Making Your Own Printed Circuit Boards

To make a PC board, resist material is applied to a copper-clad bare PC board, which is then immersed into an acid etching bath to remove selected areas of copper. In a finished board, the conductive copper is formed into a pattern of conductors or "traces" that form the actual wiring of the circuit. The following sections describe how to make your own PC boards. A special section by Chuck Adams, K7QO, discusses making your own PC boards from artwork created on a laser printer.

PC BOARD MATERIALS AND SUPPLIES

PC BOARD STOCK

PC board stock consists of a sheet made from insulating material, usually glass epoxy or phenolic, coated with conductive copper. Copper-clad stock is manufactured with phenolic, FR-4 fiberglass and Teflon base materials in thicknesses up to 1/8 inch. The copper thickness varies. It is usually plated from 1 to 2 oz per square foot of bare stock.

RESISTS

Resist is a material that is applied to a PC board to prevent the acid etchant from eating away the copper on those areas of the board that are to be used as conductors. There are several different types of resist materials, both commercial and home brew. When resist is applied to those areas of the board that are to remain as copper traces, it "resists" the acid action of the etchant.

The PC board stock must be clean before any resist is applied. This is discussed later in the chapter. After you have applied resist, by whatever means, protect the board by handling it only at its edges. Do not let it get scraped. Etch the board as soon as possible, to minimize the likelihood of oxidation, moisture or oils contaminating the resist or bare board.

Resist Pens

Several electronics suppliers sell resist pens. Use a resist pen to draw PC-board artwork directly onto a bare board. Commercially available resist pens work well. Several types of permanent markers also function as resist, especially the "Sharpie" brand. They come in fine-point and regular sizes; keep two of each on hand.

Tape

To make a single PC board, Scotch, adhesive or masking tape, securely applied, makes a good resist. (Don't use drafting tape; its glue may be too weak to hold in the etching bath.) Apply the tape to the entire board, transfer the circuit pattern by means of carbon paper, then cut out and remove the sections of tape where the copper is to be etched away. An X-Acto hobby knife is excellent for this purpose.

Paint

Some paints are good resists. Exterior enamel works well. Nail polish is also good, although it tends to dry quickly so you must work fast. Paint the pattern onto the copper surface of the board to be etched. Use an artist's brush to duplicate the PC board pattern onto bare PC-board stock. Tape a piece of carbon paper to the PC-board stock. Tape the PC-board pattern to the carbon paper. Trace over the original layout with a ballpoint pen. The carbon paper transfers the outline of the pattern onto the bare board. Fill in the outline with the resist paint. After paint has been applied, allow it to dry thoroughly before etching.

Rub-On Transfer

Several companies produce rub-on transfer material that can also be used as resist. Patterns are made with various width traces and for most components, including ICs. As the name implies, the pad or trace is positioned on the bare board and rubbed to adhere to the board.

ETCHANT

Etchant is an acid solution that is designed to remove the unwanted copper areas on PC-board stock, leaving other areas to function as conductors. Almost any strong acid bath can serve as an etchant, but some acids are too strong to be safe for general use. Two different etchants are commonly used to fabricate prototype PC boards: ammonium persulphate and ferric chloride. The section "Making PC Boards With Printed Artwork" describes a hydrogen peroxide-muriatic acid etchant. Ferric chloride is the most commonly used etchant and is usually sold ready-mixed. It is made from one part ferric chloride crystals and two parts water, by volume. No catalyst is required and it does not lose strength when stored.

Etchant solutions become exhausted as they are used. Keep a supply on hand. Dispose of the used solution safely; follow the instructions of your local environmental protection authority.

Most etchants work better if they are hot. A board that takes 45 minutes to etch at room temperature will take only a few minutes if the etchant is hot. Use a heat lamp to warm the etchant to the desired temperature. A darkroom thermometer is handy for monitoring the temperature of the bath.

Be careful! Do not heat your etchant above the recommended temperature, typically 160°F. If it gets too hot, it will probably damage the resist. Hot or boiling etchant is also a safety hazard.

Insert the board to be etched into the solution and agitate it continuously to keep fresh chemicals near the board surface. This speeds up the etching process. Normally, the circuit board should be placed in the bath with the copper side facing up.

After the etching process is completed, remove the board from the tray and wash it thoroughly with water. Use a household scrubbing pad, such as Scotchbrite**,** to rub off the resist. Using fine steel wool will also clean the board, but may leave fine steel particles behind.

WARNING: Use a glass or other non-reactive container to hold etching chemicals. Most etchants will react with a metal container. Etchant is caustic and can burn eyes or skin easily. Use rubber gloves and wear old clothing, or a lab smock, when working with any chemicals. If you get some on your skin, wash it with soap and cold water. Wear safety goggles (the kind that fit snugly on your face) when working with any dangerous chemicals. Read the safety labels and follow them carefully. If you get etchant in your eyes, wash immediately with large amounts of cool water and seek immediate medical help. Even a small chemical burn on your eye can develop into a serious problem.

PLANNING A LAYOUT BY HAND

A later section of this chapter explains how to turn a schematic into a working circuit. It is not as simple as laying out the PC board just like the circuit is drawn on the schematic. Read that section before you design a PC board.

Traditionally, amateurs have laid out artwork for the PC board by hand. While this is still quite viable, it can become tedious and error-prone for more complex circuits. There are many PC board layout software packages available for free or at very low cost. For example, the author of the section on making PC boards from printed artwork, Chuck Adams, K7QO, uses *Eagle-5.3.0* layout software running under *Linux*. The free version can be used to lay out board sizes up to 3.25 by 4.0 inches with up to two layers. It has schematic capture, auto-router, and additional output formats to allow you to send files to a commercial PC board manufacturer if you want to do a club project requiring many boards to be made. There are a number of such packages listed in the PCB section of the **Computer-Aided Circuit Design** chapter.

ROUGH LAYOUT

Start by drawing a rough scale pictorial diagram of the layout. Draw the interconnecting leads to represent the traces that are needed on the board. Rearrange the layout as necessary to find an arrangement that completes all of the circuit traces with a minimum number of jumper-wire connections. In some cases, however, it is not possible to complete a design without at least a few jumpers.

LAYOUT

After you have completed a rough layout, redraw the physical layout on a grid. Graph paper works well for this. Use graph paper that has 10 lines per inch to draw artwork at 1:1 and estimate the distance halfway between lines for 0.05-inch spacing. Drafting templates are helpful in the layout stage. Local drafting-supply stores should be able to supply them. The templates usually come in either full-scale or twice normal size.

To lay out a double-sided board, ensure that the lines on both sides of the paper line up (hold the paper up to the light). You can then use each side of the paper for each side of the board. Remember that the side opposite the components (the “circuit side” or “foil side”) will have pin arrangements that are reversed from the component top view! When using graph paper for a PC-board layout, include bolt holes, notches for wires and other mechanical considerations. Fit the circuit into and around them, maintaining clearance between parts.

Most through-hole IC pins are on 0.1-inch centers. Most modern components have leads on 0.1-inch centers. The rows of dual-inline-package (DIP) IC pins are spaced 0.3 or 0.4 inch. Measure the spacing for other components. Transfer the dimensions to the graph paper. It is useful to draw a schematic symbol of the component onto the layout.

The layout of a PCB for surface-mount devices is similar to that for through-hole components except that holes and pads are largely omitted. Traces end where parts are installed. SMT components have much smaller pin spacing than through-hole parts, so more care is required in the layout. If you lay out a board with computer software, the differences in laying out a board are largely transparent. One error that is frequently made by newcomers to SMT board layout is to make the board too compact and not leave room between components for probing.

Draw the traces and pads the way they will look. Using dots and lines is confusing. It’s okay to connect more than one lead per pad, or run a lead through a pad, although using more than two creates a complicated layout. In that case, there may be problems with solder bridges that form short circuits. Traces can run under some components; it is possible to put two or three traces between 0.4-inch centers for a 1/4-W resistor, for example.

Leave power-supply and other dc paths for last. These can usually run just about anywhere, and jumper wires are fine for these noncritical paths. (This is not the case on high-speed digital boards and if EMI from the board is to be minimized.)

Do not use traces less than 0.010-inch (10 mil) wide. If 1-oz PC board stock is used, a 10-mil trace can safely carry up to 500 mA. To carry higher current, increase the width of the traces in proportion. (A trace should be 0.200 inch wide to carry 10 A, for example.) Allow 0.1 inch between traces for each kilovolt in the circuit.

When doing a double-sided board, use pads on both sides of the board to connect traces through the board. Home-brew PC boards do not use plated-through holes (a manufacturing technique that has copper and tin plating inside all of the holes to form electrical connections). Use a through hole and solder the associated component to both sides of the board. Make other through-hole connections with a small piece of bus wire providing the connection through the board; solder it on both sides. This serves the same purpose as the plated-through holes found in commercially manufactured boards.

