Homework 2

Taavi Tammaru

February 1, 2024

1 Introduction

The background of the problem: The miniaturization of electronic devices has led to the widespread use of surface-mount components, prompting concerns about durability and the need to shift from lead- or tin-based solder to alternative materials due to issues like temperature-induced solder joint cracks.

So the idea is to do a multiphysics model of a surface-mounted resistor to explore those issues. It models the heat transport and structural stresses and deformations resulting from the temperature distribution using the Heat Transfer in Solids and Solid Mechanics interfaces. In addition we will add an electric potential at the ends of the resistor and observe the results of the different current flowing through it.

2 Implementation of the model

I was given a .mph file that already determined the shape of the resistor and some other properties. Then I had to give measurements to each component. From there I defined the type of materials for each component. I made sure to also have air between the alumina and the circuit board. The object already very cleverly uses mirror symmetry over the y-axis so you have the exact same set of elements on each side. So you have to define the elements only once.

The resistor is defined as the heat source with power of 0.2 W. The equation used to determine the temperature at each point is the widely utilized partial differential heat equation:

$$\nabla \cdot (-k\nabla T) = Q \tag{1}$$

To make this solvable you need boundary conditions given by:

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{\text{inf}} - T) \tag{2}$$

where:
h is $10\,\mathrm{W/(m^2 \cdot K)}$ and
T is 300 K

Then I defined boundaries for the different types of meshes. I used triangular meshes, other shapes could give more accuracy but for our purposes, the triangle is the most efficient option. Using a mesh is the defining feature of the finite element method which takes a physical property evaluation model that in essence has a continuous solution and separates it into discrete parts to get the solution more easily by approximating it.

In the end, the simulations can be run for the temperature distribution and the mechanical stress caused by the change of temperature. Also, simulations can be done of the stress on the solder joints alone to examine them more closely.

Finally, we were asked to introduce a current through the resistor by applying potential difference on the ends of it. and run a simulation of the current through the resistor.

3 The shortcomings of this modeling

Thermal, electrical, and mechanical stress are the most relevant phenomena to analyze but one could definitely benefit from analyzing the reaction to vibrations and humidity. Also, chemical reactions and exposure to magnetic fields could be analyzed for further understanding of the behaviour of the resistor.

There are several errors in the materials section. Namely, Alumina doesn't have a value for electric conductivity and relative permittivity and silver doesn't have a value for relative permittivity as well. Furthermore, Solder doesn't have a value for resistivity temperature coefficient, reference temperature, and relative permittivity.
These things become issues when we want to model the electric flow through these materials. Other simulations should not be affected by these errors. We can fix the issues by providing the necessary values.

4 Most probable failure points

The heat distribution across the surface of the mounted resistor is uniform, thereby mitigating the risk of localized failure points attributable to thermal concentration. As is visible from the figure 1 below:

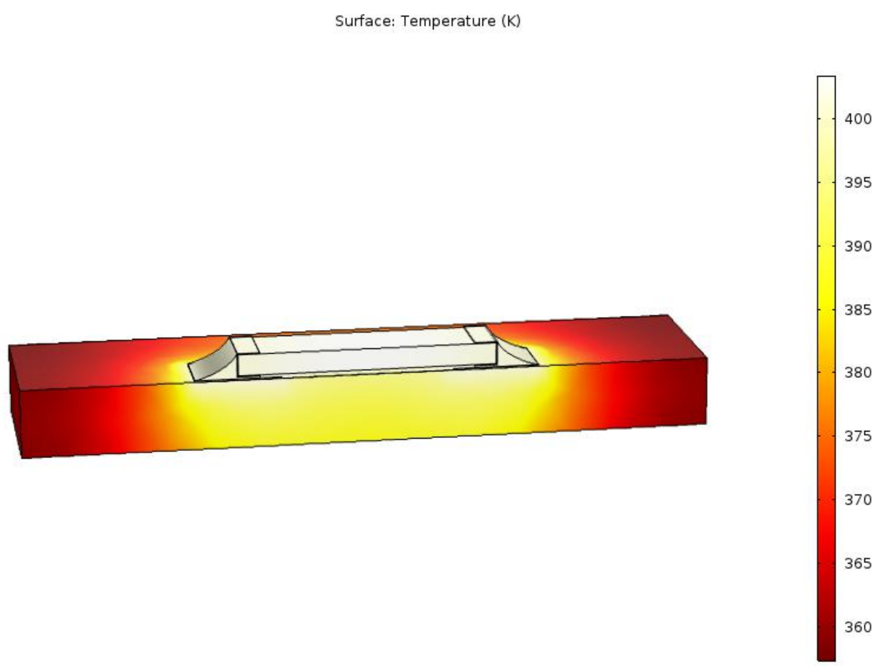

Figure 1: heat distribution

The heat-induced mechanical stress does in fact have possible points of failure. As is apparent from Figure 2 the points of failure are the copper pad and the silver termination that experience considerable force per unit area.

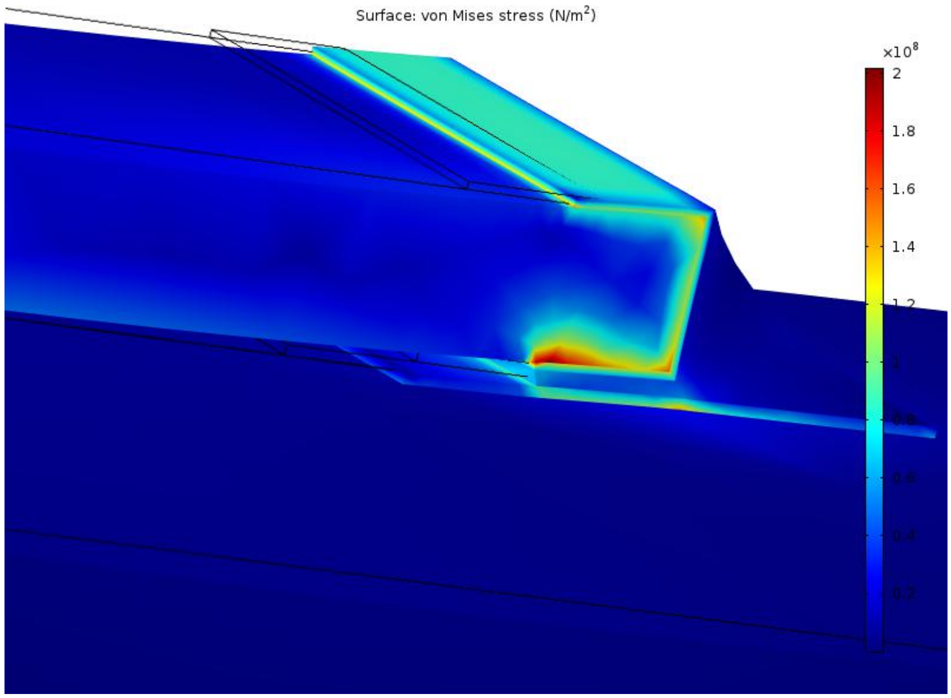

Figure 2: thermally induced stress distribution

If we examine more closely we can also see that the solder joints have a clear point of failure. This is something that could have been missed from the general stress simulation.

