CAE of Sensors and Actuators
Assignment 2

Tutorial: November 17th and November 18th, 2014
Assignment due: November 24th, 2014, 6 pm

Mechanical Field Problem: Bending plate / Bending cantilever

Consider the setup of a **steel**-plate as shown in Figure 1. The plate is fixed along the left side at a solid wall. At the bottom right edge a series of hooks is attached, which can be used to apply various types of loads. The aim of this assignment is to characterize the mechanical behavior of this plate.
In the following we assume that the depth d of the plate is much larger than its length and height. Furthermore all forces, boundary conditions, etc. which are applied to the plate can be considered uniform along the whole depth. In that particular case we can reduce the 3D plate-model to a **2D cantilever**-model featuring **only the x-y-crossection** of the plate. The length and height of the crossection shall be:

length l	=	50 cm
height h	=	1 cm

For all simulations we assume the cantilever to be **undamped** (i.e. no damping model needs to be applied).

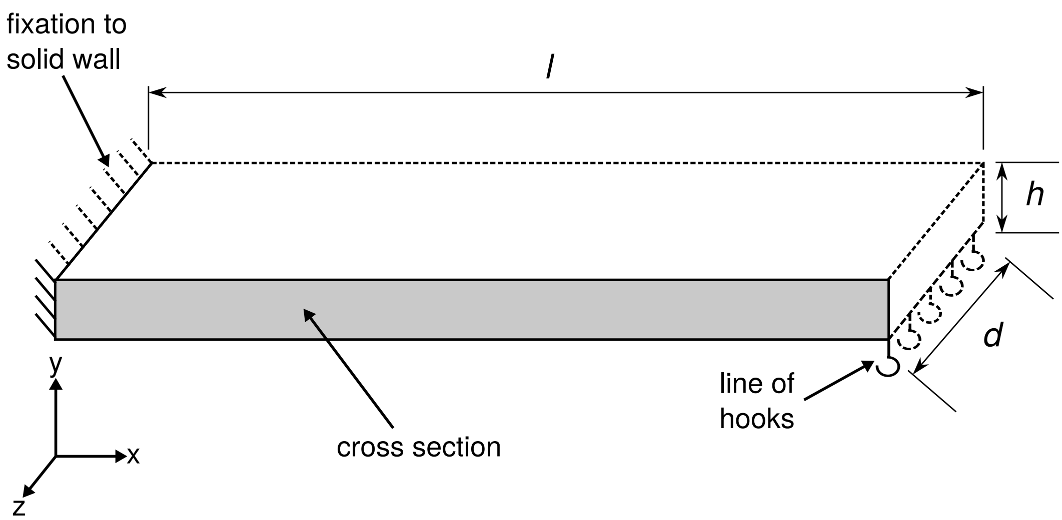

Figure 1: Steel plate with length l, height h and depth d. For the case $d \gg l$ and $d \gg h$ the x-y-crossection can be considered as a simple cantilever.

Task 1: Eigenfrequencies + Mesh study (3 points)

Simulation Tasks

1. Create at least **5** `.nmf`-mesh files representing the x-y-crossection of the plate. In y-direction use a **constant** number of 4 elements, in x-direction use 5 different numbers of elements between 10 and 1000. As element type choose standard **linear** elements (`SETELEMS,'quadr',''`).
   Hint 1: If your mesh gets distorted (i.e. fine mesh only at the boundary but very coarse inside the cantilever) add `mshkey,1` to your mesh-script before the meshing command.
   Hint 2: You do **not** have to model the hook!
2. Repeat the above meshing process for quadratic elements (i.e. elements with quadratic ansatz-functions). Quadratic elements can be created by Ansys using `SETELEMS,'quadr','quad'`.
3. Perform on all meshes created in 1. and 2. an **eigenfrequency** simulation and for each simulation note the first 3 simulated eigenfrequencies f_1, f_2 and f_3.
   Hint 1: For eigenfrequency simulations the system may not be affected by any loads or forces, however any fixations have to be considered!
   Hint 2: Set `Frequency Shift` under NACS eigenfrequency analysis to 1 Hz.

Calculations

1. Estimate the first three eigenfrequency f_1, f_2 and f_3 with the analytic formula (see Parkus: "Mechanik der festen Körper", Springer, 1995)

$$\omega_i = \gamma_i^2 \sqrt{\frac{EJ_x}{\rho A}} \tag{1}$$

   with ω the angular frequency, E the elasticity modulus (steel: $E \approx 195 \cdot 10^9$ N/m^2), J_x the axial angular impulse belonging to the x-axis (with rectangular cross section $A = lh$), $J_x = lh^3/12$ and l denoting the length and h the height of the cross section), ρ the material density (steel: $\rho \approx 7850$ kg/m^3) and γ a value belonging to your eigenmode.

$$\begin{aligned} \gamma_1 l &= 1.875 \\ \gamma_2 l &= 4.694 \\ \gamma_3 l &= 7.855 \end{aligned} \tag{2}$$

Questions

1. Which mechanical subtype do you need to model the given geometry (plane strain or plane stress)?
2. How do the simulated eigenfrequencies change if the number of elements in x-direction gets increased? Are there differences between linear and quadratic elements? Explain the observed behavior!
3. Compare the simulated eigenfrequencies with the analytically calculated ones.