After you have planned the physical design of the board, decide the best way to complete the design. For one or two simple boards, draw the design directly onto the board, using a resist pen, paint or rub-on resist materials. To transfer the design to the PC board, draw light, accurate pencil lines at 0.1- or 0.05-inch centers on the PC board. Draw both horizontal and vertical lines, forming a grid. You only need lines on one side. For single-sided boards, use this grid to transfer the layout directly onto the board surface. To make drilling easier, use a center punch to punch the centers of holes accurately. Do this before applying the resist so the grid is visible.

When drawing a pad with plenty of room around it, use a pad about 0.05 to 0.1 inch in diameter. For ICs, or other close quarters, make the pad as small as 0.03 inch or so. A “ring” that is too narrow invites soldering problems; the copper may delaminate from the heat. Pads need not be round. It’s okay to shave one or more edges if necessary, to allow a trace to pass nearby.

Draw the traces next. A drafting triangle can help. It should be spaced about 0.1 inch above the table, to avoid smudging the artwork. Use a 9-inch or larger triangle, with a rubber grommet taped to each corner (to hold it off the table). Select a sturdy triangle that doesn’t bend easily.

Align the triangle with the grid lines by eye and make straight, even traces similar to the layout drawing. The triangle can help with angled lines, too. Practice on a few pieces of scrap board.

Make sure that the resist adheres well to the PC board. Most problems can be seen by eye; there can be weak areas or bare spots. If necessary, touch up problems with additional resist. If the board is not clean the resist will not adhere properly. If necessary, remove the resist, clean the board and start from the beginning.

Discard troublesome pens. Resist pens dry out quickly. Keep a few on hand, switch back and forth and put the cap back on each for a bit to give the pen a chance to recover.

Once all of the artwork on the board is drawn, check it against the original artwork. It is easy to leave out a trace. It is not easy to put copper back after a board is etched. In a pinch, replace the missing trace with a small wire.

Applied resist takes about an hour to dry at room temperature. Fifteen minutes in a 200°F oven is also adequate.

Special techniques are used to make double-sided PC boards. See the section on double-sided boards for a description.

MAKING A PC BOARD

Several techniques can be used to make PC boards. They usually start with a PC-board “pattern” or artwork. All of the techniques have one thing in common: this pattern needs to be transferred to the copper surface of the PC board. Unwanted copper is then removed by chemical or mechanical means. Most variations in PC-board manufacturing technique involve differences in resist or etchant materials or techniques.

No matter what technique you use, you should determine the required size of the PC board, and then cut the board to size. Trimming off excess PC-board material can be difficult after the components are installed.

The bare (unetched) PC-board stock should be clean and dry before any resist is applied. (This is not necessary if you are using stock that has been treated with pre-sensitized photoresist.) Wear rubber gloves when working with the stock to avoid getting fingerprints on the copper surface. Clean the board with soap and water, and then scrub the board with #000 steel wool. Rinse the board thoroughly then dry it with a clean, lint-free cloth. Keep the board clean and free of fingerprints or foreign substances throughout the entire manufacturing process.

NO-ETCH PC BOARDS

The most straightforward way to make very simple PC boards is to mechanically remove the unwanted copper. Use a grinding tool, such as the Moto-Tool manufactured by the Dremel Company (available at most hardware or hobby stores). Another technique is to score the copper with a strong, sharp knife, then remove unwanted copper by heating it with a soldering iron and lifting it off with a knife while it is still hot. This technique requires some practice and is not very accurate. It often fails with thin traces, so use it only for simple designs.

PHOTOGRAPHIC PROCESS

Many magazine articles feature printed-circuit layouts. Some of these patterns are difficult to duplicate accurately by hand. A photographic process is the most efficient way to transfer a layout from a magazine page to a circuit board.

The resist ink, tape or dry-transfer processes can be time consuming and tedious for very complex circuit boards. As an alternative, consider the photo process. Not only does the accuracy improve, you need not trace the circuit pattern yourself!

A copper board coated with a light-sensitive chemical is at the heart of the photographic process. In a sense, this board becomes your photographic film.

Make a contact print of the desired pattern by transferring the printed-circuit artwork to special copy film. This film is attached to the copper side of the board and both are exposed to intense light. The areas of the board that are exposed to the light — those areas not shielded by the black portions of the artwork — undergo a chemical change. This creates a transparent image of the artwork on the copper surface.

Figure 1 — PC-board materials are available from several sources. This kit includes layout materials, etchant, and PC board stock.

Develop the PC board, using techniques and chemicals specified by the manufacturer. After the board is developed, etch it to remove the copper from all areas of the board that were exposed to the light. The result is a PC board that looks like it was made in a factory.

Materials and supplies for all types of PC-board manufacturing are available through a variety of electronic distributors. If you're looking for printed-circuit board kits (see **Figure 1)**, chemicals, tools and other materials, review the articles on PC board construction in the ARRL Technical Information Service at **www.arrl.org/tis** in the Radio Technology area under Circuit Construction.

IRON-ON RESIST

An artwork positive is made using a standard photocopier or laser printer. A clothes iron transfers the printed resist pattern to the bare PC board. The technique for transferring the pattern to the blank PC board is described in the section "Making PC Boards With Printed Artwork".

The key to making high quality boards with the photocopy techniques is to be good at retouching the transferred resist. Fortunately, the problems are usually easy to retouch, if you have a bit of patience. A resist pen does a good job of reinforcing any spotty areas in large areas of copper.

DOUBLE-SIDED PC BOARDS

All of the examples used to describe the above techniques were single-sided PC boards, with traces on one side of the board and either a bare board or a ground plane on the other side. PC boards can also have patterns etched onto both sides, or even have multiple layers. Most home-construction projects use single-sided boards, although some kit builders supply double-sided boards. Multilayer boards are rare in ham construction. One method for making double-sided boards is described in the final section of this article, "Double-Sided PC Boards — by Hand!"

TIN PLATING

Most commercial PC boards are tin-plated, to make them easier to solder. Commercial tin-plating techniques require electroplating equipment not readily available to the home constructor. Immersion tin plating solutions can deposit a thin layer of tin onto a copper PC board. Using them is easy; put some of the solution into a plastic container and immerse the board in the solution for a few minutes. The chemical action of the tin-plating solution replaces some of the copper on the board with tin. The result looks nearly as good as a commercially made board. Agitate the board or solution from time to time. When the tinning is complete, take the board out of the solution and rinse it for five minutes under running water. If you don't remove all of the residue, solder may not adhere well to the surface. Immersion tin plating solution is available from electronics vendors.

DRILLING A PC BOARD

After you make a PC board using one of the above techniques, you need to drill holes in the board for the components. Use a drill press, or at least improvise one. Boards can be drilled entirely "free hand" with a hand-held drill but the potential for error is great. A drill press or a small Moto-Tool in an accessory drill press makes the job a lot easier. A single-sided board should be drilled after it is etched; the easiest way to do a double-sided board is to do it before the resist is applied.

To drill in straight lines, build a small movable guide for the drill press so you can slide one edge of the board against it and line up all of the holes on one grid line at a time. See **Figure 2.** This is similar to the "rip fence" set up by most woodworkers to make accurate and repeatable cuts with a table saw.

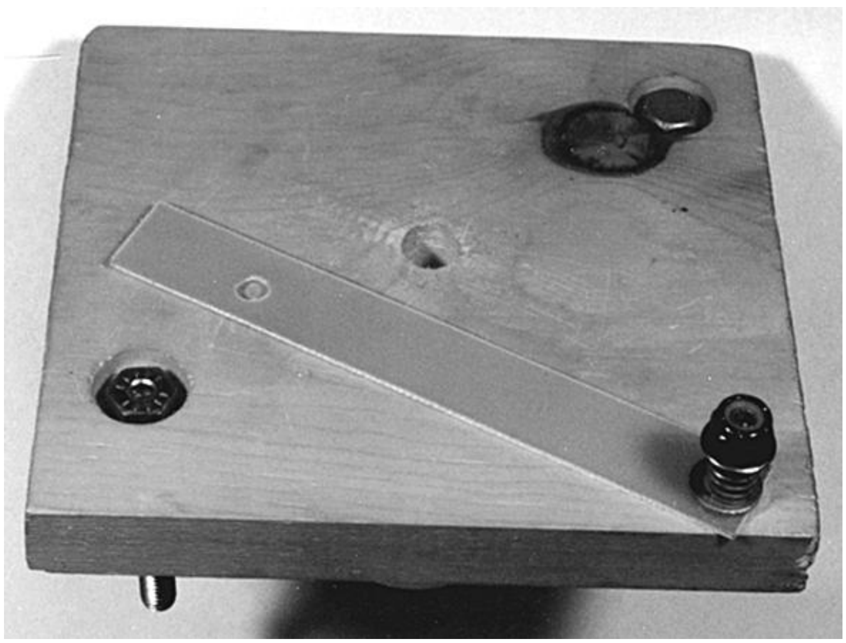


Figure 2 — This home-built drill fence makes it easy to drill PC-board holes in straight rows.

