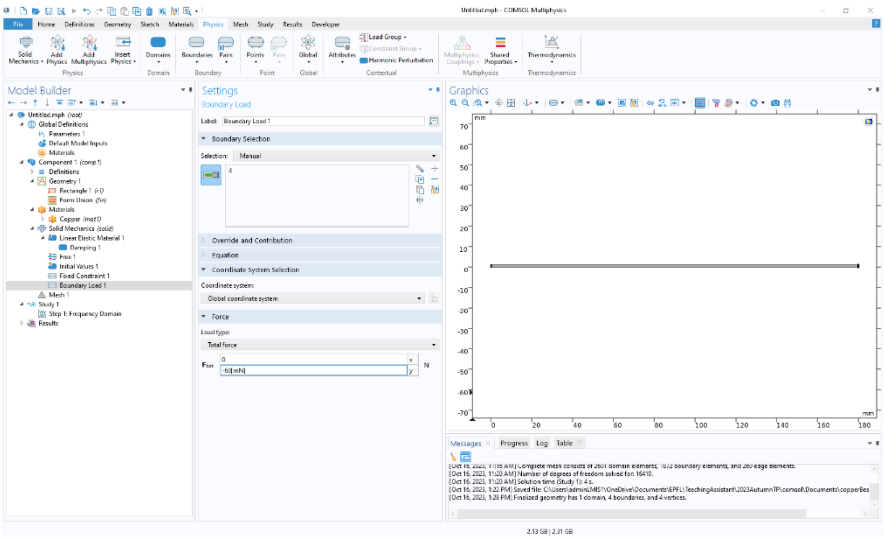
*Fixed Constraint 1*

- In the **Physics** toolbar, choose **Boundary > Boundaries > Fixed Constraint**
- In the **Settings** window for **Fixed Constraint 1**, locate **Boundary Selection**
- In the **Graphics** window, choose the boundary of one end of the beam by left-click



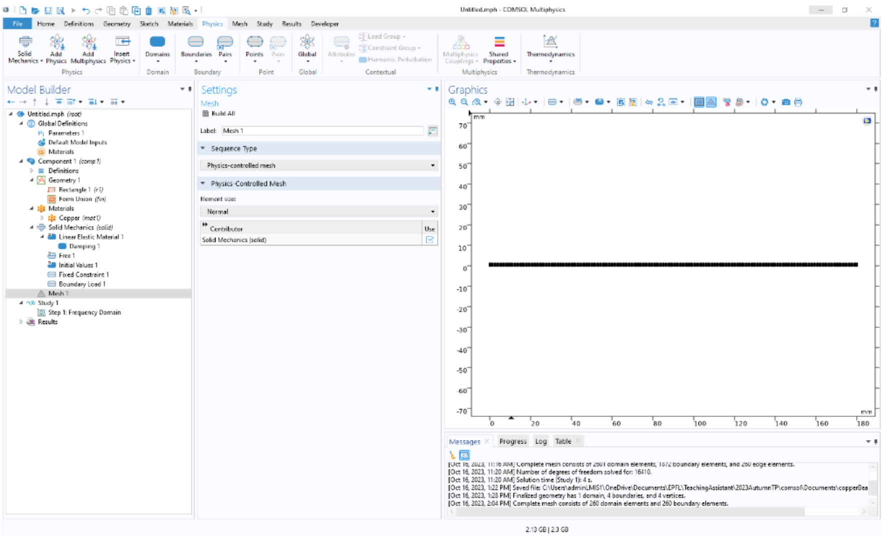
*Boundary Load 1*

- In the **Physics** toolbar, choose **Boundary > Boundaries > Boundary Load**
- In the **Settings** window for **Boundary Load 1**, locate **Boundary Selection**
- In the **Graphics** window, choose the boundary of another end of the beam by left-click
- Locate the **Force**, choose **Total force** in the **Load type** selection list
- In the **y** text field of $\mathbf{F}_{\mathbf{tot}}$, enter the value of the force and unit (notice the direction)



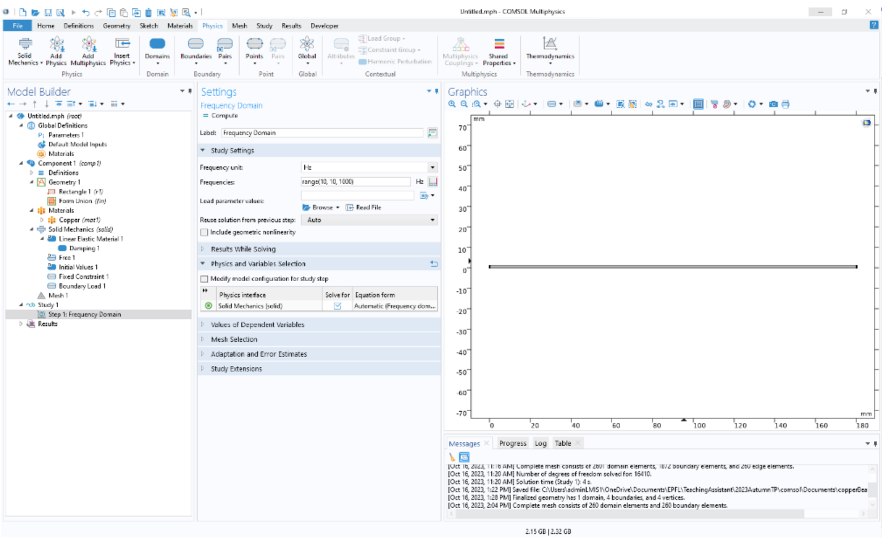
## MESH 1

- In the **Settings** window of **Mesh 1**, check that **Physics-controlled mesh** is chosen in **Sequency Type** selection list, and **Normal** is chosen in **Physics-Controlled Mesh > Element size** selection list
- Click **Build All**