Files to submit / Grading

- Ansys mesh script for linear and quadratic elements as well as all created meshfiles (0.5 points)
- Nacs simulation script(s) (`.py`) used for the eigenfrequency analysis (0.5 points)
- `Results.txt` file containing the analytic and simulated eigenfrequencies and the intermediate steps of the analytic computation (1 point)
- `Results.txt` file answering the questions (1 point)

Task 2: Harmonic behavior (2 Points)

Simulation Tasks

1. Take the finest and coarsest mesh with **linear** elements from task 1 and perfrom a **harmonic** analysis of the cantilever in a frequency range which captures the first 3 eigenfrequencies f_1, f_2 and f_3. For those simulations assume a weight of 0.5 kg ($\rightarrow$ a force of $\approx$ 5 N) which **hangs** on the hook.
   Hint 1: You do **not** have to model the hook! Simply apply the force to the point where the hook is attached to the cantilever!
   Hint 2: Use logarithmic sampling under NACS harmonic analysis and use at least 100 frequency steps.

2. Repeat the above simulations for the finest and coarsest mesh with **quadratic** elements.

Plots

1. Create a **double-logarithmic** (x-axis and y-axis logarithmic) plot showing the y-displacement of the top right corner over the frequency. The plot should contain a graph for each of the four cases:

   - linear elements, finest mesh
   - linear elements, coarsest mesh
   - quadratic elements, finest mesh
   - quadratic elements, coarsest mesh

   Hint: Do not forget to label the plots and add a legend!

Questions

1. How does the harmonic behavior change if the number of elements in x-direction gets increased?

2. How does the harmonic behavior change if you use quadratic instead of linear elements?

Files to submit / Grading

- Nacs simulation script(s) (`.py`) used for the harmonic analysis (0.5 points)
- Plot file showing the y-displacement of the top right tip for linear and quadratic elements for a fine and a coarse mesh respectively (1 point)
- `Results.txt` file answering the questions (0.5 point)

Task 3: Transient behavior + Time discretization study (2 Points)

Simulation Tasks

1. Take the finest mesh with **linear** elements from task 1 and perform a **transient** simulation of the cantilever. Set the timestep dt of the transient analysis to $1/(10 * f_2)$ at first, where f_2 is the second eigenfrequency calculated above. The number of timesteps N shall be large enough to simulate 10 full periods at a frequency of f_2. As excitation of the mechanical system apply a sinusoidal burst (`sinusBurst`) with an amplitude of 5 N, a frequency of f_2, 3 periods in total, 1 period fade-in and 1 period fade-out. Apply this excitation to the top right tip of the cantilever in negative y-direction.
   Hint: You can define the shape of excitation in NACS next to the field where you set the excitation value, i.e. under:
   `Boundary Conditions -> $$$ -> Components -> $$$-value -> ...`

2. Repeat the simulation above for $dt = 1/(15 * f_2)$, $dt = 1/(20 * f_2)$ and $dt = 1/(25 * f_2)$. Adapt the number of timesteps N in each case such that always 10 full periods at f_2 can be simulated.

Plots

1. Create a **linear** plot showing the y-displacement of the top right corner over time. The plot should contain a graph for each of the four used timesteps dt.
   Hint: Do not forget to label the plots and add a legend!

Questions

1. Which time discretization can be seen as fine enough, i.e. for which timestep dt are the results nearly the same as for the next finer timestep?
2. Look at the plot you created. Do you see a pure sinusoidal signal with a frequency of f_2? Explain your observation.

Files to submit / Grading

- Nacs simulation script(s) (`.py`) used for the transient analysis (0.5 points)
- Plot file showing the y-displacement of the top right tip for the four tested timesteps (1 point)
- `Results.txt` file answering the questions (0.5 point)

General hints and remarks

- As reference for the `Results.txt` file take a look at the sample file `Results_sample.txt`
- Submit all your files until the due date by copying your results (`Results.txt`, scripts, screenshots, ...) to
  `/home/userHome/stud/CAESAR/group<Group#>/assignment<Assign#>/`
- Points are only given if submitted data contain correct results, answers are comprehensible and plots are labeled correctly!