The drill-bit sizes available in hardware stores are too big for PC boards. You can use high-speed steel bits, but glass epoxy stock tends to dull these after a few hundred holes. (When your drill bit becomes worn, it makes a little "hill" around each drilled hole, as the worn bit pushes and pulls the copper rather than drilling it.) A PC-board drill bit, available from many electronic suppliers, will last for thousands of holes! If you are doing a lot of boards, it is clearly worth the investment. Small drill bits are usually ordered by number. Here are some useful numbers and their sizes:

Number	*Diameter (in)*
68	0.0310
65	0.0350
62	0.0380
60	0.0400

Use high RPM and light pressure to make good holes. Safety goggles and good lighting are a must! Count the holes on both the board and your layout drawing to ensure that none are missed. Use a larger-size drill bit, lightly spun between your fingers, to remove any burrs. Don’t use too much pressure; remove only the burr.

Making PC Boards with Printed Artwork

Chuck, K7QO, is a long time homebrewer. He operates what he builds and builds what he operates. Since he was first licensed 50 years ago, he has built rigs using nearly every conceivable construction technique available. He has recently developed this technique for fabricating printed circuit boards at home with easily available materials and a laser printer.

MATERIALS

- 1/4-cup and 1/2-cup plastic measuring cups (do NOT use these cups to prepare food)
- An ordinary clothes iron with a plain metallic ironing surface rather than one with a nonstick coating. The nonstick coating will wear off quickly with use in making printed circuit boards.
- Pyrex dish with 3-cup or 750-ml capacity with plastic/rubber snap-on cover.
- Rubber gloves, heavy duty and acid proof.
- Safety goggles
- Surgical gloves, disposable, for handling boards and chemicals.
- Small plastic tongs for lifting board from harsh chemicals.
- Disposable plastic food forks for moving board around in harsh chemicals.
- A larger glass container with screw-on cover for storage of used etchant.
- Bottle of 2% hydrogen peroxide.
- Clear enamel spray.
- Plastic scrubbing sponge such as ScotchBrite used on kitchen items. (Using steel wool leaves small iron filings behind.)
- Bottle of Tarn-X for cleaning copper.
- Large bag of large cotton balls. This for using both the Tarn-X to remove corrosion from the PC board material and for using acetone to remove the toner after the board is etched.
- Acetone, which can usually be found in the paint department of hardware stores or home improvement stores. Acetone is a hazardous chemical and should be used in a ventilated area and

not breathed. Handle it with rubber gloves as it can cause skin irritation. Keep it tightly capped when not use.

- Muriatic acid. This can be found in the swimming pool section of home improvement stores or in the outdoor section as it is used for cleaning bricks, concrete, and pools.
- Glossy photo-printing paper. Commercial papers that work include Domtar Microprint (32 lb, 112 brightness, 8.5 × 11 in, 20 g/m2) or Staples color laser photo supreme (SKU # 651611). Or you can use a commercial resist paper as discussed earlier in the section.

PC BOARD MATERIAL

One ounce or heavier copper-clad board on FR-4 substrate is best for homebrew PCBs. This is available surplus from a number of channels. The 0.060-inch thickness is best for all-around use.

While Chuck uses a band saw to cut the material into the smaller sizes, a shear, a paper cutter or a hacksaw also work well. The FR-4 substrate material is quite abrasive and will dull blades quickly. Use all safety precautions in using any cutting tool. Use a sanding block immediately after cutting to remove sharp edges along both top and bottom surfaces and the substrate.

INEXPENSIVE AND EFFECTIVE ETCHANT

The secret to etching PC boards is the etchant. Chuck uses a mixture of hydrogen peroxide and muriatic acid. Contrary to what you might think, the acid is not doing the etching. It is the hydrogen peroxide. The oxygen in the peroxide oxidizes the copper and then dissolves into the acidic solution. While ferric chloride is often recommended for etching printed circuit boards, this mixture works as well or better and it is less expensive to make.

The proper ratio of the etching solution is 2:1 for 2 parts hydrogen peroxide to 1 part muriatic acid. To make the solution, while wearing the heavy rubber gloves and safety goggles, first pour 1/2 cup of hydrogen peroxide into the Pyrex container. The order is important. The peroxide goes into the Pyrex container first, then slowly add in 1/4 cup of muriatic acid. This mixture will etch about two double-sided 3.25 × 3.0-inch boards.

Only make a single batch of the hydrogen peroxide/muriatic acid mixture. The mixture will not hold its etching strength for very long, so be frugal and don't make it unless you are going to use it. The bottle of hydrogen peroxide should always be tightly capped and stored in a cool dark place as it readily oxidizes and loses its strength.

TRANSFERRING THE LAYOUT TO THE BOARD

Design your layout. (Don't forget to reserve a small blank area to put your name, call sign and date!) Print the layout on the glossy paper. The printed layout should be a mirror of the final board. The software will do this automatically.

Clean the board with the plastic abrasive sponge. It should be shiny. Put some blank paper under the board to protect the underlying surface.

Carefully place the glossy image with the printed circuit board image on the board where you want it, face down with the toner in contact with the board surface. With the iron on its hottest setting and fully warmed up, carefully iron the image onto the board. Do not use the steam setting. Use heavy pressure; 20 lbs or more is adequate.

Use the pointy end of the iron to apply pressure everywhere on the paper. Don't scrimp here. Try to think about where all the traces are underneath the paper and aim for them all and especially the smallest ones. Patience is key. Be thorough and detailed. This makes a difference in how well your final board will turn out. Don't touch the paper when you are done as it will be hot. Take a break while the paper cools.

After it has cooled, put very hot water in a container and let the board and paper soak for about 15 to 30 minutes. Then come back and carefully peel the paper away from the board. Don't force it. It is easiest to do it from all four edges carefully. Don't touch the toner area. Handle the board by the edges.

Don't try to get all the paper the first time. Just avoid pulling the toner traces off the board. You should not see any toner on the paper being pulled from the board if you are doing this right.

With the heat used to apply the toner to the board, the board will darken and become oxidized. Don't worry about it—that part of the board will be etched away.

After you do the first paper removal, put the board back into some fresh *hot* water and let it soak some 15 minutes more. Then, using only the fingers and your skin, rub the rest of the paper off as much as possible. Don't use your fingernails or anything to help. Doing this under running lukewarm water from the faucet helps. It is not important that you get every bit of paper off the toner. It will look gray or white in some sections because the paper fibers are absorbed or stuck within the toner itself. Some people use an old toothbrush for this.

ETCHING THE BOARD

Take the board and place it in the Pyrex dish with the peroxide/muriatic acid mix. You'll know you are doing this correctly and the mix is right when the board starts to darken almost immediately. This is the copper oxidizing. The clear liquid will start to turn a light green with dissolved copper in short order. Stay away from the fumes and stand upwind from any draft and do this in a well ventilated area.

Use the two plastic forks to gently push the board back and forth in the mixture, being extra careful not to touch the toner. If it comes lose, then that area of the copper will be etched away.

Move the board around until all the copper is etched away from the board except the areas covered by the toner. Don't go off and leave and forget about the etching process as it will ruin the board. Take the board out using the tongs and wash it under running water to stop the etching process.

FINISHING THE BOARD

To remove the toner after the etching process, use acetone and cotton balls, while wearing surgical gloves. A well-ventilated area is a must for this. Do it outdoors if possible. This procedure takes some practice, as sometimes the toner smears and sticks to the board a little if not enough acetone is used and kept on the surface. Don't overdo it, but practice on a few boards to get the hang of it.

After you get the toner off and rinse the board, use cotton balls and the Tarn-X to clean the copper surface.

Now the surface is exposed to the nasty atmosphere of the Earth. Protect it by very lightly coating the surface with the clear enamel in a clean, dust-free, well ventilated area also. This coating protects the surfaces and it also acts as a solder mask.

Chuck uses a 0.7 millimeter silicon carbide drill bit in a Dremel Moto-Tool mounted in a drill press to drill holes for through hole parts. A drill press is necessary as the drill bit is very small and easily damaged if you do this by hand. The 0.7 mm drill will work for most of the holes. One can go to a larger sized bit for parts that have larger leads.

You are now ready to stuff the board with your parts. Congratulations — you have just designed, fabricated and built a printed circuit board!

