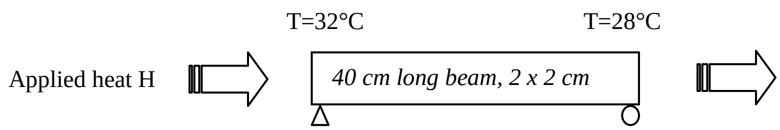
OPTI 421/521 – Introductory Opto-Mechanical Engineering

Homework 7 –Thermal stress, deformation, Finite element modeling

1.) Thermal expansion : Axial

Consider a beam made of 6061 aluminum, simply supported at the ends.
The beam is 40 cm long at 20°C, 2 cm x 2 cm uniform cross section.

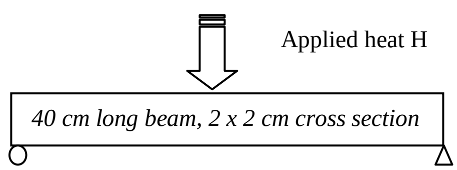
a) Find the coefficient of thermal expansion and the thermal conductivity for this material
b) Find the change in length if the beam is heated uniformly to 30° C (a change of 10° C).
c) Consider the case where one end is heated, creating a linear thermal gradient of 4°C from one end of the beam to the other. The temperature profile below is is created. Plot the temperature as a function of position on the bar.



d) Calculate the thermal power in Watts required to maintain this steady state gradient
e) Calculate the maximum stress due to the thermal gradient
f) Integrate the expansion along the bar to determine the total change in length due to this applied thermal load. Plot the shift due to thermal expansion as a function of position on the bar.
g) Consider an alternate boundary condition where both ends are fixed, overconstraining the beam so it cannot expand. When the beam is heated as above in c), determine the reaction forces and calculate the stress in the bar.

2) Thermal distortion : Lateral

Consider the same beam as above with the simple support, but look at an alternate case where the beam is heated from the top so that the top of the beam is 0.002° C warmer than the bottom. (The top is 20.001 C and the bottom is 19.999 C).



a) Calculate the thermal power in Watts required to maintain this steady state gradient
b) Calculate the maximum stress due to the thermal gradient

c), d) Sketch the shape of the beam deformation. Determine the slope at the ends of the beam and the deflection in the middle.

e) Consider a different set of boundary conditions that overconstrains the bar such that both ends are fixed and not allowed to rotate. Determine reactions from the boundary conditions that will keep the beam straight in the presence of the thermal gradient above.

f) Calculate the maximum stress in the beam in this condition.

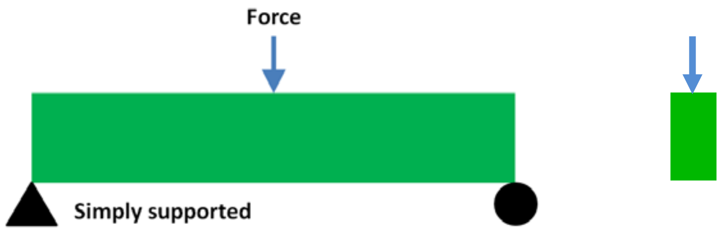
Introductory finite element modeling in Solid Works

3. Finite element modeling : deflections and stress from loading

Use Solidworks to model an aluminum beam, 15 cm long, 2 cm x 1 cm cross section with appropriate boundary conditions and loading to simulate the following cases with an applied force of 10 N.

- Cantilever bending, force applied at the end

- Simply supported beam bending, force applied in the middle



Create two different finite element models for each case:

- Model this using beam elements
- Use solid tetrahedral element

For each model, set the element size appropriately.

a) Show the deflection from each of the four Solidworks FE models.
b) Compare the maximum deflection with the hand calculations and comment on differences.
c) For the simulations with tetrahedral elements, show the stress distribution.
d) Compare these with the hand calculations and comment on the differences.

4. Vibration analysis

Use Solidworks finite element modeling to evaluate the vibrational modes for two models above, replacing the applied force with a 1 kg mass (which weighs 10 N) at the same location. Compare the results for the lowest mode to a hand calculation. Make a comment about the shapes of the higher order modes. Do they make sense?

5 Rules of Thumb

Provide three rules of thumb using the standard format.