

Keywords: PCB, circuit board, prototype, smd, surface mount

Jun 14, 2002

TUTORIAL 1110

Prototyping with Surface-Mount Devices

By: Roger Griswold

Jun 14, 2002

Abstract: Surface-mount devices can often be prototyped without etched printed circuit boards (PCBs) by using plain copper-clad and simple cutting tools.

As surface-mount technology (SMT) has become more prevalent (and many devices are only available in surface-mount), prototyping techniques have changed.

Many engineers and technicians are comfortable with through-hole devices, perforated fiberglass boards, IC sockets, and wire-wrap or point-to-point wiring. Prototyping with surface-mount devices (SMDs) requires different materials and tools. In some ways, these techniques are easier, once the proper skills have been developed.

This application note presents prototyping methods for surface-mount ICs. This technique is readily used for ICs with a lead pitch down to 0.05 inches (1.27mm), and with practice can be used down to 0.025 inches (0.64mm), so it works for well SOICs and virtually all passive SMT components.

The basic idea is to cut narrow slots in a copper-clad board, creating rectangular pads, and place the components across the slots. Each rectangular pad is a node in the circuit. It requires copper-clad board and a rotary hand held grinding tool, plus a few other basic tools.

To illustrate this technique, the circuit of Figure 1, a 5W DC-DC step-down voltage converter, is constructed.

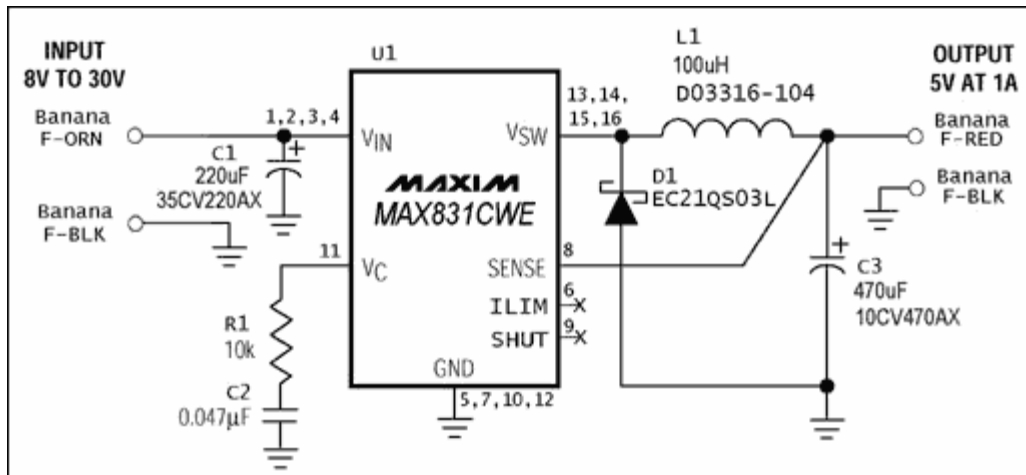


Figure 1. This DC-DC converter circuit is used to illustrate SMT prototyping technique.

Start with a fully annotated schematic and gather the parts so you have a good idea of their relative sizes. It is also helpful to have the pin configuration drawings for any ICs in the circuit. Use blank paper, and two pencils of different colors to make a rough drawing of the physical circuit layout. An eraser is essential, and quite often a few iterations are required. Make the drawing larger than life, but stay more-or-less true to scale. You don't need to be very picky with size, but placement matters. Use the lighter color to draw the component outlines, placing parts that share a node close together where practical.

Once all the part outlines are drawn, use the darker color to draw lines separating the different nodes of the circuit (Figure 2). A typical line will go around a few IC pads, "through" a resistor, and back around (the resistor will straddle the line). The lines represent where cuts to the copper-clad will be made. If only straight lines and 90° turns are used, it makes the cutting easier.