Double-Sided PC Boards — by Hand!

Forget those nightmares about expensive photoresists that didn't work; forget that business of fifty bucks a board! You don't need computer-aided design to make a double-sided PC board; just improve on the basics, and keep it simple. Anyone can make low-cost double-sided boards with traces down to 0.020 inch, with perfect front-to-back hole registration.

To make a double-sided board, drill the holes before applying the resist artwork; that is the only way to assure good front-to-back registration. The artwork on both sides can then be properly positioned to the holes. PC-board drilling was discussed earlier in the text.

After you have drilled the board, clean its surface thoroughly. After that, wear clean rubber or cotton gloves to keep it clean. One fingerprint can really mess up the application of resist or the etchant.

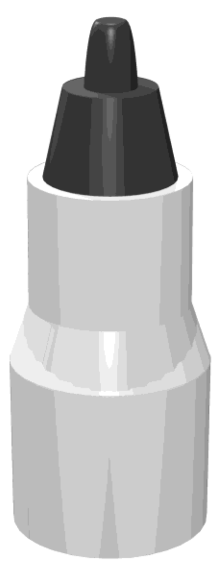


Figure 3 — Make a permanent marker into a specialized PC-board drawing tool. Simply press the marker point into a drilled hole to form a modified point as shown. More pressure produces a wider shoulder that makes larger pads on the PC board.

Tape the board to your work surface, making sure it can't move around. Transfer the artwork from your layout grid to the PC board, drawing by hand with a resist pen.

Allot enough time to finish at least one side of the artwork in one sitting. Start with the pads. To make a handy pad-drawing tool, press the tip of a regular-size Sharpie indelible ink felt-tip marker into one of the drilled PC-board holes. This "smooshes" the tip into the shape of the hole, leaving a flat shoulder to draw the pad. See **Figure 3**. The diameter of the pad is determined by how hard the pen is pressed; pressing too hard forms a pad that is way too large for most applications. Practice on scrap board first. Use this modified pen to fill in all the holes and draw the pads at the same time. Use an unmodified resist pen to draw all of the traces and to touch up any voids or weak areas in the pads. For the rest of the drawing, the procedure described for single-sided boards applies to double-sided boards, too.

After the resist is applied to the first side, carefully draw the second side. Inspect the board thoroughly; you may have scratched or smudged the first side while you were drawing the second.

Etching a double-sided board is not much different than etching a single-sided board, except that you must ensure that the etchant is able to reach both sides of the board. If you dunk the board in and out of the etchant solution, both sides are exposed to the etchant. If you use a tray, put some spacers on the bottom and rest the board on the spacers. (The spacers must be put on the board edges, not where you want to actually etch.) This ensures that etchant gets to both sides. If you use this method, turn the board over once or twice during the process. — *Dave Reynolds, KE7QF*

OR PHOTO-ETCHED

You can also make double-sided boards at home without drawing the layout by hand. This procedure can't produce results to match the finest professionally made double-sided boards, but it can make boards that are good enough for many moderately complex projects.

Start with the same sort of artwork used for single-sided boards, but leave a margin for taping at one edge. It is critical that the patterns for the two sides are accurately sized. The chief limiting factor in this technique is the requirement that matching pads on the two sides are positioned correctly. Not only must the two sides match each other, but they must also be the correct size for the parts in the project. Slight reproduction errors can accumulate to major problems in the length of a 40-pin DIP IC. One good tool to achieve this requirement is a photocopy machine that can make reductions and enlargements in 1% steps. Perform a few experiments to arrive at settings that yield accurately sized patterns.

Choose two holes at opposite corners of the etching patterns. Tape one of the two patterns to one side of the PC board. Choose some small wire and a drill bit that closely matches the wire diameter. For example, #20 AWG enameled wire is a close match for a #62 or a #65 drill, depending on the thickness of the wire's enamel coating. Drill through the pattern and the board at the two chosen holes. Drill the chosen holes through the second pattern. Place two pieces of the wire through the PC board and slide the second pattern down these wire "pins" to locate the pattern on the board. Tape the second pattern in position and remove the pins. From this point on, expose and process each side of the board as if it were a single-sided board, but take care when exposing each side to keep the reverse side protected from light.

Reflow Soldering for the Radio Amateur

How to use modern production soldering techniques at home.

Jim Koehler, VE5FP

I've always been interested in homebrewing equipment and I think I have probably tried every form of electronic assembly at one time or another. As a young lad, I built crystal sets on wooden planks using wood screws to hold down components; this is the classic *breadboard* form of construction. After I got my first license in 1952, and for many years afterward, I constructed vacuum tube equipment in what was then the standard way — components mounted on a metal chassis with punched holes for tube sockets. When transistors came along in the late 1950s and early 1960s, I gradually moved in using printed circuit (PC) boards. Later, in my career as an experimental physicist, I designed electronic equipment using PC boards and oversaw the construction of complex electronic devices using these boards, all using through-hole mounted components.

After I retired, I continued to design and build equipment using PC boards but, despite reading an excellent introduction to amateur use of surface mounted components, I resisted using them for several years because it just looked to be too difficult.[1] Nevertheless, it was becoming apparent that the latest devices were likely to be only available in surface mounted packages.

I decided to see if I really could assemble surface mount components to a PC board. To my great surprise, I found that it is easier and quicker to assemble a PC board using surface mount components than it is using through-hole parts. If you're designing boards, there is the additional advantage that surface mount boards can be smaller, lighter and cheaper to fabricate because when having them made, you pay at a rate depending on the surface area of the board. Now I wouldn't dream of going back to through-hole components on a PC board.

It really is not difficult to acquire the necessary skill in hand soldering surface mount components if they are not too small; I try to design boards for 0805 sized resistors and capacitors; these are 0.08 × 0.05 inches length × width. With a fine-tipped soldering iron, a low-power dissecting microscope and a little practice, even my somewhat shaky hands, combined with 72 year old hand-eye coordination can do it quickly and reliably.

There is nothing I can add to the excellent instructions given by Sam Ulbing, N4UAU, in the earlier referenced article, except to say that high magnification, in the form of a dissecting microscope, is an invaluable aid. I find that 7 power is about right. The large distance between the board and the objective lens of the microscope means that you can easily manipulate soldering iron, solder and tweezers in the field of view. It is not the aim of this article to proselytize surface mount usage, however, but rather to describe how to go to the next step.

While it is easy to hand solder a board, it becomes a bit tedious if you decide to make the same board several times over, perhaps for friends or as part of a club project. That is when you start thinking about mass production techniques.

Reflow Soldering

Reflow soldering is the most common method of producing surface mount PC boards in commercial quantities. It is the technique that is universally used to make computer equipment, radios, MP3 players and, indeed, virtually all modern electronic equipment. You can see an example of the results of this method by looking inside any

Figure 2 — Inexpensive multimeter with type-K thermocouple probe.

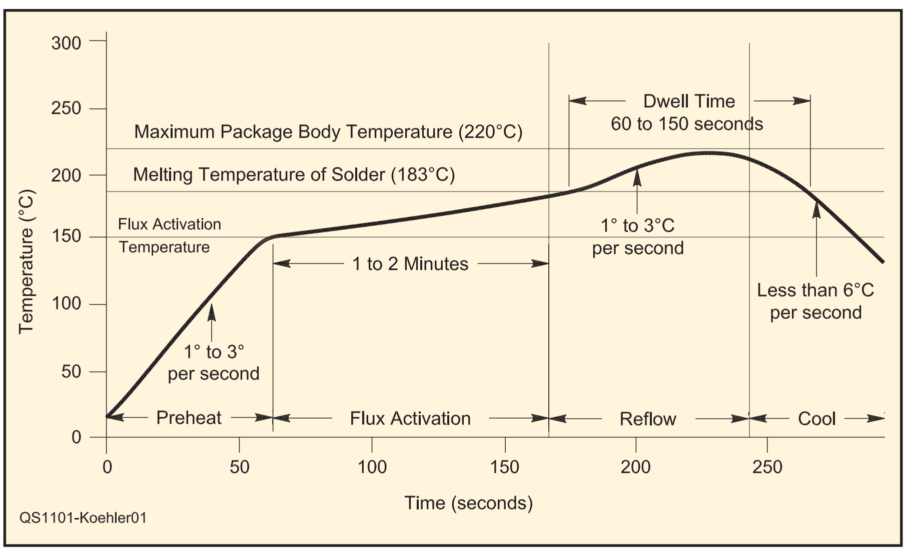


Figure 1 — Temperature profile redrawn from data in Altera Application Note AN081.

[1]Notes appear on page 35.

piece of mass-produced electronic hardware that has been built in the last 5 to 10 years.

