

Assignment 3: Mechanical Field – Cantilever Beam

In Fig. 1, the setup of a cantilever beam made of steel is shown. The cantilever is fixed at its left hand side to a solid wall and is allowed to bend freely at the right hand side. The aim of this assignment is to characterize the mechanical behavior of this beam by means of numerical simulations.

In the following, we can assume that the length l is much greater than both the depth d and the height h of the beam. The depth d lies roughly in the order of the height h , i.e. the cantilever has roughly a square y - z -cross-section. Furthermore, the beam shall only be loaded with forces acting in x - or y -direction, i.e. no force acting into z -direction is to be expected. Under these circumstances, we can reduce the 3D model of the beam to a representative 2D model given by the x - y -cross-section.

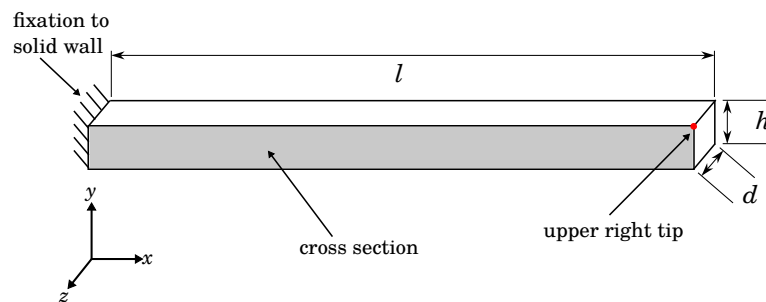


Figure 1: Cantilever beam fixed at its left hand side. The x - y -cross-section is colored in gray.

Consider the following dimensions for the geometry of Fig. 1:

- length $l = 0.8$ m
- height $h = 0.012$ m
- depth $d = 0.02$ m

General hints and remarks

Please make sure to:

- Save all relevant plots you generate (e.g., as **png**-file) while working on the assignment.
- Always label the axes of the plots and add a legend if necessary.
- Format your plots so that they can be read easily (e.g., appropriate font size).
- Prepare a **Results.pdf** file with the plots and short answers to all questions (label each answer with its task number) for submission.
- If you are unsure on how to perform the tasks, refer to
 - [Comsol Tutorial: Electrostatic Field](#)
 - [COMSOL Multiphysics Reference Manual](#)

1 Preparation

- 1.1. The geometry can be simplified to 2D, yielding significantly faster computations. For the given values of h , d and l , what mechanical 2D model can be used (plane strain, plane stress or axi-symmetric)?
- 1.2. Consider the analytic formula for the estimation of the first three bending modes (bending eigenfrequencies) for bending around the z -axis of the cantilever shown in Fig. 1 (see Parkus: "Mechanik der festen Körper", Springer, 1995):

$$\omega_i = \gamma_i^2 \sqrt{\frac{E J_z}{\rho A}} . \quad (1)$$

Thereby, ω_i stands for the **angular** eigenfrequency, E for the elasticity modulus (Young's modulus), J_z for the second area moment belonging to the z -axis ($J_z = dh^3/12$), ρ for the density and A for the $y - z$ -cross-section. Additionally, γ_i is a correction-constant given as

$$\begin{aligned} \gamma_1 l &= 1.875, \\ \gamma_2 l &= 4.694, \\ \gamma_3 l &= 7.855 \end{aligned} \quad (2)$$

for the first three eigenfrequencies.

Combine (1) with (2) and the given information to derive a formula for the actual eigenfrequencies $f_1 = \omega_1/(2\pi)$, f_2 and f_3 . These formulas shall only depend on the product $\gamma_i l$ as well as l, h, d, E and ρ (one of these parameters will cancel out). Write the derived formula to `Results.pdf`.

