Heat Transfer and Solid Mechanics with a Cantilever

In this set of tutorials, we study heat conduction and its coupling to solid mechanics via thermal expansion. Thus, this is a multiphysics problem coupling heat transfer to solid mechanics. The coupling in this case is one way, i.e. solid mechanics is coupled to heat transfer via thermal expansion caused by temperature differences in the model.

In Tutorial 1, we build a cantilever (1000 micrometers in length and 100 micrometers in width; Figure 1) made up of aluminum. The cantilever is heated at one end and maintained at room temperature at the other end. All other surfaces are insulated. We study heat conduction along the length of the cantilever and the associated stresses caused by thermal expansion.

In Tutorial 2, the cantilever is made of two separate materials with widely differing coefficients of thermal expansion (Aluminum and Silicon Oxide). In both cases the physics, boundary conditions and mesh are the same. However, the model exhibits differing behaviors due to the materials that comprise the model.

Figure 1

Theory Background:

In COMSOL, one can see the equations we are solving for via the “Equations” tab on any module. Here we will be solving the equations for heat transfer and solid mechanics via COMSOL. The equations for heat transfer (energy conservation) are:
Panel 1.

This problem involves only conduction. Hence, the first term in Eq. 1 vanishes. There are no heat sources either. Hence Eq. 1 combined with Eq. 2 can be used to determine the heat flux or temperature at any point. Once the temperature at every point is known, the strain ($\varepsilon_{th}$ due to thermal expansion) can be calculated with:

\[ \varepsilon_{th} = \alpha \left( T - T_{ref} \right) \]  

Panel 2.

where $\alpha$ is the coefficient of thermal expansion. The strain enters as a "source" term in the momentum conservation equation that determines the stresses in the material.

Panel 3.

where $S$ in Eq. 4 is the state of internal stress.
COMSOL INSTRUCTIONS --- Cantilever

1. Click on “Model Wizard”
2. Choose 2D
3. Under “Select Physics” choose
   a. Structural Mechanics → Solid Mechanics (solid); click “add”
   b. Heat Transfer → Heat Transfer in Solids (ht); click “add”
   c. Click on “Study”
4. Under “Select Study”, click on “Stationary”
5. Click “Done”

Step 1: Geometry

1. In the model builder (“Model Build”) panel, click on Geometry 1.
2. In the Settings window, click under Length Unit and choose μm.
3. We will build the cantilever geometry as a union of three rectangles. The reason for this is to allow for the flexibility to specify materials and boundary conditions as we explore different aspects of the model.

Rectangle 1:

1. Right click on Geometry, choose Rectangle.
2. In the settings panel for “Rectangle 1”, fill in as follows.
3. Under “Size” in the Settings window:
   a. Width: 1000
   b. Height: 50
4. Click on “Build Selected” at the top (rectangle ‘a’ in Figure 2).

Rectangle 2:

1. Right click on Geometry, choose Rectangle.
2. In the settings panel for “Rectangle 2”, fill in as follows.
3. Under “Size” in the Settings window:
   a. Width: 900
   b. Height: 50
4. Under Position:
   a. x: 0
   b. y: 50
5. Click on “Build Selected” at the top (rectangle ‘b’ in Figure 2)

Rectangle 3:

1. Right click on Geometry, choose Rectangle.
2. In the settings panel for “Rectangle 3”, fill in as follows.
3. Under “Size” in the Settings window:
   a. Width: 100
   b. Height: 50
4. Under Position:
   a. x: 900
   b. y: 50
5. Click on “Build Selected” at the top (rectangle ‘c’ in Figure 2)

Form Union:
1. Click on “Form Union” under “Geometry 1” and select Build All. The resulting geometry is in shown in Figure 2.

Figure 2. Cantilever comprised of three rectangles. In the first part of tutorial, they are all aluminum. In the second part, rectangles b,c are aluminum and rectangle a is silicon oxide.

**Tutorial 1: Aluminum Cantilever**

We first examine the behavior of the cantilever when it is made of a single material, in this case aluminum.

**Step 2: Materials**

1. Under “Component 1” in the Model Builder, right click on Materials under “Geometry 1” and choose “Add Material”.
2. In the “Add Material” window on the right, click on “MEMS”, “Metals” and choose Aluminum, and click on “Add to Component” at the top.
3. In the “Settings” window for Aluminum, it should show the entire geometry (i.e. selection box shows 1, 2, 3).
4. Click on “Add Materials” on the top panel (next to “livelink”) to exit materials selection.

Step 3: Physics and Boundary Conditions:

1. We now need to specify the boundary conditions at all the boundaries for the two physics problems: a) Solid Mechanics and b) Heat Transfer in Solids.