Reflow soldering uses solder paste, a mixture of microscopic spheres of solder mixed with a semi liquid, viscous flux. A small dab of solder paste is placed on each pad of the PC board and then the components are all placed onto the pads. Finally, the board plus solder paste and components is placed into an oven. The oven heats the boards and components until the solder paste turns into liquid solder. The board is then allowed to cool and the solder solidifies again. Because all the soldering is done at the same time, the process can be very quick.

Of course, it is not quite that simple. The process of heating and cooling the board with its load of solder paste and components needs to be fairly closely controlled in order to get reliable results. Figure 1 shows the temperatures needed in a typical cycle of heating and cooling. As you can see, the board and its load are first heated quickly to a point close to the melting point of solder. It is then held at this temperature for a time to allow all the components to get to the same temperature — this is called the heat *soak* period. The oven is then raised to a temperature high enough to ensure that all the solder melts and, with the melted flux, forms a good bond. Then the whole assembly is cooled relatively quickly, but not so quickly that it will cause thermal stresses that could break some components. The whole process is over in just a few minutes.

The commercial ovens used for reflow soldering are large, complex devices costing tens of thousands of dollars. In them, the PC board travels on a conveyor belt through regions of different temperature in order to produce the temperature sequence shown in Figure 1. This is somewhat similar to the way pizzas are made in the big franchised pizza stores.

There are also smaller tabletop versions of reflow ovens, for amateur use or for small production lines, that go through a heating and cooling cycle with the PC board stationary.[2] I am going to describe how you can build an even simpler, but still usable, reflow oven in an afternoon and for little cost and trouble.

The Poor Man's Reflow Oven

One day, while shopping for groceries, I saw a pallet of Black and Decker toaster ovens (catalog number TR0964) in a local supermarket for $20 each. They appeared to be made of stainless steel, their power rating was 1.2 kW and the oven was not too large. It occurred to me that, with the aid of a simple electronic thermometer, one could easily approximate the type of heat cycle shown in Figure 1, without the complexity of a PID (proportional integral derivative) controller.

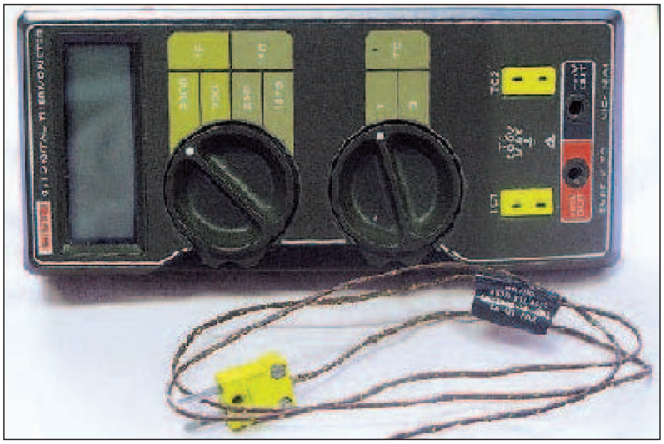


Figure 3 — Keithley electronic thermometer with type-K thermocouple probe.

Figure 4 — Tip of type K thermocouple from Keithley instrument with a pin for comparison.

Besides an electric toaster oven, you need an electronic thermometer. These are widely available at low cost. A few years ago, I bought a digital multimeter at a discount electronics warehouse in Phoenix that had a type K thermocouple temperature probe. It is shown in Figure 2. I have recently seen a similar one advertised online for about $25. On another occasion, I bought a used Keithley electronic thermometer, also with a type K thermocouple probe, on an Internet auction site for about $25 (see Figure 3).

The active part of these thermometers is a thermocouple, the tiny dissimilar metal junction at the tip of the probe as shown in Figure 4. As you can see, these probes are small and so require little heat to come to equilibrium with the temperature of the air. This means that the temperature of the probe tip is going to be close to the temperature of the air in its vicinity.

The idea I had was to drill a small hole in the back wall of the toaster oven and insert the thermocouple probe into it. Then, monitoring the temperature of the inside of the oven, I would just manually turn the oven on and off in order to approximate the heating profile needed to do the reflow soldering. I tried it, it worked wonderfully and it was simple to do.

Modifying the Toaster Oven

The toaster oven I bought was made by Black and Decker. It has two heating bars inside the oven, one at the top and one at the bottom. Both have a metal strip between them and the interior of the oven to provide some shade from the element so that things put inside the oven will not be heated directly by radiation but, instead, by convection of the air.

I made a small bushing from a piece of Teflon that I placed into a ¼ inch diameter hole I'd drilled in the back wall of the oven. This bushing is just to protect the insulated covering of the wire to the thermocouple probe from chafing at the edges of the hole. A small hole was drilled in the bushing to allow the probe to be inserted into the oven. The bushing is shown in Figure 5. Figure 6 shows the bushing inserted into the back wall of the oven. If you can't make a similar bushing, you could get the same result with a piece of sheet Teflon bolted to the rear panel and with a small hole in it going through a larger hole in the metal wall.

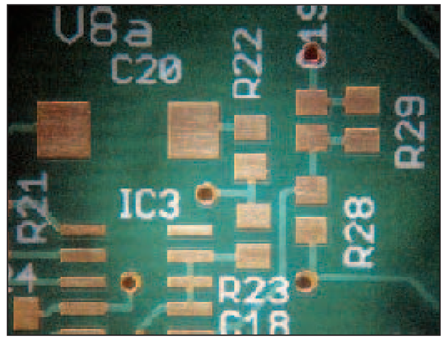
The oven door had a hook that connected to a sliding grill so that when the door was opened it would pull the grill forward slightly for easier access. I didn't want to have the grill move when the door was opened because I didn't want to shake the board while the solder was melted. I bent the hooks over so that they would not interfere with the grill. I did not use the metal pan that came with the oven.

The probe of the thermometer is inserted into the hole in the bushing and the wires are bent so that the junction at the tip of the probe is located about ½ inch above the surface of the grill and near the center. Circuit boards will be placed just under this probe tip. That completes the preparation of the oven.

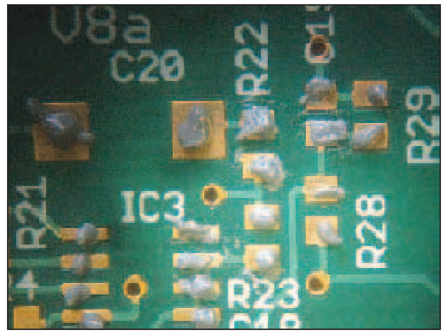
Applying the Solder Paste

Solder paste can be bought in small quantities from suppliers such as Digikey, Mouser or Allied. If you order it, you will get a warning that it can only be shipped by courier because the usable lifetime of the paste is short unless it is refrigerated. My impression is that this warning is there because the viscosity and texture of the paste are important in the commercial methods used to put it on the pads of the PC boards. Also, if the temperature gets too warm, the little balls of solder tend to separate from the flux paste in which they are embedded. For an amateur, the exact viscosity is not

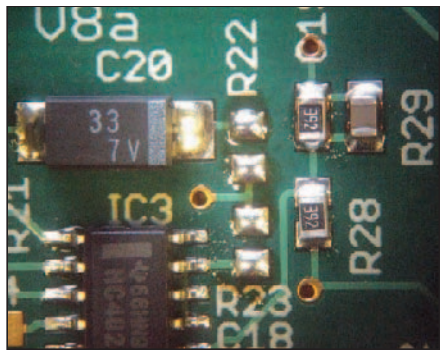
Figure 5 — Teflon bushing ready to be inserted in hole in rear of oven.



(A)



(B)



(C)

Figure 7 — Soldering the PC board in three stages. At (A) the bare board, at (B) the board with solder paste applied by hypodermic syringe, at (C) the board with the components in place after reflow soldering.

important. I have had the same 35 gram tube of solder paste in my refrigerator for several years and it seems to work just as well today as it did when I bought it. I probably have made several dozen boards and still have only used a fraction of the 35 grams.

In commercial houses, solder paste is placed on the board by using stencils with cutouts over all the pads that are to be soldered. Then solder paste is squeegeed over it in the same manner as silk screen printing is done. A method used in large scale production is to have a machine controlled head move over each pad on the PC board and dispense a fixed amount of solder paste through a nozzle. Neither of these methods is really suitable for an amateur who wants to make a few boards.

I bought a small hypodermic syringe at a drug store and ordered a few dispensing tips from Digikey (**www.digikey.com**). I found that a tip with a #18 AWG hole was suitable for putting solder paste onto the pads for 0805 sized components — this tip has Digikey part number KDS18TN25. I take the tube of solder paste from the refrigerator and place a small amount into the hypodermic syringe with the dispensing tip on it. I then place the tube back into the refrigerator for future use.

