Printed circuit board layout and manufacture

Resources and methods for learning about these subjects (list a few here, in preparation for your research):

Question 1

When a PCB designer speaks of dimensions in *thous* or *mils*, what does he or she mean?
file 03122

Answer 1

Both a "thou" and a "mil" refer to *one-thousandth of an inch.*

Notes 1

Be sure to remind your students that the "mil" (which is one "milli-inch") is *not* to be confused with a millimeter, which is of course a metric dimension rather than an Imperial dimension.

Question 2

An important parameter when specifying traces on printed circuit boards is the *ounce* rating of the copper used. "1 ounce" copper is very common for general-purpose work. Explain how this "ounce" rating is defined, and why you might want to use "2 ounce" or "4 ounce" copper instead of "1 ounce" for certain traces on your board designs.

file 03121

Answer 2

The "ounce" rating of PCB copper is defined as weight of that thickness of copper trace, for one square foot of trace area. Heavier ounce ratings are used for high-current and/or high-reliability traces.

Notes 2

Ask your students to compare and contrast the "ounce" rating of PCB traces to wire "gauge."

Question 3

What is the difference between a *pad* and a *via*?
file 03120

Answer 3

A *pad* is a ring of copper on one side of the board only, while a *via* is a "double-pad" where two pads on opposite sides of the board are electrically joined through the board to form one connection.

Follow-up question: vias are often used to do "stitching" in multi-layer printed circuit boards. Explain what this means.

Notes 3

This question requires the students to do some research into common PCB layout terms.

Question 4

What are *blind* and *buried* vias, compared to normal vias?
file 03125

Answer 4

A "blind" via is one that connects a surface layer to an inner layer of a multi-layer PCB. A "buried" via is one that does not appear on any surface layer at all, but rather goes between inner layers.

Notes 4

One students grasp the concept of a multi-layer board, the necessity of different types of vias will become obvious. Asking this question, then, is just one more way to communicate the concept of a multi-layer board.

Question 5

Compare and contrast these three techniques for printed circuit board soldering:

- Hand
- Wave
- Reflow

file 03124

Answer 5

Hand soldering is exactly what is sounds like: soldering done manually, by a human being.

Wave soldering momentarily passes the PCB just over a bath of molten solder, allowing the solder to "wick" onto the pads and through via holes.

Reflow soldering uses solder "paste" applied to each pad, and a hot oven to melt the solder for joining components to the board.

Follow-up question: which technique is most appropriate for the following applications?

- Mass production with through-hole components
- Mass production with surface-mount components
- Single-board prototyping

Notes 5

If possible, show videos of automated soldering processes to your students. It is quite educational (and awe-inspiring) for students familiar with tedious hand-soldering techniques to see how large volumes of circuit boards can be soldered using modern wave and reflow techniques.

Question 6

A *solder mask* is a very important feature on professional-quality printed circuit boards. Explain what a solder mask is, and why it is so important.

file 03123

Answer 6

A "solder mask" is a thin coating of polymer covering every part of the board except for those spots where you intend it to be soldered. A solder mask is absolutely essential if the board is to be wave soldered!

Follow-up question: what is the *mask expansion* dimension on a PCB?

Notes 6

Anyone who has ever worked with a home-made, bare-copper (etched) PCB will appreciate the function of a solder mask, even when the soldering is all done by hand.

Question 7

Professionally manufactured printed circuit boards almost never have bare copper exposed, but rather coat it with some other material. Sometimes the coating is tin, while other times the solder mask serves as a coating. Explain why it is important to coat the copper on a high-quality PCB.

file 03126

Answer 7

Copper is very chemically reactive, so leaving it exposed invites corrosion, which will cause a variety of electrical problems. Tin is a much less reactive metal, and solder mask is a polymer which cannot corrode.

Notes 7

Anyone who has ever etched their own board at home (and not coated it) either has seen or *will* see what happens to bare, unprotected copper over time!