## ❖ STUDY 1

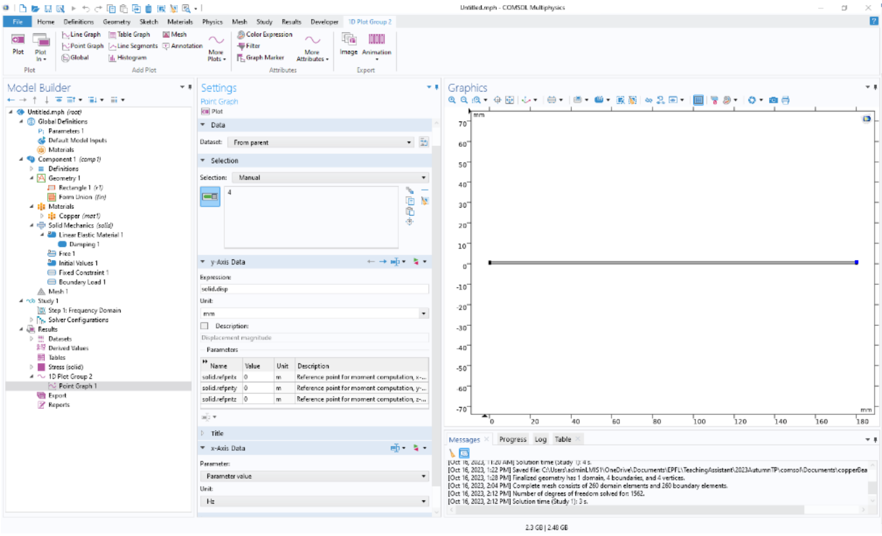
- ➢ Click **Study 1 > Step 1: Frequency Domain** in the **Model Builder**
- ➢ In the **Settings** window, locate **Study Settings**
- ➢ In the **Frequencies** text field, enter **range(10, 10, 1000)**
- ➢ Click **Compute**



## ❖ RESULTS

*1D Plot Graph 2*

- ➢ In the **Results** toolbar, click **Plot Group > 1D Plot Group**
- ➢ In the **1D Plot Group 2** toolbar, click **Point Graph**
- ➢ In the **Settings** window of **Point Graph 1**, locate **Selection**
- ➢ In the **Graphics** window, choose one point at the free end of the beam by left click
- ➢ In the **y-Axis Data > Expression** text field, enter (or choose in the **Replace Expression**) **solid.disp**
- ➢ Click **Plot**
- ➢ To change the scale to logarithm, click **1D Plot Group 2**, locate **Axis** in the **Settings** window, and click to activate **x-axis log scale**



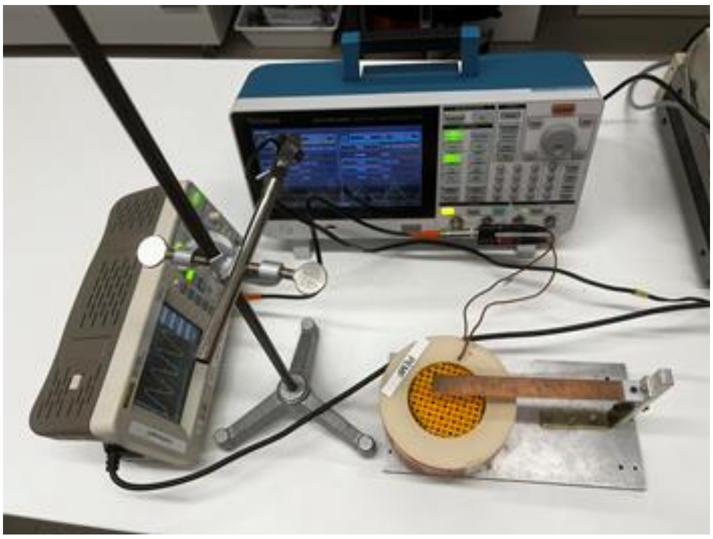
*Export*

- In the **Results** toolbar, click **Export > Data > Plot**
- In the **Settings** window of **Plot 1**, locate **Plot > Plot group** choosing list, and choose **1D Plot Group 2**
- Locate **Output > Filename**, select the place and filename with **Browse** for your exported file
- Click **Export**
- Use the exported data to answer the question in section 3.1.2

## 4.2 Experiment

The goal of the experiment here is to retrace the frequency response of the copper beam subjected to a magnetic AC excitation between 1Hz and 100Hz. Refer to the numerical simulation to identify the frequency range of interest and make a number of measurements sufficient to draw a response.

The reflection of a laser off the beam onto the wall allows for the measurement of the deflection. The trace of the laser is measured on a graph paper taped to the wall.



Use the obtained data to answer the question in section 3.1.2.

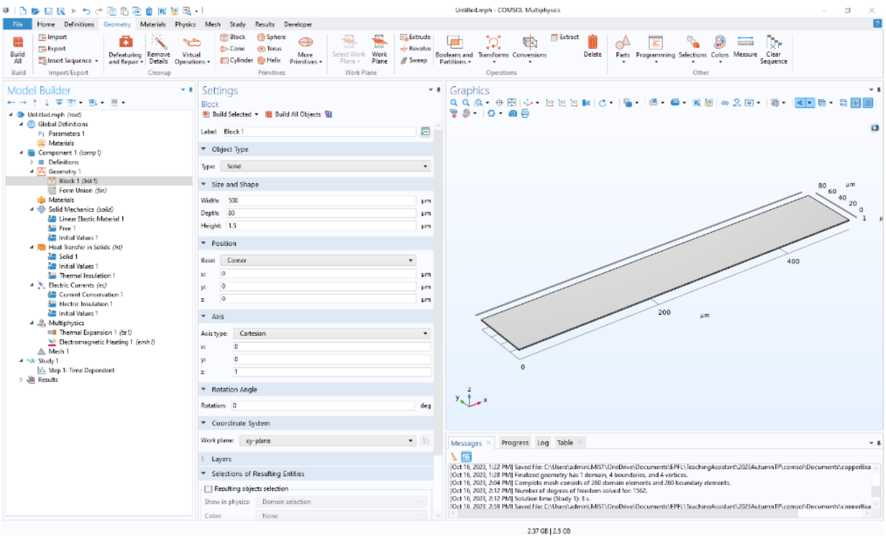
# 5 MEMS Beam

In this section, you will simulate on the MEMS thermal bimorph cantilever you will fabricate / fabricated in the cleanroom session. You will do a time dependent simulation and a eigenfrequency simulation on this model.

- From the **File** menu, choose **New**.
- **NEW**
  In the **New** window, click **Model Wizard**
- **MODEL WIZARD**
  - In the **Model Wizard** window, click **3D**.
  - In the **Select Physics** tree, select **Structural Mechanics > Thermal-Structure Interaction > Joule Heating and Thermal Expansion**
  - Click **add**
  - Click **Study**
  - In the **Select Study** tree, select **General Studies > Time Dependent**
  - Click **Done**
- **GEOMETRY 1**
  - Click **Model Builder > Geometry 1**
  - In the **Settings** window for **Geometry 1**, locate the **Units** section.
  - Find the **Length unit** sub selection, and choose **µm**.