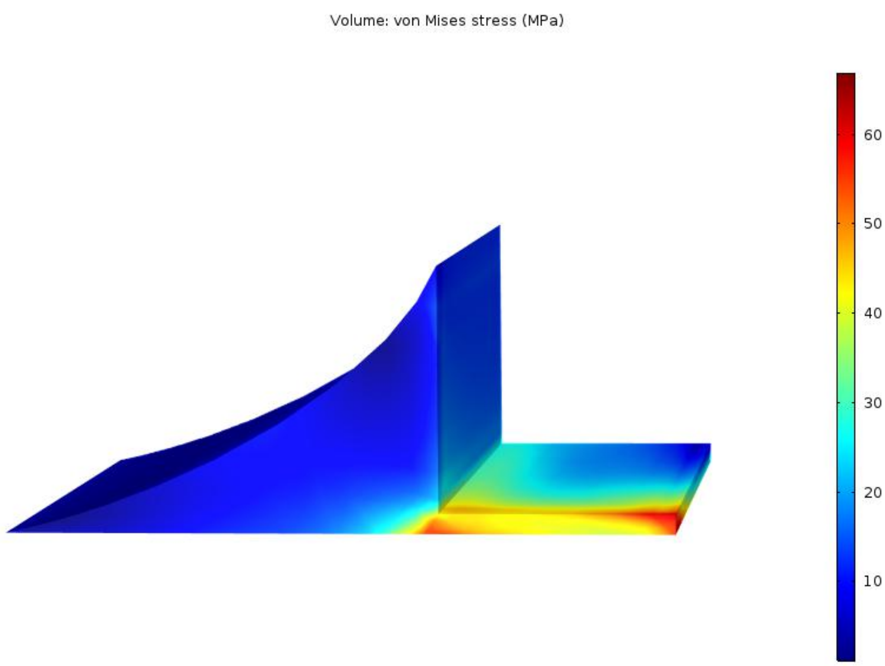

Figure 3: thermally induced stress distribution in the solder joint

5 Changing the mesh density

My personal computer is able to easily handle anything above element size 0.001 but finer than that gets tricky. Below element size 0.001 it takes a while to load the mesh but as I had much unfinished work on my computer I did not want to push it to its complete limits and crash it.

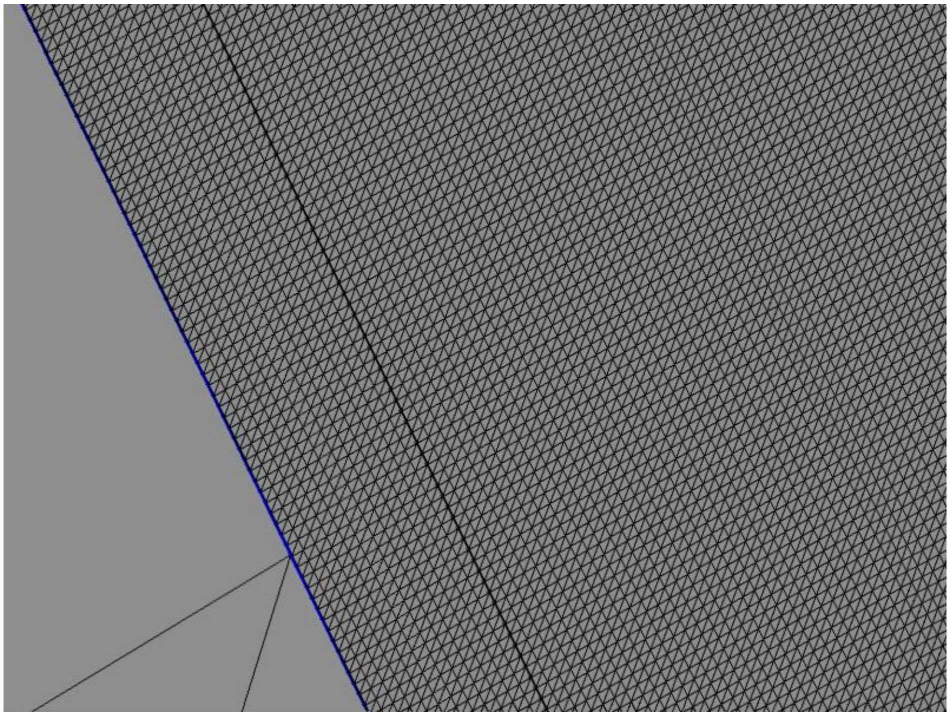

Figure 4: element size 0.001 on the resistor vs element size 0.5 on the board