Solid Mechanics:

1. In the “Model Builder”, under “Solid Mechanics”, right click on “Linear Elastic Material 1” and select Thermal Expansion.
2. In the “Settings” window for “Thermal Expansion”, change the setting for “Temperature” from “User Defined” to “Temperature (ht)”. This couples the heat transfer problem to the solid mechanics problem. Specifically, changes in temperature will be reflected in material dimension changes and resulting stresses and strains in the solid mechanics problem.

Fixed Constraint:

1. By default, all boundaries are “Free”, i.e free to move. We change that by fixing the boundaries of the two strips on the left hand side.
2. Right click on “Solid Mechanics” and choose “Fixed Constraint”.
3. In the “Settings” window for Fixed Constraint, choose the boundaries on the left (boundary “B1” in Figure 2) by clicking on the two boundaries.

Heat Transfer in Solids:

In this problem, we consider all the boundaries except for B1 and B2 (in figure 2) insulated. Comsol, by default, treats all the boundaries as insulated. We need to relax this for B1 and B2.

Temperature 1: We now specify the temperature at the left boundary B1.

1. Right click on “Heat Transfer in Solids” and choose “Temperature”.
2. In the settings window for “Temperature 1”, select the boundary by clicking on the left hand boundary B1 (see Figure 2).

Temperature 2: Specify the temperature at boundary B2 as a time varying temperature

1. Right click on “Heat Transfer in Solids” and choose “Temperature”.
2. In the settings window, select the portion of the aluminum strip labeled B2.
3. Under “Settings” for “Temperature 2”, set the temperature to be 400[K].
Step 4: Meshing

1. Now we discretize the computational domain. As this is a 2-dimensional domain made up of rectangular shapes, let us choose quadrilateral elements.
2. Right click on “Mesh 1” in the Model Builder and choose “Free Quad”.
3. Under “Mesh 1”, click on “Size”. In the settings window, under “Element Size”, click under “Calibrate for” and choose “General Physics”. Set the mesh size to “Normal” on Predefined and click on “Build All” at the top.

Step 5: Study 1

1. Right click on “Study 1” and choose “compute”. It should take only a few seconds for the computations to finish.
2. If the computation successfully completes, three plots are produced by default: 1) Stress 2) Temperature and 3) Isothermal Contours.
3. The default plot, which shows the stress, also shows the maximum displacement of the strip, which, in this case is thermal expansion (Figure 4).
4. Before proceeding to the next tutorial, you may want to save the model.

Figure 3
Tutorial 2: Bi-material Cantilever

We now change the materials of the cantilever from Aluminum to a combination of Aluminum and Silicon Oxide. The top half of the cantilever (rectangles b and c in Figure 2) are made of aluminum and the bottom half (rectangle a in Figure 2) is made of silicon oxide. To do this, we revisit the material selection section of Tutorial 1.

Step 1: Materials

1. Under “Component 1” in the Model Builder, under Materials, right click on Aluminum/Aluminum and delete the material.
2. Now choose the material for the strip composed of rectangles “b” and “c”.
3. Under “Component 1” in the Model Builder, right click on Materials under “Geometry 1” and choose “Add Material”.
4. In the “Add Material” window on the right, click on “MEMS”, “Metals” and choose Aluminum/Aluminum, and click on “Add to Component” at the top.
5. In the “Settings” window for Aluminum, first clear selection by clicking on the icon shown by the red rectangle in Figure 4.
6. Choose the rectangles “b” and “c” by clicking on them in the “Graphics” window.
7. We need to choose the material for the lower strip (rectangle a).
8. Click on “Add Material” → “MEMS” → “Insulators” → Silicon Oxide; click on “Add to Component” at the top.
9. In the “Settings” window for Silicon Oxide (SiO2), choose the geometry by clicking on the rectangle “a”; exit materials by clicking on “Add Material” on the toolbar.

---

**Figure 4**
Step 2: Study 1

1. With the materials specified, we can repeat the computation (with the same physics, boundary conditions and mesh).
2. Right click on Study 1 and choose “Compute”.
3. Computations should take only a few seconds. Once they are complete, again three plots are produced by default: Stress, Temperature, and Isothermal Contours. Figure 5 shows the stresses along with displacements. As the cantilever is made of two materials with widely differing coefficients of thermal expansion, the cantilever will bend as shown in Figure 5.
4. Due to the differential heating of the two strips, stresses are generated resulting in stored energy in the material. If the source of heat is removed, the cantilever reverts back to the original position. Thus, by repeatedly heating and cooling the tip of the cantilever, we can convert thermal energy into mechanical energy. This is the inspiration for the MEMS energy harvester in:

https://www.youtube.com/watch?v=DXtmsd5fijo
Figure 5