# Comparing FEA Models for Finned Passive Heat Exchangers

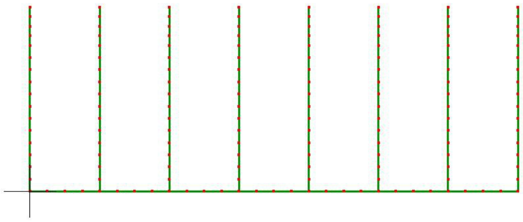
The following summary compares three different FEA model thermal analysis results with a "standard" prediction, including comments on the time required to create each model.

The device to be considered is a commercial black anodized aluminum heat exchanger with 8 fins, 0.06 inches thick spaced on a pitch of .334 inch. The height of the fins is 0.86 inches, the thickness of the heat exchanger base is 0.1 inches and the length of fins and baseplate is 2.28 inches.

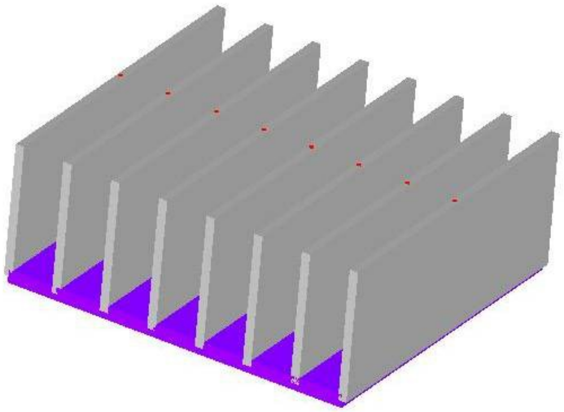
These steady-state models assume vertical fin orientation and sea-level operation in still air at an ambient temperature of 300 degrees Kelvin (radiation ignored for now). Manual calculations predicted a temperature rise of 32.5 degrees for a thermal load of 5 watts located on the baseplate. The convection coefficient was also manually calculated at 6.53 degrees K/watt/meter$^2$ or 0.0042 degrees C/watt/inch$^2$.

*LISA*, a popular finite element analysis application, was used to estimate temperature rise for three different models of the heat exchanger. The three types of models, in order of simplicity and ease of construction, were a line element model, a shell model and a solid model.

For line elements, *LISA* provides a menu of often-used structural shapes, the user need only add the element dimensions in a dialog box plus the thermal conductivity in another dialog box. A line model looks like this in the LISA pre-processing window after meshing:

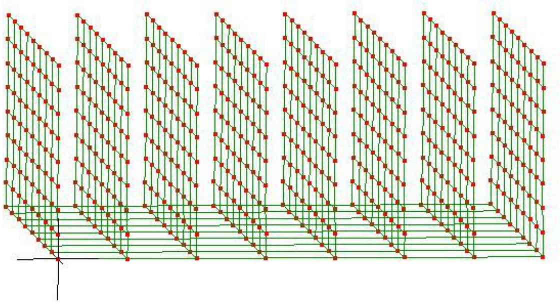


The simplicity and ease of construction of this model is obvious. If one desires to view the elements from any arbitrary angle and at scale, that's also easy and the model would then look like this:

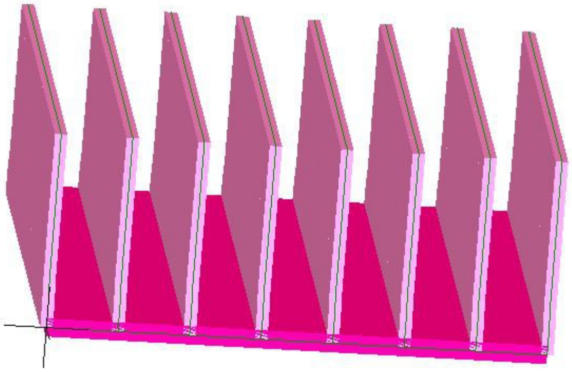


In the above depiction, I chose to color code the two line element types of the model, the baseplate and the fins. It's a user option.

