Simulation Lab 5

Analysis of a Bimorph Actuator using Solid187 elements in Ansys

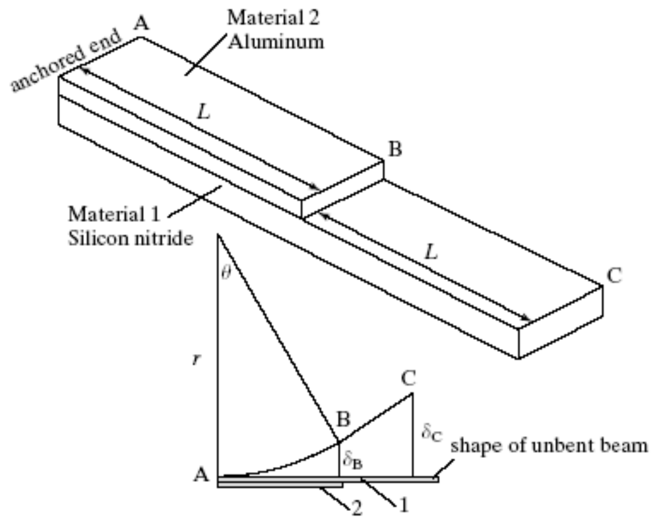
1 Introduction

In this lab, you will use ANSYS to analyze the deflection of the bimorph actuator shown and described below. In addition to specifying the deflections at point C and B, we are interested to know the stress conditions at the anchored wall. Finally, you will repeat the problem by constraining the tip face at location C.

Example 5.2. Displacement of a Bimorph Actuator

A bimorph cantilever beam is made of two layers of different lengths. The layer on top is made of aluminum (material 2), whereas the layer on the bottom is made of silicon nitride (material 1). The width of both layers is 20 μm. The length of the segment between points A and B is 100 μm, so is the length of the segment from point B to C. Young's modulus of aluminum and silicon nitride are $E_2 = 70$ GPa and $E_1 = 250$ GPa, respectively. The thickness of aluminum and silicon nitride sections is $t_2 = 0.5$ μm and $t_1 = 1$ μm, respectively. The thermal expansion coefficients of aluminum and silicon nitride are $\alpha_2 = 25$ ppm/°C and $\alpha_1 = 3$ ppm/°C, respectively. At room temperature, the cantilever is straight.

Find the radius of the curvature (r) of the cantilever beam when the beam is uniformly heated to 20°C above the room temperature. Determine the amount of vertical displacement at the free end of the beam under this condition.



2 Procedure

Read the problem statement described above (the TA will help describe the problem statement and the outcome of the analysis). You will model the beam using the SOLID187 element, which is a 10-node brick element in ANSYS. Using the GUI may be the simplest but you can write a batch file if you so desire. The steps for ANSYS are discretized below:

Specify the element you will use
Create 2 material properties
Specify Celsius as working T scale
Create 2 volumes

Specify which material goes to which volume
Glue the volumes
Mesh the volumes (if using the Meshtool, use a Smart Size of 6 – average mesh)
Specify your boundary conditions
Apply temperature load (uniform or to a volume)
Solve
Display Displacements at location B and C as well as stresses at location A

If you used the GUI, you should output the log file that was created. **Add a constraint to apply a fix boundary condition on the tip face at location C and resolve.**

3 Deliverables

Turn in a professional memo that describes what you have learned. Report the displacement at location B and C as well as the stresses at location A. Include a displacement plot. Compare to the solution provided in your textbook for Example 5.2. Are the results comparable? If not, provide an explanation for the differences in the results. What about the stresses at Location A. Share what you like and dislike about FEA models. Are they useful? Why?

Report results for fixed-fixed boundary conditions (at location A and C). Provide a displacement plot. Compare all results (displacement and stress) to the fixed-free. Describe possible applications for **both** boundary conditions.