Then looking at the board with my dissecting microscope and using the hypodermic syringe with the dispensing tip, I place a little dab of solder paste onto each pad. I find that the pressure needed to dispense a tiny bit of solder paste onto a pad with the hypodermic syringe is quite a lot unless you let the solder paste warm up a bit. It doesn't have to be very hot; if you hold the syringe in your hand for a while so that the solder paste comes up to body temperature, it seems to work just fine.

Figure 7 shows a sequence of three photos of a small section of a PC board. In

Figure 6 — Teflon bushing in the back of the oven.

7(A), you see the bare pads where surface mount components will be placed. In 7(B), you see a small blob of solder paste applied to the pads using a hypodermic syringe as described above. Figure 7(C) shows the resulting bond to the surface mounted components after the oven has been used as I will describe in the next section. The solder joints are quite nice, clean and electrically sound.

Calibrating the Oven

Before you use the oven, it is necessary to make one calibration measurement. If the oven is turned on, the temperature will start to rise and will finally get to a relatively constant rate of increase of temperature; so many degrees per second. If it is then switched off, the temperature inside the oven will continue to rise because of the thermal inertia of the heating elements. It is necessary to measure how far the temperature *coasts* after turning off the oven. So, you need to do the following:

- Turn on the oven heating elements,
- Monitor the temperature and, when the temperature gets to 180°C,
- Turn off the heating elements and observe the maximum temperature that the oven reaches.

Make a note of the amount of this temperature overshoot. For example, if you turned the oven off at 180°C and it reached 195°C, then the overshoot was 15°C. This amount of overshoot will be approximately valid for the temperature region around 180°C.

Making Your First Reflow Soldered Board

First, you need to prepare the PC board with solder paste, as described above, put the components in their places and then put the board into the oven, being very careful not to jostle the components from their places on the board. Then carefully close the oven door and turn on the heating elements. Watch the temperature increase and turn the heating elements off when the temperature gets to 170°C minus the overshoot (for example, if the overshoot is 15°C, then turn off the elements at 155°C). As you now watch the temperature, it will coast up toward 170°C.

This pause in the heating will produce a plateau in temperature similar to the one in Figure 1 before the peak. As the temperature starts to slow down in its approach to 170° but a few degrees before it actually gets there, you can turn on the heating elements again and the temperature will start to rise again. Watch the temperature and when it gets to 220° minus the overshoot (in our example, this would be 205°C), turn off the heating elements again. Now, the temperature should coast up to 220°C and then start to fall. You will now have gone past the temperature peak shown in Figure 1. As it keeps cooling, when it gets down to 160°C, you can open the door to the oven to allow the board to cool off more quickly. You're done! When the board has cooled off enough to handle, take it out and inspect the solder joints to make sure they are all good — they should be.

The reflow soldering process takes just a few minutes so it is a good way to make a large number of boards in a batch. You do not have to wait till the oven cools down to room temperature between boards. As soon as it gets cool enough so you can put in another prepared board without burning yourself, you can start the process again.

Good luck and happy reflow soldering!

Notes

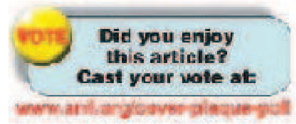
[1]Sam Ulbing, N4UAU, "Surface Mount Technology — You Can Work with It," *QST*, Apr 1999, p 33. This is part one of a four part article.

[2]Elektor SMT Reflow Oven, Elektor Shop, **www.elektor.com/shop**.

Photos by the author.

ARRL International Member Jim Koehler, VE5FP, was first licensed in 1952 at age 15. He graduated from the Australian National University with a PhD in astronomy and went on to become a professor of physics and engineering physics at the University of Saskatchewan in Canada. He taught classes in instrumentation design and conducted research in upper atmospheric physics until he retired in 1996.

Jim now lives on Vancouver Island where he indulges in his hobbies of electronics, model aircraft and photography. You can reach Jim at 2258 June Rd, Courtenay, BC V9J 1X9, Canada or at **jark@shaw.ca**. QST



Going Once, Going Twice, GONE!

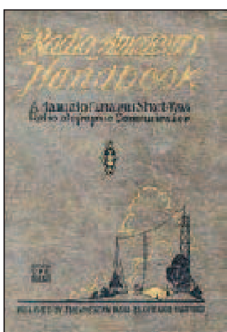
In its five year history, the ARRL On-Line Auction has raised almost $200,000 to promote educational activities.

S. Khrystyne Keane, K1SFA
ARRL News Editor

With almost 200 items up for bid — and almost 1000 bids placed — the Fifth Annual ARRL On-Line Auction closed on October 25 with winning bids on every auction item. ARRL Business Services Manager Deb Jahnke, K1DAJ, was happy with the responses the auction received during its run. The generosity of many donors, Jahnke says, made it possible for the auction to offer a diverse list of items that included transceivers, ARRL Lab-tested and reviewed equipment, vintage gear, one-of-a-kind treasures and mystery "junque" boxes.

Items that appeared in *QST*'s Product Review column included an RF Concepts Alpha 9500 HF Linear Amplifier, a Yaesu FTdx9000MP and an ICOM ID-880H Dual Band Transceiver with D-STAR. In all, 38 product review items were sold. The Yaesu FTdx9000MP attracted five bids and sold for the highest amount in the Auction — $7680. Among the items in this year's Auction was a rare First Edition *ARRL Handbook*. Originally listed for $40, this collectible book from 1926 sold for $775 after garnering 22 bids, the most in the auction!

Among the most popular items in this year's auction were the three "junque boxes," donated by the ARRL Lab. Garnering 38 bids between them, these Amateur Radio treasure troves went for more than $200 each, raising almost $650. "We featured the 'junque boxes' in our first auction, and they proved to be extremely popular with our bidders," Jahnke said. "We've brought them back each year, and once again, we couldn't believe how they were all the rage. I can't even begin to describe how well received they were this year." The contents of each box are a mystery, Jahnke said, known only to the ARRL Lab staff. "And they won't tell!" she said.



After 22 bids, this First Edition *ARRL Handbook* from 1926 went for $775.

Jahnke said that donors also added to the Auction excitement by donating gear and products and services. Thanks to our 2010 ARRL On-Line Auction donors: A&A Engineering, ASA Inc, Barker Specialty Company, Engraved Memories, Hotpress Ham Hats, K8RA Iambic & Single Lever CW Keys, KB3IFH QSL Cards and TNT Electrical Trades Gift Store. Special thanks to Hi-Q Antennas, the ARRL On-Line Auction Golden Gavel Sponsor.

Proceeds from the auction benefit various ARRL education programs, as well as fund efforts to license new hams, programs to strengthen Amateur Radio's emergency service training and the creation of new instructional materials. The proceeds also allow the League to offer continuing technical and operating education. QST

A No-Special-Tools SMD Desoldering Technique

You don't need a fancy fabrication shop to get surface-mount devices off the board.

Wayne Yoshida, KH6WZ

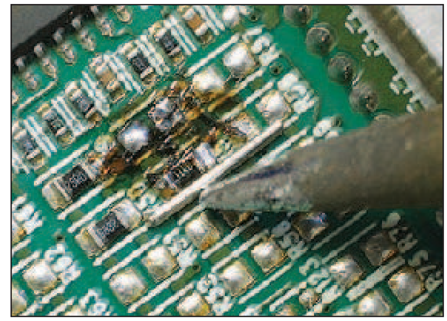


Figure 1 — A small piece of bus wire is used to transfer heat to both sides of the component at the same time.

While modifying a commercial 23 GHz assembly for ham radio use on the 24 GHz band, I had to remove and replace several small surface-mount devices (SMDs). Although several techniques have been described to remove an SMD from circuit boards, I have developed a way to remove these tiny parts without any special tools or a need for a second soldering iron. In addition, this technique does not destroy the component or the circuit board.

Flood, Float and Flick

I call this the *flood, float and flick* (*F3* for short) technique: The component and board are heated, and as the solder begins to melt, more solder is added to the area, flooding the component area. Then, a flick of the iron tip moves the part so that it can be picked up with a pair of tweezers. I use a standard Weller WTCPT soldering iron, with an 800° screwdriver tip for all work.

The main idea is to heat all sides of the component at the same time, so that the device can be picked up and removed from the circuit board. Since surface-mount components are so small, it may be possible to use a large chisel-point soldering iron tip to heat both sides of the component.

The general concept is based on the old trick of wrapping a piece of solid copper wire around a soldering iron tip, to solder small components. In this technique, a small piece of 20 gauge tinned bus wire is used to transfer heat to both sides of an SMD resistor, as shown in Figure 1. In this photo you can see that a component has already been removed, using this technique.

Although I have not tried this, it should be possible to shape the wire into a small circle, rectangle or square to remove other components, such as SOT-223 packaged transistors or SMT DIP packaged devices.