2 Modeling

- 2.1. Use the **Model Wizard** to create a new 2D model. Add a **Structural Mechanics** → **Solid Mechanics** physics. This physics solves *Navier's equations* in 3D or 2D. Pick the **Eigenfrequency** study before concluding with the wizard. Note: you could instead add all of those nodes manually, if you prefer.
- 2.2. Make sure that the geometric parameters l , h and b , as well as the mesh's element size in x - and y -direction (h_x , h_y) can be adapted easily by defining them as parameters. Note: **h** is a reserved name in COMSOL and you can't use it as a parameter name. It is a good practice to use longer parameter names (e.g., **h_cant**) in order to avoid conflicts with reserved variable names.
- 2.3. Model the relevant cross-sectional area as shown in Fig. 1.
- 2.4. Go to your physics node (**Solid Mechanics (solid)**) and make sure that you are using the correct mechanical 2D approximation.
- 2.5. The depth d of the cantilever is not modeled in the 2D geometry. Some quantities, however, scale with d , e.g., the total stored elastic energy. In order to consider this scaling, set the value for d in the **Solid Mechanics (solid)** node under the *Thickness* section (**d**).
- 2.6. What type of shape functions are set by default? → physics node, *Discretization* section.
- 2.7. How many dependent variables are we considering for the mechanical simulation and how are these variables denoted in COMSOL? → physics node, *Dependent Variables* section.
- 2.8. Add the material **Built In** → **Structural steel** to your component. Assign it to all domains.
 - 2.8.1. What is the value of the Young's modulus (E) of this material?
 - 2.8.2. What is the density ρ of the material?

- 2.9. The **Solid Mechanics (solid)** → **Linear Elastic Material 1** node adds the PDEs that are to be solved.
- 2.9.1. What variable name is being used for the mechanical strain?
 - 2.9.2. What variable name is being used for the mechanical stress?
 - 2.9.3. What variable name is being used for the stiffness tensor?
 - 2.9.4. What equation is being solved for?
 - 2.9.5. In COMSOL, the eigenvalue is denoted with λ . How does it relate to the angular eigenfrequency ω ?
- 2.10. The default boundary conditions (BCs) are set by the **Free 1** node. What kind of BC is this? Why is the node called *Free* in COMSOL?
- 2.11. Add any further boundary conditions that are necessary to model the cantilever beam.
- 2.12. Use a **Mapped Mesh** with quadrilateral elements of size $h_x \times h_y$.
- 2.12.1. Add a **Mapped** node to your **Mesh 1** node. Apply it to the *Entire geometry*.
 - 2.12.2. Add a **Distribution** node to the **Mapped 1** node and apply it to one of the horizontal lines of your geometry. Select *Fixed number of elements* as *Distribution type* and set the *Number of elements* to l/h_x .
 - 2.12.3. Add a second **Distribution** node to the **Mapped 1** node and apply it to one of the vertical lines of your geometry. Select *Fixed number of elements* as *Distribution type* and set the *Number of elements* to h/h_y .
 - 2.12.4. Build the mesh and make sure that you obtained a nicely structured mesh.

3 Eigenfrequency/Modal analysis

- 3.1. Apply the analytic formula, which you derived, to estimate the first three eigenfrequencies of the assembly from Fig. 1 using the given set of parameters. The required material data has been extracted in task 2.8. You can do the computations in Matlab, as you will be using the results in task 3.8. again. Please write the three calculated eigenfrequencies into **Results.pdf**.
- 3.2. Do you need an excitation to perform an eigenfrequency analysis? Why?
- 3.3. Perform a mechanical eigenfrequency analysis for the values $h_x = 8$ mm and $h_y = 2$ mm. The simulation should only take a few seconds. COMSOL will automatically plot the **Mode Shape** for the first eigenfrequency f_1 . The value of f_1 can be read off in the *Data* section of the plot. Inspect the first three mode shapes of the cantilever beam.
- 3.4. How many eigenfrequencies does the cantilever have in theory? Why? How many have been calculated by COMSOL? Hint: the *Data* section in the *Mode Shape* plot shows all computed eigenfrequencies.
- 3.5. In our analysis we are only considering the first three modes. Go to the **Study 1** → **Step 1: Eigenfrequency** node and set the *Desired number of eigenfrequencies* to 3. This setting will speed up your simulation.
- 3.6. Perform a parameter study for:
 - 3.6.1. linear elements with $h_x = 2$ mm, 4 mm, 8 mm, 16 mm, 32 mm.
 - 3.6.2. quadratic elements (Serendipity) of the same size as above (task 3.6.1.).
 Note: keep $h_y = 2$ mm constant for the above simulations.
- 3.7. Export the resulting first eigenfrequency from 3.6.1. and 3.6.2. to MATLAB:

- 3.7.1. Right click on **Derived Values** and choose *Global Evaluation*. In the settings windows select the *Study 1/Parametric Solutions 1* data set and as *Eigenfrequency selection* choose *First*. In the *Table columns* drop-down menu select *Inner solutions*. Under the *Expressions* section add *Solver → freq - Frequency*. Finally, click on *Evaluate* at the very top. This will create a *Table 1* node as a child of the *Tables* node. Every time you evaluate, COMSOL will add a column to your table, so make sure to click only once. Otherwise, you will need to delete the *Table 1* first.
- 3.7.2. Right click on **Table 1** and select *Add Table to Export*. Make sure to *unselect* the *Include header* check-box, then click on *Export* at the very top.
- 3.7.3. Import the data to Matlab by clicking on *Import*. The data is delimited by space. Preferably, use *Numeric Matrix* as *Output Type*. Select only the cells which contain the frequency values.
- 3.8. In Matlab, calculate the relative deviation $e_{\text{rel},i}$ between your simulated eigenfrequencies (f_{sim}) and the analytic estimations (f_{analytic}) from task 3.1. Use the following formula to compute the relative deviation
- $$e_{\text{rel}} = \frac{f_{\text{sim}} - f_{\text{analytic}}}{f_{\text{analytic}}}, \quad (3)$$
- for the first eigenfrequency. Perform this calculation for all five simulations with linear shape functions as well as for all five simulations with quadratic shape functions. Gather the calculated values of e_{rel} for both cases (linear and quadratic shape functions) and create two **semi-logarithmic** (`semilogy` in MATLAB: y -axis logarithmic, x -axis linear) graphs showing e_{rel} versus h_x . Plot **both** graphs to the same figure and do not forget to label the figure correctly (including a legend to determine which graph belongs to linear and quadratic shape functions). For help plotting in Matlab, you can type `help semilogy` at the Matlab prompt, or refer to the Matlab tutorial linked on StudOn. Remember to save the plot to a `png`-file for submission.
- 3.9. What can you observe regarding the relative deviation between analytic and numerical solution? Is there a difference between linear and quadratic shape functions? If yes, what numerical effect is responsible for it?
- 3.10. What is the smallest relative deviation in percent that you were able to achieve? What are the reasons for this discrepancy?

4 Transient analysis - Time step study

In the following, use elements of size $h_y = 2$ mm, $h_x = 8$ mm with quadratic shape functions.

- 4.1. Excite the cantilever by applying a force in y -direction at the upper right tip marked in Fig. 1. The time-dependent force should consist of a sinusoidal burst with the following properties:
- amplitude `Aexc`: 1 N
 - frequency `fexc`: 125 Hz
 - number of periods `Nperiods`: 3.
- 4.1.1. Define `Aexc`, `fexc` and `Nperiods` as parameters.
- 4.1.2. The above dependence on time t can be modeled by windowing a sin-function. Add a `Rectangle`-function to your `Global` node and set the parameters correctly (do not smooth). This function will be used as window function.
- 4.1.3. Add an `Analytic`-function to your `Global` node and set its argument variable name to `t`. This shall be the exciting function. For *Expression* enter `Aexc*rect1(t/1[s])*sin(2*pi*fexc*t)`. Comsol knows the constant value of `pi` = π . Set the argument and the units appropriately. Plot this function from 0 s to 0.32 s and verify that you obtain a sin-burst with three periods.