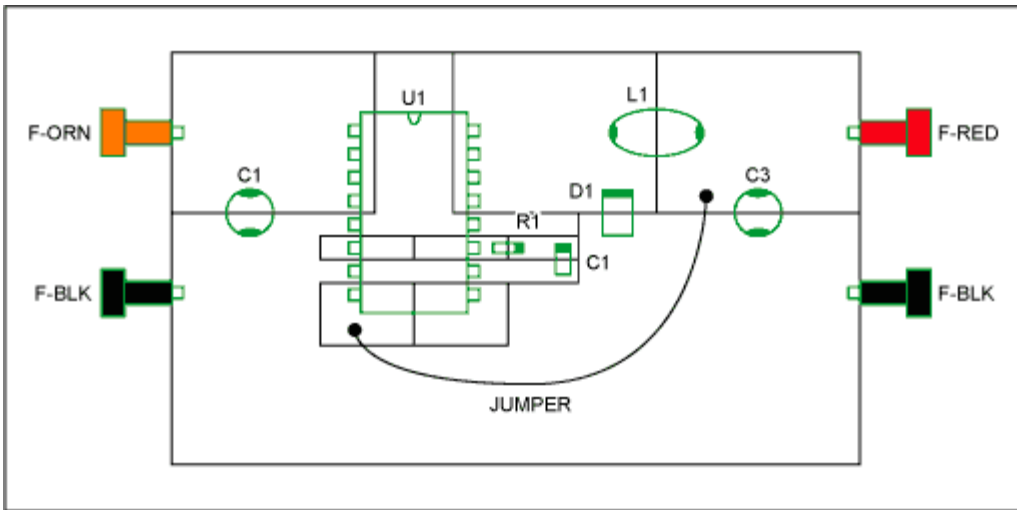


Figure 2. Layout drawing for the SMT prototype of the circuit in Figure 1, showing the component outlines (green) and the node outlines (black). Note that the components bridge the node outlines.

Note that in the circuit implemented here, one jumper wire is used, connecting U1-8 to the output node. The connection could be cut into the copper-clad, but doing so would either interrupt the ground plane or force the trace to run under the IC, neither being desirable. On a multilayer circuit board the trace would likely be placed on a different layer.

To transfer the layout to the copper-clad board, set the board on a flat surface and place the components on it, using the figure as a guide. With an awl or other sharp object, bore a small dot into the copper-clad at points corresponding to the end of each dark line segment in the drawing. This takes some practice because all those little parts tend to bounce around. After all the dots are in place, scribe lines into the copper-clad, connecting the dots as indicated by the drawing.

Now cut the slots into the board. Use a roto-tool with a small, flat tip bit (a Dremel® bit #194 works well, see Figure 3), to cut slots into the copper-clad along the scribed lines. This will also take a little practice. After all the slots are finished, go over them with an X-Acto® knife to cut away any pesky little shorts, double checking the corners where burrs are usually the worst. A once-over with some Scotch-Brite® adds a nice finishing touch. Wash and dry the board, and then use an Ohmmeter to verify that no pad is shorted to any of its neighbors. Now all that's left is to install the components and test the circuit. Figure 4 shows a photo of the slotted board, and Figure 5 shows the finished product.

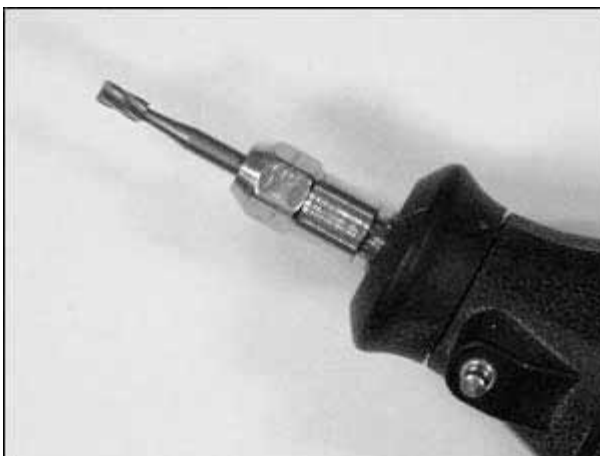


Figure 3. Example of a roto-tool bit useful for cutting slots into copper-clad board. Shown is Dremel bit #194.



Figure 4. The cut circuit board, made using copper-clad and a roto-tool, realizes the layout drawing of Figure 2.

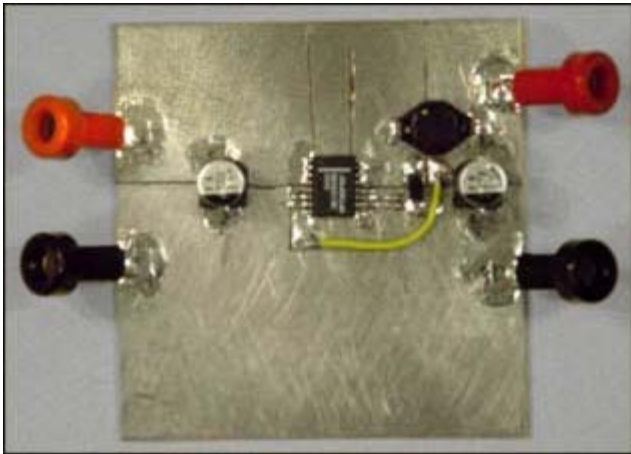


Figure 5. The finished SMT prototype circuit board, a working version of the Figure 1 schematic.