Making it Happen

Refer to Figure 2. As the leads heat and the solder re-flows, quickly add more solder to flood the area. I use Kester rosin-core solder, in a 63/37 (tin/lead) alloy. The component will float, and you can use a wiping action to flick the component away from its mounting pads.

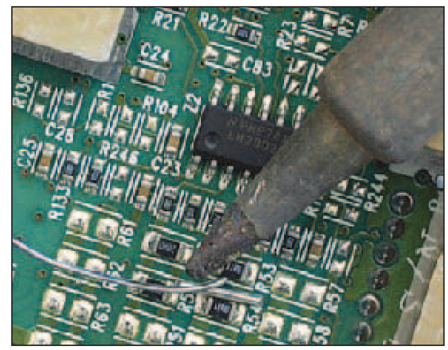


Figure 2 — As the component heats and solder begins to reflow, quickly add more solder to flood the component area. The component will float and you can flick the iron tip to push the component off of its pads.

Next, a length of solder wicking is used to remove excess solder from the board, as shown in Figures 3 and 4. The flux can also be removed with some alcohol and a little scrubbing.

The removed chip resistors ended up stuck to the 20 gauge bus wire. Although a bit discolored, these little bits can also be cleaned up with solder wick and alcohol, and be reused in another project. Of course, the little chip resistors are cheap enough to go into the trash bin (the hazardous waste bin). If the component were more valuable, such as a PIN diode or a transistor, however, it would be worth saving.

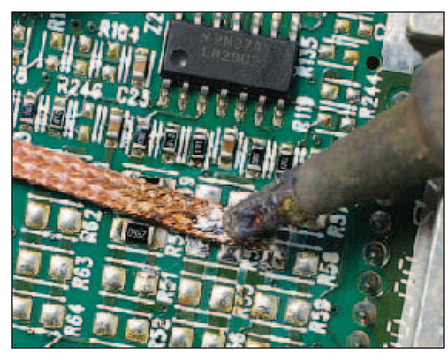


Figure 3 — Remove the excess solder and flux from the board with solder wick. Additional scrubbing with a swab and alcohol will remove the flux residue.

SMT is Here to Stay

This is another chapter in the *don't be afraid to use surface-mount components* story. As electronic products get more complex and component densities increase, SMT is the only way to go. Experimenters must practice using (and repairing) this construction method, and become comfortable with using these types of components.

By the way, the 24 GHz project I am referring to is discussed on a Yahoo! interest group site started by Frank Kelly, WB6CWN, of the San Bernardino Microwave Society (SBMS).[1]

[1] **groups.yahoo.com/group/24GHzHam/**.

ARRL Life Member and Amateur Extra class licensee Wayne Yoshida, KH6WZ, is employed by M/A-COM Technology Solutions in Torrance, California, and has been a ham since 1976. His most memorable ham radio experience was working in the press room at the NASA Johnson Space Center (Mission Control, Houston) during the 1983 Owen Garriott, W5LFL, operation aboard STS-9/SpaceLab-1. You can reach Wayne at 16428 Camino Canada Ln, Huntington Beach, CA 92649 or at **kh6kine@earthlink.net**.

Figure 4 — The board should look like this. Two 0603 size resistors have been removed, and the cleaned board is ready to receive two new resistors.

By Sam Ulbing, N4UAU

Surface Mount Technology—You *Can* Work with It!

Part 1—Start building your projects with surface-mount devices! I'll show you how!

As I look through the various electronic manufacturing companies' product datasheets, three things strike me. First, the large number of available ICs that perform functions formerly requiring several ICs. Second, the continuing shift to lower-power requirements, smaller size and usability at higher operating frequencies. Finally, the increasing number of new products are available *only* in surface-mount packages. It all fits together: Products today are smaller and more energy efficient. Look at modern H-Ts, cell phones, GPS equipment, laptop computers, microwave ovens, intelligent electronic ovens, TV remote controls and pocket calculators: One thing they have in common is their use of surface-mount (SM) ICs.

On the other hand, when I look at Amateur Radio projects, I see continued use of many discrete components and bulky DIP ICs that perform limited functions. Recently, I saw a voltage-controller project based on the use of transistors and relays! Frankly, it bothers me that there seems to be a growing divergence between the technology used by industry and that used by hams. The *Maxim Engineering Journal Vol. 29*, for instance, showcases such new ICs as an image-reject RF transceiver, a low-phase-noise RF oscillator that replaces VCO modules, a 3 V, 1 W, 900 MHz RF power transistor, a direct-conversion down-converter IC that replaces an IF mixer, an IF LO and SAW filter, and a low-voltage IF transceiver that includes the FM limiter and RSSI. All these multifunction ICs are available *only* in SM packages! I think hams are being left behind because they feel that SMT (surface-mount technology) is something they can't handle.

Since I built my first SM project two years ago, I have assembled a dozen others. I find that my skill levels have increased tremendously with practice, and I now routinely tackle projects I never thought possible just a year ago. Based on my experience, I know that amateurs *can* work with SMT. Perhaps when we show this ability, there will be more truly state-of-the-art projects in the amateur publications. How about a very small 2 meter rig, or a 900 MHz personal communicator? The ICs already exist and we need to adapt them to ham use. First however, it is necessary to develop a few basic building skills. This article series will help you develop those skills by showing what I have learned and presenting several useful and easy-to-build projects. Once you have built these, you will be able to handle most of the SM ICs I have seen used in the industry.

N4UAU

Nothing New

The concept of surface mounting parts is not new to Amateur Radio. In a September 1979 *QST* article,[1] Doug DeMaw, W1FB (SK), discusses a quick and easy circuit-board design that was basically SMT; Doug also proposed a universal PC-board layout for this kind of construction. You may think that there will *always* be DIP versions of all the SM ICs so engineers can experiment, but even today, many manufacturers are making evaluation boards available to designers so they can test the part using SM devices! I suspect it's cheaper for them to sell evaluation boards than to set up a production line to make a very limited number of DIP ICs when their real volume is in SM devices.

Some of the advantages of building with SM devices include:

- Smaller projects: I built a time-out switch that fits on a PC board one-sixth the size of a postage stamp! I was able to put the circuit into the battery compartment of a voltmeter I had so it could automatically power itself down.[2]
- Many SM versions of devices outperform the original DIP versions. Lower operating voltages and quiescent currents in the microampere range offer more efficient operation.
- Most RF projects require the use of short signal leads. SM capacitors are often recommended for use as bypass capacitors because they can be placed close to an IC and exhibit very low lead inductance. Nearly all VHF projects benefit from the use of SM devices.
- Once you've had some experience in working with SM devices, you'll feel more con-

[1]Notes appear on page 38.

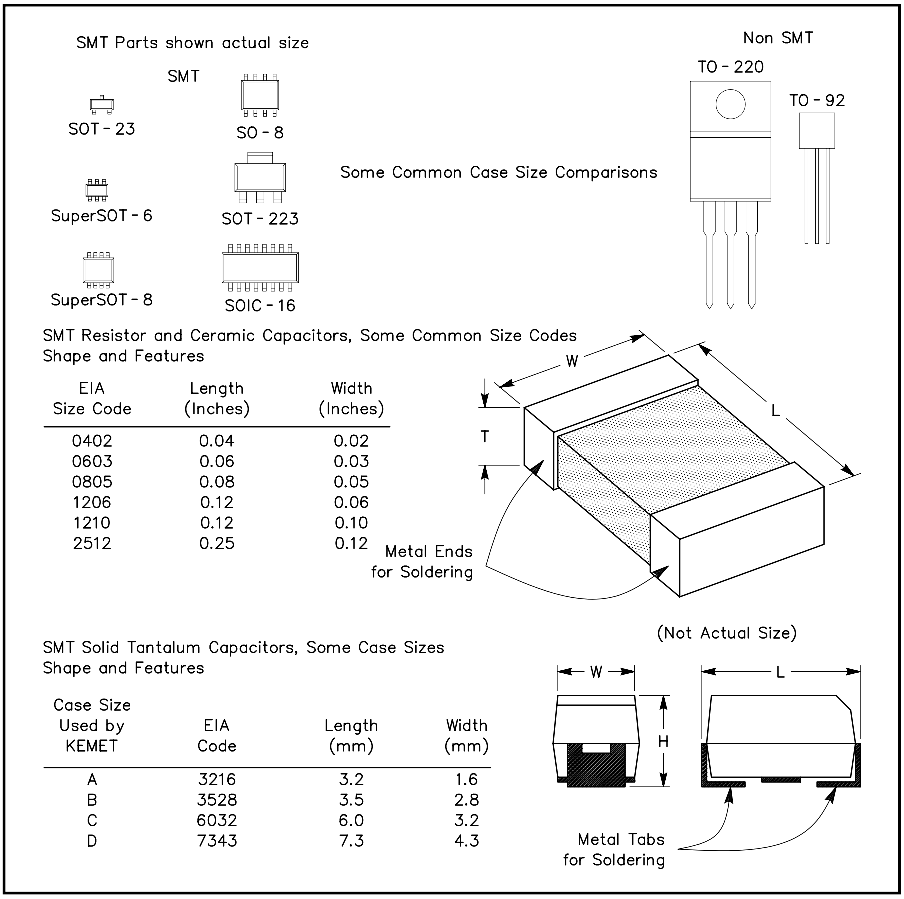
fident about repairing your own gear.