The next simplest model is shell construction, where the thickness of the elements and thermal conductivity are entered in dialog boxes. The planar dimensions are manually created by the user and the model, after meshing, looks like this in the pre-processor display:

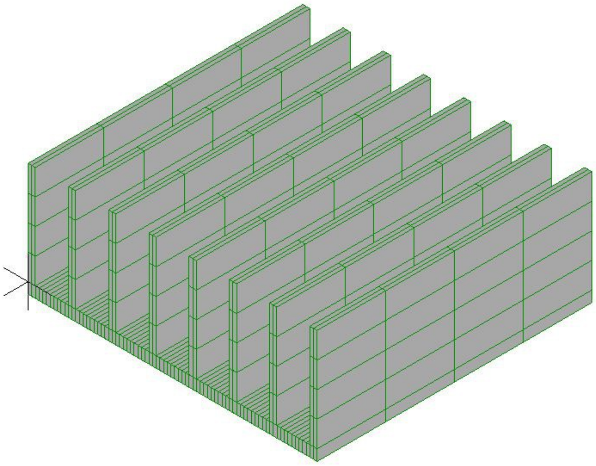


If a more "solid" appearance is desired, a single click produces this view, which can be rotated to any viewpoint:



(As previously noted, coloring and angle of orientation are user-defined.)

The most time-consuming effort is the construction of a solid model. Since all of these models are dimensionally equivalent, the solid model doesn't look much different from the others:



I find creating a shell model to be about two times more time-consuming than the line element model. The solid model requires about twice again the time to construct than similar shell element models. These are obviously personal observations.

When applying boundary conditions to the *line element model*, some care must be exercised. *LISA* selects ALL surfaces of the line element when a "face" selection is made (i.e. both "ends" of the line and all "sides" of the line).

Because the baseplate of this type of heat sink is usually affixed to a housing or chassis, the mounting surface exhibits no convection. Therefore the convection coefficient entered for *this* analysis differs from the other two models. All fins and the uppermost mounting surface have the same convection coefficient for ALL models.

For the line element model only, the baseplate surface convection coefficient must be specified as one-half the value used elsewhere *because we cannot exclude convection from the mounting surface of the baseplate with the face selection tool*. This is just a matter of common sense.

For the other two models, excluding the mounting surface from convection is simple - we just don't select that surface. Internal heat generation was used in each case, the heat source assumed to be the volume of the entire baseplate.

<u>Results of temperature rise analyses:</u>

**Calculated result** (control model): 32.5 degrees C

**FEA Line element model**: 32.4 degrees C

**FEA Shell model**: 32.3 degrees C

**FEA Solid model**: 33.0 degrees C

Agreement is quite good: mean temperature rise is 32.25 degrees C with a standard deviation of 0.3 degrees (0.1 % error). *It is clear that the simple* and very time-efficient *line element model is completely adequate for this type of analysis.*

The FEA models were altered to include radiation then compared with results measured from the commercial heat sink. The heat sink is black anodized so an emissivity of 0.9 was assumed. Internal heat generation was used for the analytical models and area/volume of the heat source modified to resemble the "footprint" of the power resistor mounted to the standard heat sink during measurements.

Because of the small area of the heat source, a calculated "constriction thermal resistance" of 0.246 degrees C/watt was also added to the three model predictions. (Note: this was unnecessary for the previous analysis due to the large heat source area.)

<u>Results of thermal resistance analyses:</u>

**Measured result**: 3.58 degrees C/watt

**FEA Line element model**: 3.59 degrees C/watt

**FEA Shell model**: 3.73 degrees C/watt

**FEA Solid model**: 3.75 degrees C/watt

As with the previous analyses, correlation was solid. Mean thermal resistance is 3.66 degrees C/watt and the standard deviation = .09 (0.3 % error).

Charles R. Couch
Consulting Engineer, RF and Microwave Components and Systems