*Block 1 (blk 1)*

- In the **Geometry** toolbar, click **Primitives > Block**
- In the **Settings** window of **Block 1**, locate **Size and Shape**
- In the **Width**, **Depth**, and **Height** text fields, enter dimensions of the beam's $SiO_2$ part
- Locate **Axis**, choose **Cartesian** in the **Axis type** selection list
- In the **z** text field of **Axis**, enter **-1**, which turns reverts the axis so that the $SiO_2$ top surface is placed at z=0
- Click **Build Selected**



*Work Plane 1 (wp1)*

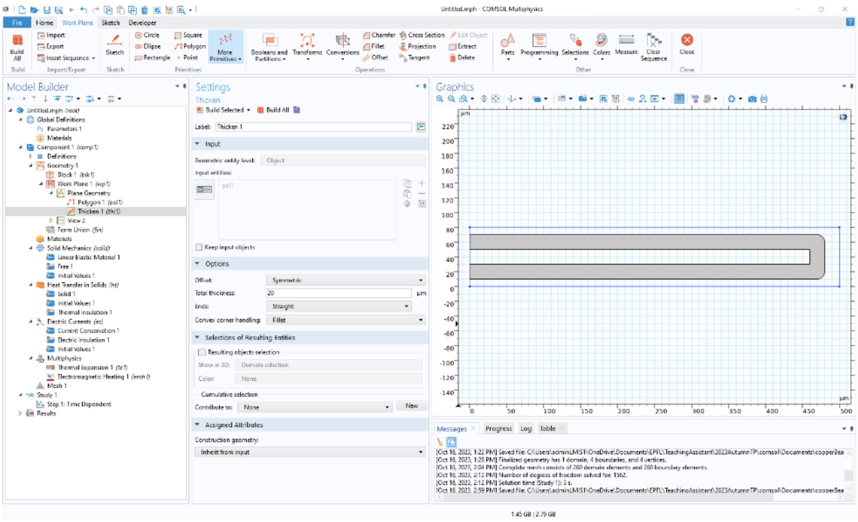
- In the **Geometry** toolbar, click **Work Plane > Work Plane**

*Polygon 1 (pol1)*

- Click **Work Plane 1 > Plane Geometry**
- In the **Sketch** toolbar, click **Polygon**, **Circular Arc**, **Interpolation Curve**, or any other tools, to draw the trace of the metal line on the beam, finishing by right-click and choose **Finish xxx**

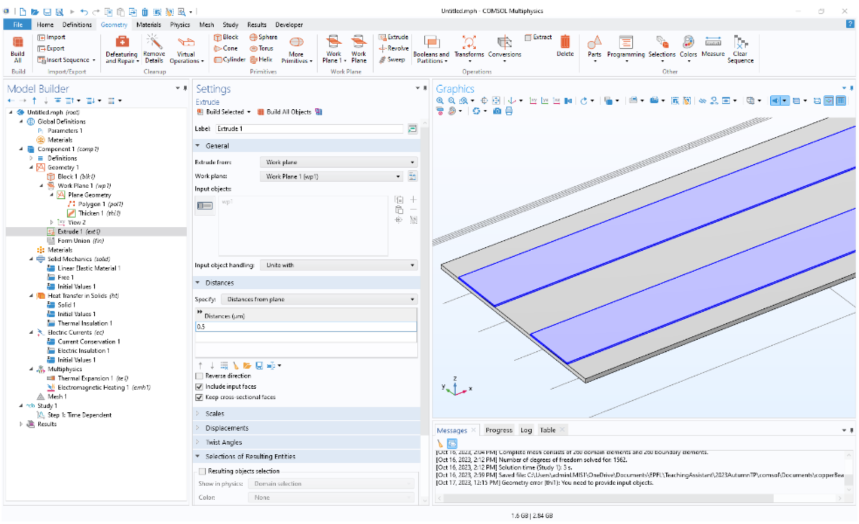
*Thicken 1 (thi1)*

- In the **Work Plane** toolbar, choose **Operations > Conversions > Thicken**
- In the **Settings** window of the **Thicken 1**, locate the **Input > Input entities**
- In the **Graphics** window, choose the metal trace you drew by left-click
- Locate **Options**, enter the width of the metal trace in the **Total thickness** text field
- Click **Build Selected**



*Extrude 1 (ext1)*

- Left-click **Work Plane 1 (wp1)**
- In the **Geometry** toolbar, click **Operations > Extrude**
- In the **Settings** window of **Extrude 1**, Locate **General**
- In the **Work plane** choosing list, choose **Work Plane 1 (wp1)**
- Locate **Distances**, enter the thickness of metal trace (**0.5µm**) in the **Distances** text field
- Click **Build All**



## ❖ MATERIALS

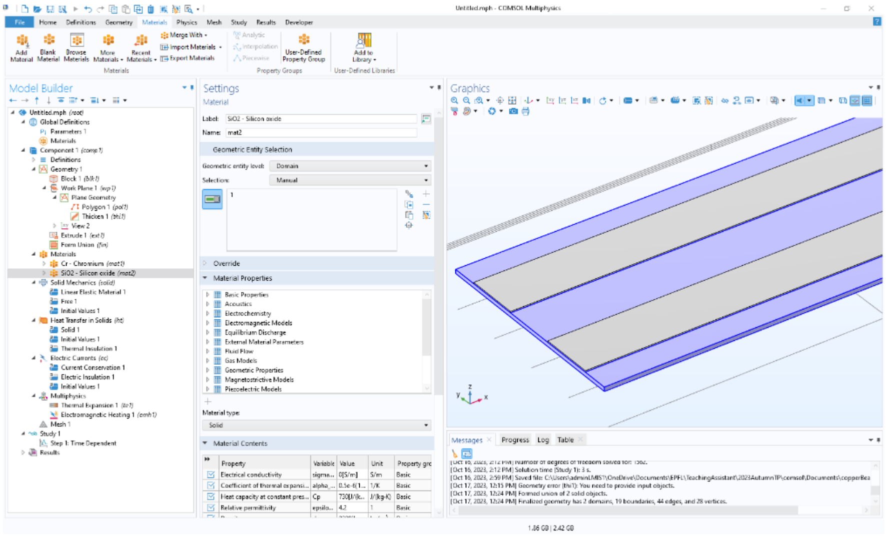
- In the **Materials** toolbar, click **Add Material**