One of the nice features of this technique is the natural ground plane that usually emerges. If you are combining SMT and through-hole technology, through-hole devices can be easily adapted to mount on copper-clad by splaying their legs and soldering them down. Prototyping with SMT is not difficult, and with a little practice many prefer it to wire-wrap or point-to-point wiring.

Another method for prototyping SMDs is to use an adapter board (Figure 6). These adapter boards are readily available from many manufacturers. The SMD is soldered on the adapter board, and a small number of resistors and capacitors can be soldered or connected to the adapter. This method is suitable for testing or prototyping low-power devices, as the traces on these boards are usually thin. For applications that require a solid ground, connect the ground pin of the adapter board to a large copper-clad board with a short wire. The copper-clad board serves as the ground plane for the circuit.

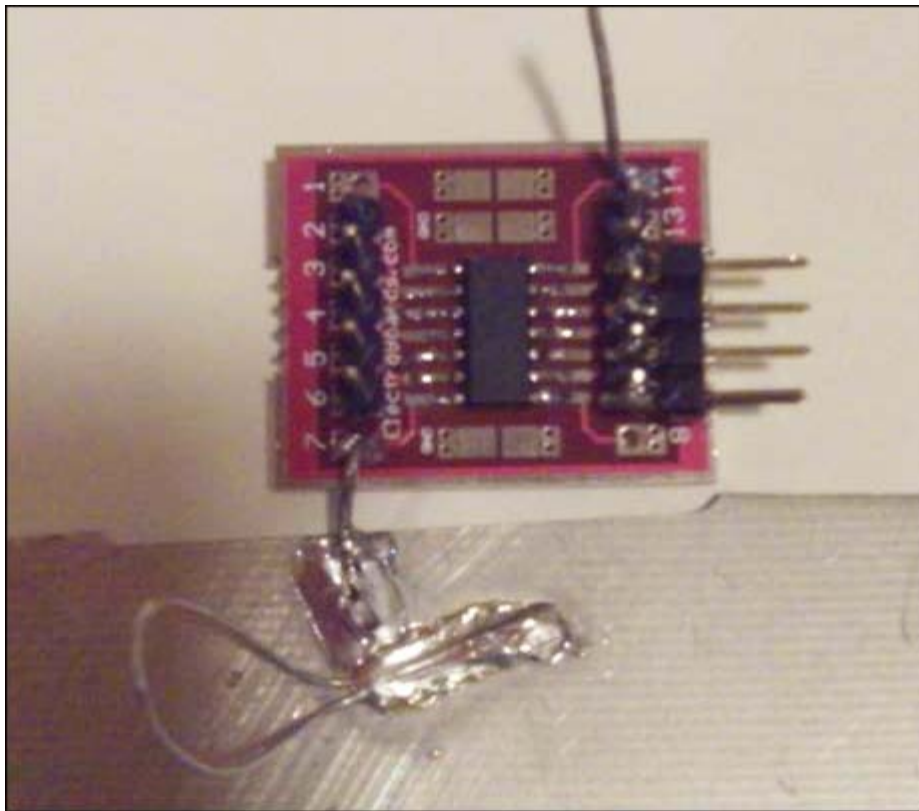


Figure 6. Example of a surface-mount adapter board.

Dremel is a registered trademark of Robert Bosch Tool Corporation.

Scotch-Brite is a registered trademark of 3M Company.

X-Acto is a registered trademark of Elmer's Products, Inc.

Next Steps

EE-Mail [Subscribe to EE-Mail](#) and receive automatic notice of new documents in your areas of interest.

Download [Download, PDF Format \(68kB\)](#)

Share [Other Channels](#) [E-Mail](#) this page to an associate or friend.

More Information

For Technical Support: <http://www.maxim-ic.com/support>

For Samples: <http://www.maxim-ic.com/samples>

Other Questions and Comments: <http://www.maxim-ic.com/contact>

Application Note 1110: <http://www.maxim-ic.com/an1110>

AN1110, AN 1110, APP1110, Appnote1110, Appnote 1110

Copyright © by Maxim Integrated Products

Additional Legal Notices: <http://www.maxim-ic.com/legal>