- 4.1.4. Add a **Point Load** node to your physics and apply it to the upper right tip point marked in Fig. 1. Enter $\mathbf{an1}(\tau)$, which is your function defined in 4.1.3., as time-dependent value of the y -component of the force. The variable τ is a global variable in COMSOL which represents time.
- 4.2. Add a new **Study** to your COMSOL project: right click the root node \rightarrow *Add Study* \rightarrow double click *Time Dependent*. This study can be computed independently from previously configured studies.
- 4.3. Select **Study 2** \rightarrow **Step 1: Time Dependent**. Change the time unit to ms. In the *Times* field, enter the points at which to simulate. Simulate a total of $\mathbf{Tend} = 320$ ms with a total of $\mathbf{Nsteps} = 1600$ time steps.
- 4.4. The time points specified in task 4.3. are actually the time points at which COMSOL will **save** the results. This does not necessarily correspond to the points taken by the **time stepping algorithm** (the numerical solver). Change this behavior by:
 - 4.4.1. Right clicking on **Study 2** and selecting *Show Default Solver*.
 - 4.4.2. Disclosing the *Solver Configurations* \rightarrow *Solution x* and selecting *Time-Dependent Solver 1*. Under *General* \rightarrow *Times* you could explicitly specify the time points that the solver should take. Instead, force the solver to take at least the same steps as specified in task 4.3. For this purpose, disclose *Time Stepping* and set *Steps taken by solver* to *Strict*.
- 4.5. Compute **Study 2**. This should take some seconds.
- 4.6. Animate the result: right click **Export** and select *Animation* \rightarrow *Player*. Choose appropriate settings and view your simulation result. Does the vibration stop after the excitation? Why? Hint: Your result is correct if the vibration consists in a superposition of the mode structures you inspected in task 3.3.
- 4.7. Perform a parametric sweep on \mathbf{Nsteps} with the values: 200, 400, 800, 1600, 3200.
- 4.8. Plot a series of curves showing the y -component of the displacement at the upper right tip over time t . Which is the smallest value of \mathbf{Nsteps} that yields converged results?

5 Fourier Transform of the Time Step Study

- 5.1. The time domain signal is difficult to interpret. Calculate the Fourier transform of the y -displacement at the upper right tip. You can do this in COMSOL with a new *Study* using the following procedure:
 - 5.1.1. Add a new *Time-Dependent Study* (each study can be computed separately and you can have as many as you like).
 - 5.1.2. Add to this **Study 3** node a *Time to Frequency FFT* study step by right clicking the node and selecting \rightarrow *Study Steps* \rightarrow *Frequency Domain* \rightarrow *Time to Frequency FFT*. The two study steps will be performed one after another.
 - 5.1.3. Adjust the settings of the *Time to Frequency FFT* study step:
 - Time unit \rightarrow ms,
 - End time \rightarrow \mathbf{Tend} ,
 - Maximum output frequency \rightarrow 300 Hz.
 - 5.1.4. Set the steps to be taken by the solver as in 4.4.
- 5.2. Compute the **Study 3** which you just set up with $\mathbf{Nsteps} = 1600$.
- 5.3. Plot the magnitude spectrum: Add a *1D Plot Group*, then add a *Point Graph* to it and choose the upper right tip. Choose the Data set that corresponds to the frequency domain data. Plot $\mathbf{abs(v)}$ over the frequency. You should see some peaks in the plotted spectrum.

- 5.4. The resolution of the Fourier transform is not good. You can increase the resolution by increasing the length of the time domain signal using zero-padding. Go back to the *Step 2: Time to Frequency FFT* node settings. Increase the *End time* to $5 \cdot T_{\text{end}}$. Additionally, check the *Use window function* checkbox, choose a *Hanning* window and let the window end at T_{end} .
- 5.5. Right click the *Step 2* node and select *Compute Selected Step* to re-compute just the Fourier transform.
- 5.6. Go back to your *Point Graph* plot with the magnitude spectrum. The graph should be much smoother now. Where are the maxima located? Compare this to the eigenfrequencies computed previously.

Note: The Fourier transform shows that the vibration of the cantilever excited with the sin-burst can be interpreted as a **superposition of its modes**.

Submitting your assignment

Hand in your group's

- COMSOL *mph*-file (clear all meshes and results beforehand, see the COMSOL Tutorial), and
- *Results.pdf* file with the plots and short answers to all questions (label each answer with its task number),

before the due date by uploading to the corresponding StudOn section.

When handing in for the first time you need to create a group. Please add all members when doing so! Once the group has been create, you can and should re-use the group for the subsequent submissions.