*Cr – Chromium (mat1)*

- In the **Add Material** window, choose **MEMS > Metals > Cr – Chromium**, and click **Add to Component**
- In the **Settings** window of **Cr – Chromium (mat1)**, locate **Material Contents**
- Enter **1** in the text field of **Value** of **Relative permittivity**
- Enter **6.6e5[S/m]** in the text field of **Value** of **Electrical conductivity** (thin film conductivity rather than bulk conductivity, measured my CMi staff after deposition by evaporation)
- Locate **Geometric Entity Selection > Selection**
- In the **Graphics** window, choose only the metal trace by left-click

*SiO2 – Silicon oxide (mat2)*

- In the **Add Material** window, choose **MEMS > Insulators > SiO2 – Silicon oxide**, and click **Add to Component**
- In the **Settings** window of **SiO2 – Silicon oxide (mat2)**, locate **Geometric Entity Selection > Selection**
- In the **Graphics** window, choose the SiO2 cantilever by left-click
- Close **Add Material** window




## ❖ GLOBAL DEFINITIONS

- Click **Model Builder > Global Definitions > Parameters 1**
- Fill in the table with the following parameters (the **Value** column will be filled automatically)

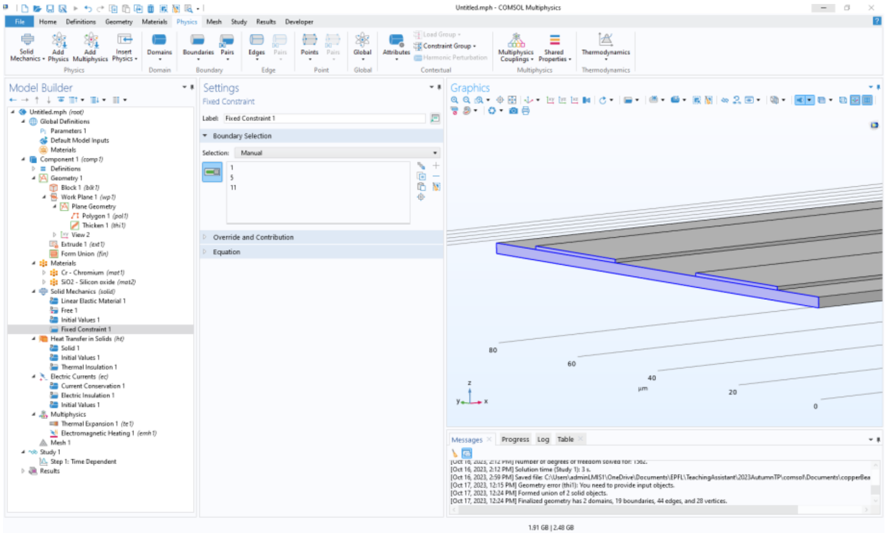
| Name | Expression | Description |
|---|---|---|
| f_exc | 1[Hz] | Excitation frequency |
| V_exc | 0.5[V] | Max excitation voltage |
| h_air | 10[W/(m^2*K)] | Air convection constant |
| T_init | 300[K] | Initial temperature |

## ❖ SOLID MECHANICS (SOLID)

- Left-click **Model Builder > Solid Mechanics (solid)**

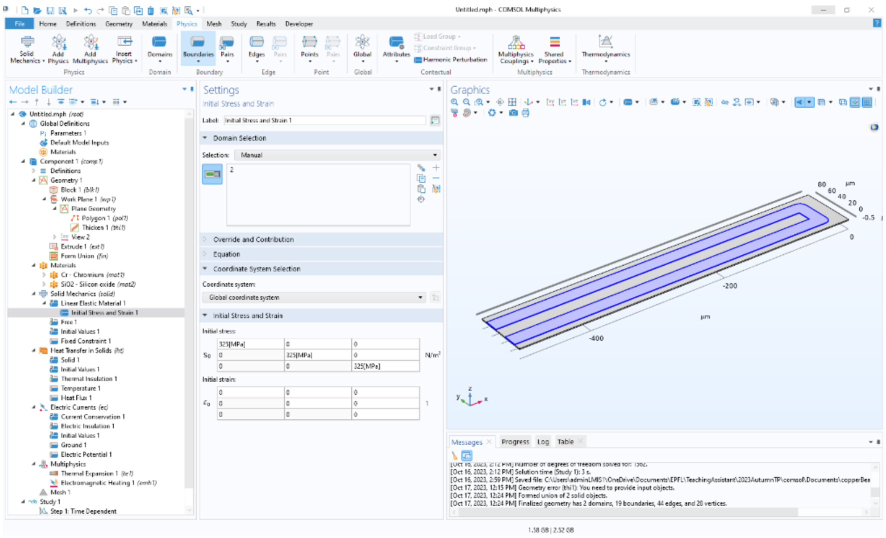
*Fixed Constraint 1*

- In the **Physics** toolbar, click **Boundary > Boundaries > Fixed Constraint**
- In the **Settings** window of **Fixed Constraint 1**, locate **Boundary Selection**
- In the **Graphics** window, choose all the surfaces that are fixed (including $SiO_2$ cantilever and Cr trace)



*Initial Stress and Strain 1*

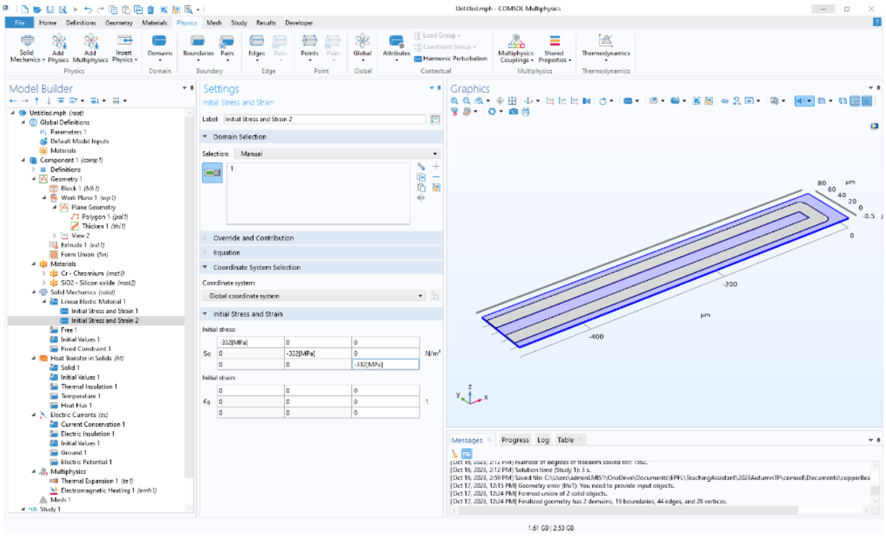
- Click **Model Builder > Solid Mechanics (solid) > Linear Elastic Material 1**
- In the **Physics** tool bar, choose **Contextual > Attributes > Initial Stress and Strain**
- In the **Settings** window of **Initial Stress and Strain 1**, locate Domain Selection
- In the **Graphics** window, choose the domain of Cr trace
- In the **Settings** window of **Initial Stress and Strain 1**, locate **Initial Stress and Strain**
- Enter **325[MPa]** along the diagonal of the text fields of **Initial stress**



*Initial Stress and Strain 2*

- Click **Model Builder > Solid Mechanics (solid) > Linear Elastic Material 1**
- In the **Physics** tool bar, choose **Contextual > Attributes > Initial Stress and Strain**
- In the **Settings** window of **Initial Stress and Strain 2**, locate Domain Selection

- In the **Graphics** window, choose the domain of $SiO_2$ cantilever
- In the **Settings** window of **Initial Stress and Strain 2**, locate **Initial Stress and Strain**
- Enter -**332[MPa]** along the diagonal of the text fields of **Initial stress**

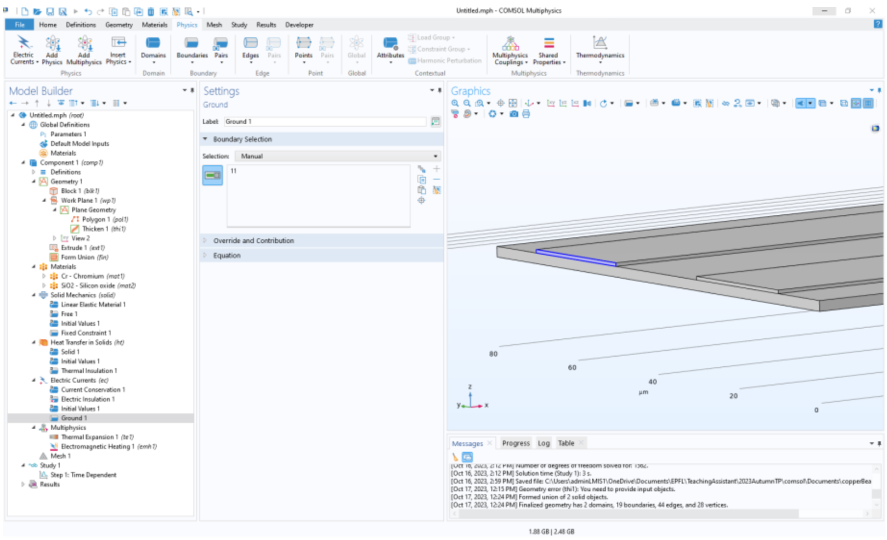


❖ **ELECTRIC CURRENTS (EC)**

- Left-click **Model Builder > Electric Currents (ec)**

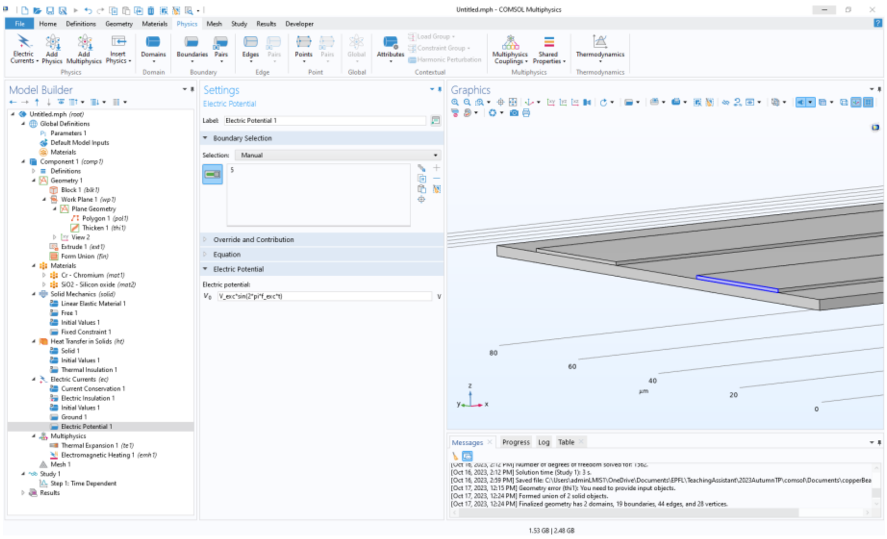
*Ground 1*

- In the **Physics** toolbar, choose **Boundary > Boundaries > Ground**
- In the **Settings** window of **Ground 1**, locate **Boundary Selection**
- In the **Graphics** window, choose one surface at the end of the Cr trace by left-click



*Electric Potential 1*

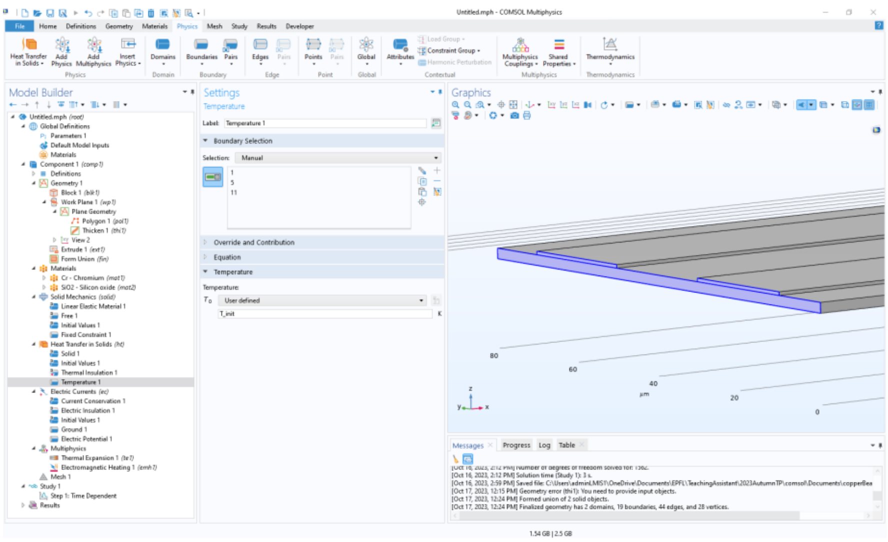
- In the **Physics** toolbar, choose **Boundary > Boundaries > Electric Potential**
- In the **Settings** window of **Electric Potential 1**, locate **Boundary Selection**
- In the **Graphics** window, choose another surface at the end of the Cr trace by left-click
- In the **Settings** window of **Electric Potential** 1, locate **Electric Potential**
- In the **Electric potential** text field, enter **V_exc*sin(2*pi*f_exc*t)**



- **HEAT TRANSFER IN SOLIDS (HT)**
  - Left-click **Model Builder > Heat Transfer in Solids (ht)**

*Temperature 1*

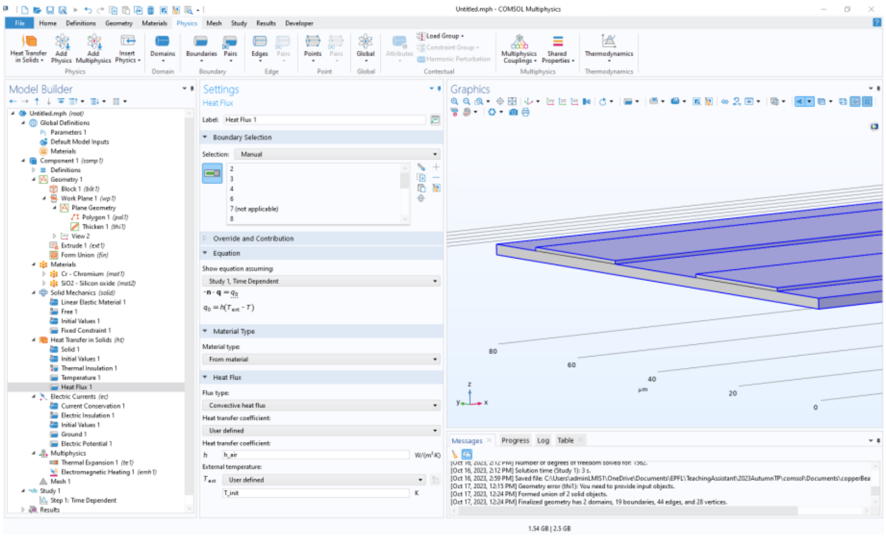
- In the **Physics** toolbar, choose **Boundary > Boundaries > Temperature**
- In the **Settings** window of **Temperature 1**, locate **Boundary Selection**
- In the **Graphics** window, choose all the surfaces that are fixed by left-click
- In the **Settings** window, locate **Temperature**
- In the **Temperature** text field, enter **T_init**



*Heat Flux 1*

- In the **Physics** toolbar, choose **Boundary > Boundaries > Heat Flux**
- In the **Settings** window of **Heat Flux 1**, locate **Boundary Selection**
- In the **Selection** list, choose **All boundaries**
- In the **Graphics** window, deselect the 3 surfaces that are fixed by left-click
- In the **Settings** window of **Heat Flux 1**, locate **Material Type**
- Choose **From material** in the **Material type** selection list
- In the **Settings** window of **Heat Flux 1**, locate **Heat Flux**
- Choose **Convective heat flux** in the **Flux type** selection list

- In the **Heat transfer coefficient** text field, enter **h_air**
- In the **External temperature** text field, enter **T_init**



- **MESH 1**
  - Left-click **Model Builder > Mesh 1**

*Free Triangular 1*

- In the **Mesh** toolbar, choose **Generators > Boundary > Free Triangular**
- In the **Settings** window of **Free Triangular 1**, locate **Geometric Entity Selection**
- In the **Graphics** window, choose the bottom surface of the Cr trace
- In the **Mesh** toolbar, choose **Attributes > Normal > Extremely Fine**

*Swept 1*

- In the **Mesh** toolbar, choose **Generators > Swept**
- In the **Settings** window of **Swept 1**, locate **Domain Selection**
- In the **Geometric entity level** choosing list, choose **Domain**
- In the **Graphics** window, choose the Cr trace domain
- In the **Mesh** toolbar, choose **Attributes > Distribution**
- In the **Settings** window of **Swept 1 > Distribution 1**, locate **Distribution**
- Choose **Fixed number of elements** in the **Distribution type** choosing list
- Enter **2** in the **Number of elements** text field

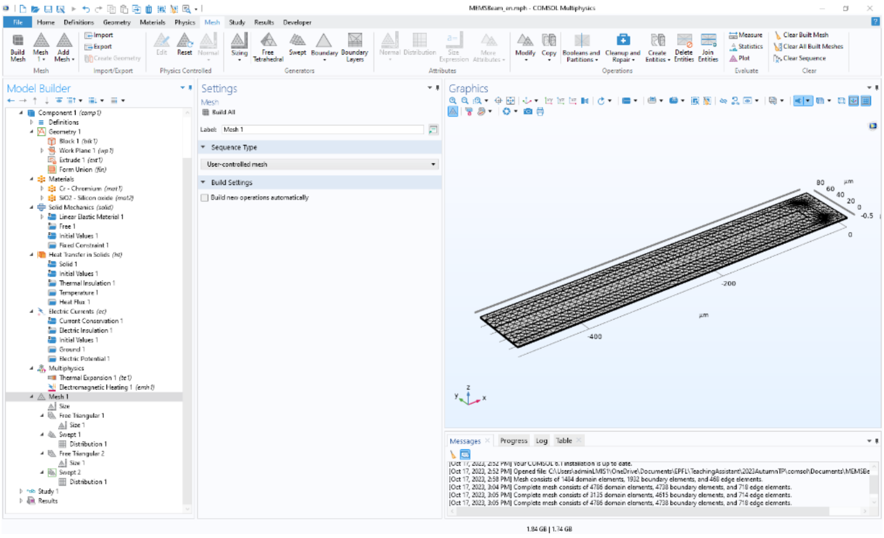
*Free Triangular 2*

- In the **Mesh** toolbar, choose **Generators > Boundary > Free Triangular**
- In the **Settings** window of **Free Triangular 2**, locate **Geometric Entity Selection**
- In the **Graphics** window, choose all the top surfaces of the $SiO_2$ cantilever (multiple surfaces)
- In the **Mesh** toolbar, choose **Attributes > Normal > Extremely Fine**

*Swept 1*

- In the **Mesh** toolbar, choose **Generators > Swept**
- In the **Settings** window of **Swept 2**, locate **Domain Selection**
- In the **Geometric entity level** choosing list, choose **Domain**
- In the **Graphics** window, choose the $SiO_2$ cantilever domain
- In the **Mesh** toolbar, choose **Attributes > Distribution**

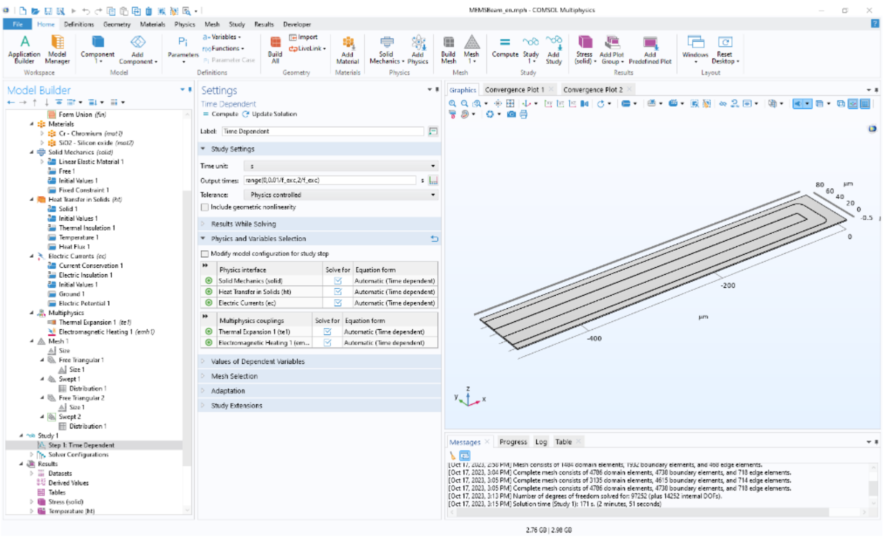
- In the **Settings** window of **Swept 2 > Distribution 1**, locate **Distribution**
- Choose **Fixed number of elements** in the **Distribution type** choosing list
- Enter **2** in the **Number of elements** text field
- Click **Build All**



- **STUDY 1 (TIME DEPENT)**
  - Left-click **Model Builder > Study 1 > Step 1: Time Dependent**
  - In the **Settings** window of **Step 1: Time Dependent**, locate **Study Settings**
  - In the **Output times** text field, enter text according to the chart below

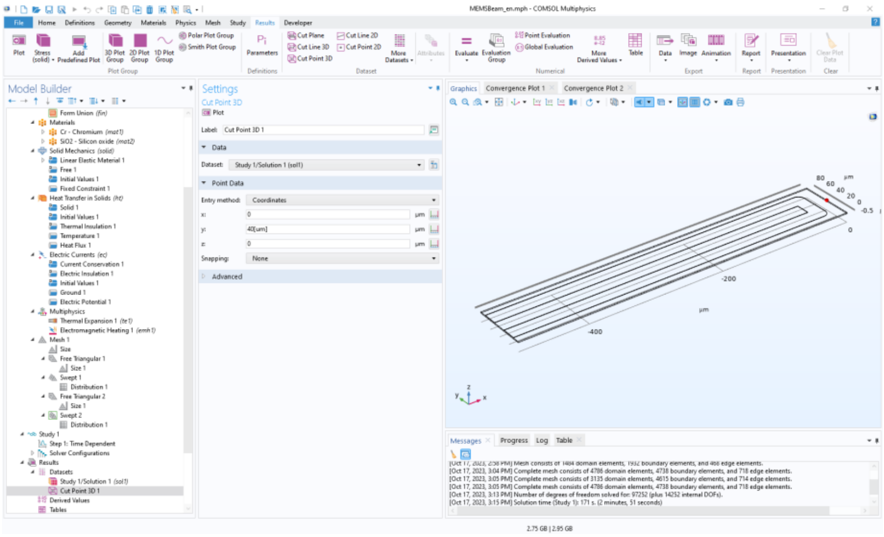
| Frequency f_exc | Output times |
|---|---|
| 1Hz | range(0,0.01/f_exc,2/f_exc) |
| 100Hz | range(0,0.05/f_exc,3/f_exc) |

  - Click **Compute**
  - To change the simulation frequency, remember to change the **f_exc** in **Global definitions > parameters** and **Step 1: Time Dependent > Output times**



- **RESULTS (TIME DEPENT)**
  - In the **Results** toolbar, choose **Dataset > Cut Point 3D**
  - In the **Settings** window of **Cut Point 3D 1**, locate **Point Data**

- In the **x**, **y**, and **z** text fields, enter the coordinates of the point in the middle of the free end part, for example, **(0, 40[um], 0)**
- Click **Plot** to confirm the position of this point



*1D Plot Group 6*

- In the **Results** toolbar, choose **Plot Group > 1D Plot Group**
- In the **1D Plot Group 6** toolbar, choose **Add Plot > Point Graph**
- In the **Settings** window of **Point Graph 1**, locate **Data**
- Choose **Cut Point 3D 1** in the **Dataset** choosing list
- In the **Settings** window of **Point Graph 1**, locate **y-Axis Data**
- Enter **solid.disp** in the **Expression** text field, or choose from the **Replace Expression** on the top-right of this section
- Click **plot**
- Enter **T** in the **Expression** text field, and click **plot** again for the temperature change

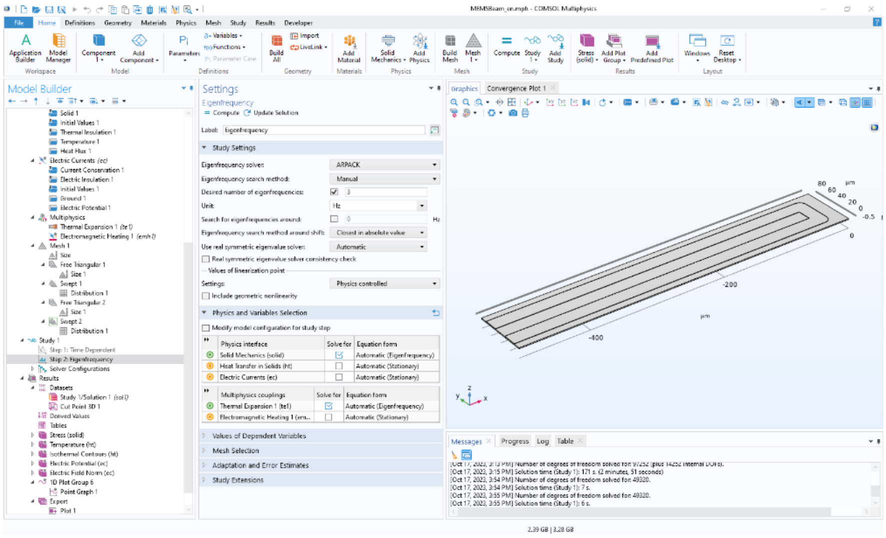
*Export*

- In the **Results** toolbar, click **Export > Data > Plot**
- In the **Settings** window of **Plot 1**, locate **Plot > Plot group** choosing list, and choose **1D Plot Group 6**
- Locate **Output > Filename**, select the place and filename with **Browse** for your exported file
- Click **Export**
- After change the **1D Plot Group 6 > Expression** to **T**, or change the frequency, repeat the upper steps again to export another set of data
- Use the exported data (2 frequencies, for both temperature and mechanical displacement) to answer the question in section 3.2

❖ **STUDY 1 (EIGENFREQUENCY)**

- Right-click on the **Step 1: Time Dependent**, choose **Disable**
- Right-click on the **Study 1**, choose **Study Steps > Eigenfrequency > Eigenfrequency**
- In the **Settings** window of the **Step 2: Eigenfrequency**, locate **Study Settings**
- Click to activate the **Desired number of eigenfrequencies**, and enter **3** in the following text field
- In the **Settings** window of the **Step 2: Eigenfrequency**, locate **Physics and Variables Selection**

- Deselect **Heat Transfer in Solids (ht), Electric Currents (ec),** and **Electromagnetic Heating 1 (emh1)**. Leave only **Solid Mechanics (solid)** and **Thermal Expansion 1 (te1)**
- Click **Compute**



## ❖ RESULTS (EIGENFREQUENCY)

- Left-click **Model Builder > Results > Stress (solid)**
- In the **Settings** window of the **Stress (solid)**, locate **Data**
- Choose different frequency in the **Eigenfrequency** choosing list
- Click **Plot**
- In the **Graphics** window, find the camera icon **Image Snapshot** to take a snapshot
- Record both the eigenfrequency and the shape of cantilever with snapshot, and use them to answer the question in the section 3.2

# 6 Report guidelines

After the COMSOL Simulation 4-hour session, you need to submit a report in the required time. Here are the instructions when you are writing your report.

## 6.1 Timeline

Your report must be submitted on Moodle latest one week after the practical session. You can make multiple submission, only the latest submitted file will be considered for grading. Please submit only one report per group entitled: "GRx_report_LastName1_LastName2_LastName3.pdf". This report is independent of the other sessions' report.

If you feel your answers on some questions are wrong, you are encouraged to seek advice from the TA. If this is the case, submit a complete report on Moodle and contact your TA with defined questions; still within one week after the practical session. Given you did all this, your TA will give you some additional time to improve your initial report.

## 6.2 Contents

### 6.2.1 Introduction

Explain how this practical session fits in the overall course (theory, simulation, fabrication, characterization). What is the role of simulation in a scientific framework? When do you normally do it (before / after fabrication / experimentation)?

Give a quick introduction to finite element analysis. List the various part of this practical session. State what kind of simulation we perform for the copper beam and MEMS actuator respectively.

### 6.2.2 Theoretical estimate

Add your answers to the preparation questions in section 2.2 here.

### 6.2.3 Results

For each section, describe the modelling / experimental work you performed with the help of the answers to the questions in the section 3. Your discussion should at least include answer to the questions below.

- Copper beam
  - Q1. Briefly describe the stationary study conducted. Plot the simulated deflection of the copper beam. Compare this result to the theoretical result from the preparation questions. How much does it deviate from theory?
  - Q2. Introduce the experiment you conducted with the copper beam. Describe your set up: how the beam was clamped, what electronics, light source and mechanical clamp you used.
  - Q3. Explain how the beam was actuated. Discuss whether copper on its own should experience a force in a magnetic field.
  - Q4. What way of measuring the beam deflection did you use? Is it a direct measurement? What oscillation frequency do you expect with respect to the stimulation applied?
  - Q5. Briefly describe the frequency response study. Plot the experimental and simulated response in the amplitude of the copper beam on the same graph. Comment on the two units of the data plotted. Are they comparable (in amplitude) and why? Discuss about the agreement between the theory, experiment and simulation. What is your opinion the main cause of discrepancy?
- MEMS actuators
  - Q6. Briefly describe the microfabrication process of the actuators. Why are they bent at rest? How do we account for this bending in the simulation?

- Q7. Explain the actuators functioning principle: why do they oscillate under an AC current and how is this originating from difference of thermal expansion coefficient? What equation rules the heating of the actuators? What frequency of oscillation do you expect with respect to the stimulation frequency?
- Q8. Briefly describe the time dependent study you performed. Plot the simulated displacement and temperature of the actuator on the same graph (use two y-axis). Make a separate graph for the stimulation at 1Hz and 100Hz.
- Q9. Explain how and why the result of the 100Hz stimulation are different. What behaviour would you expect at 10kHz (average displacement / temperature, oscillations amplitude)? Remember the equation describing the heating of the actuator from Q7. Is it dependent on the stimulation frequency?
- Q10. In order to complete a finite element model simulation faster, what parameter can we change and what is the trade-off? Which one did you influence in the MEMS actuator simulation? Is convergence of the solver always guaranteed?
- Q11. Report the eigenfrequencies of the actuator. Given the trend of the oscillations amplitude for increasing frequencies, do you need to prepare a microscope to observe the actuators during the future characterization of the actuator session?

### 6.2.4 Conclusion

Summarize your work in this session. What result is the most important for the continuation of the course?