- Making a PC board for SM devices is easier than for through-hole parts because no component-mounting holes need to be drilled.
- Many new SM ICs have entire modules built into them making it much easier to build a complex circuit than with older ICs.[3]

Equipment Needed

Many people think you need lots of expensive equipment to work with SM devices.[4] Not so! You don't need an eagle's eyesight, either! My optometrist describes my eyesight as "moderately near-sighted, needing bifocals (2½ diopters)." My wife thinks I am as blind as a bat.

- A fundamental piece of equipment for SM work is an illuminated magnifying glass. I use an inexpensive one with a 5-inch-diameter lens (see the accompanying trio of tools photographs). I use the magnifier for *all* my soldering work, not just for SM use. Such magnifiers are widely available (see the sidebar "Manufacturers and Distributors of SMT Equipment and Parts") and range in price from about $25 to several hundred dollars. Most offer a 3× magnification and have a built-in circular light.
- A low-power soldering iron is necessary; one that is temperature-controlled (such as the Weller WCC100) practically eliminates the possibility of overheating a part. Use a soldering iron with a grounded tip as most SM parts are CMOS devices and are subject to possible ESD (static) failure. I have found the Weller 1/16-inch (EJA) screwdriver tip works well. I used to use an ETJ with its finer conical tip, but it does not seem to transfer the heat as well as the screwdriver tip.
- Use of thin (0.020-inch diameter) rosin-core solder is preferred because the parts are so small that regular 0.031-inch diameter solder will flood a solder pad and cause bridging.
- A wet sponge for cleaning the soldering-iron tip.
- A flux pen comes in handy for applying just a little flux at a needed spot. I find that RadioShack's flux is too sticky and it leaves a messy residue. The Circuit Works CW8200 flux pen with a type R flux is much cleaner.
- Good desoldering braid is necessary to remove excess solder if you get too much on a pad. Chem-Wik Lite 0.100-inch wide works well.
- ESD protective devices such as wrist straps may be necessary if you live in a dry area and static is a problem. I live in humid Florida, have never used these and have not had a problem.
- Tweezers help pick up parts and position them. I find that a pair of nonmagnetic, stainless-steel drafting dividers work well as tweezers. They have two very sharp needle-like points that allow me to pick up the smallest parts; and the parts seem less likely to slip from grasp perhaps because I use less force to hold them. The sharp points are useful tools for marking the PC-board copper foil before cutting out traces (more on this later). The nonmagnetic property of stainless steel means the chip doesn't get attracted to the dividers.
- If you want to make your own SM PC boards, I recommend using a Dremel Mototool (or something similar) and some ultra-fine cutting wheels.



EIA Size Code	Length (Inches)	Width (Inches)
0402	0.04	0.02
0603	0.06	0.03
0805	0.08	0.05
1206	0.12	0.06
1210	0.12	0.10
2512	0.25	0.12

Case Size Used by KEMET	EIA Code	Length (mm)	Width (mm)
A	3216	3.2	1.6
B	3528	3.5	2.8
C	6032	6.0	3.2
D	7343	7.3	4.3

Figure 1—Size comparisons of some surface-mount devices and their dimensions.

Parts

Figure 1 shows some common SM parts. Resistors and ceramic capacitors come in many different sizes, and it is important to know the part size for two reasons: Working with SM devices by hand is easier if you use the larger parts; and it is important that the PC-board pad size is larger than the part. Tantalum capacitors are one of the larger SM parts. Their case code, which is usually a letter, often varies from manufacturer to manufacturer because of different thicknesses. As you can see from Figure 1, the EIA code for ceramic capacitors and resistors is a measurement of the length and width in inches, but for tantalum parts, those measurements are in millimeters times 10! Keep in mind that tantalum capacitors are *polarized*; the case usually has a mark or stripe to indicate the positive end. Nearly any part that is used in through-hole technology is available in a SM package.[5, 6]

SMT Soldering Basics

Use a little solder to pre-tin the PC board. The trick is to add just enough solder so that when you reheat it, it flows to the IC, but not so much that you wind up with a solder bridge. Putting a little flux on the board and the IC legs makes for better solder flow, providing a smooth layer. You can tell if you have the proper soldering-iron tip temperature if the solder melts within 1.5 and 3.5 seconds.[7] I use my dividers (or my fingers) to push and prod the chip into position. Because the IC is so small and light, it tends to stick to the soldering iron and pull away from the PC board. To prevent this, use the dividers to

LODER BROOKS, KD4AKW

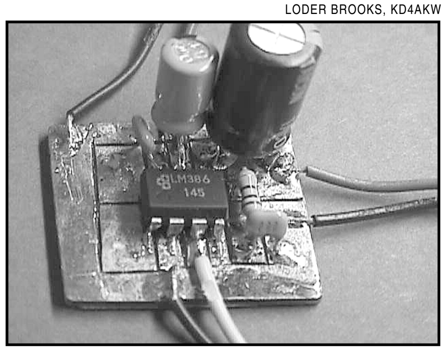


Figure 2—An LM386 audio amplifier built on a homemade PC board. The board's isolated pads are made by using a hobby tool to grind separating lines through the copper foil.

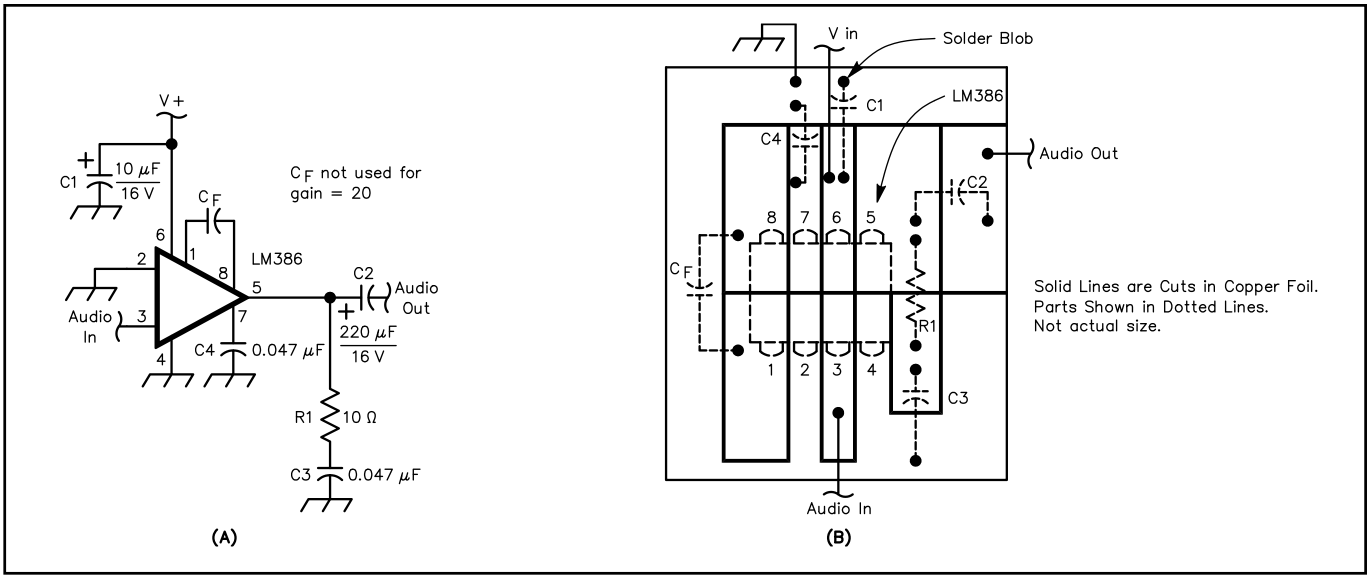


Figure 3—At A, the schematic of the LM386 audio amplifier. The component layout and PC board are shown at B. The solid, heavy lines indicate cuts made in the copper foil. This drawing is not to scale. The board is 1 inch long by 3/4 inch wide. No SM parts are used in this project, but my board-making method is shown. It allows one to get a feel for the process before tackling the smaller SM chips.

C1—10 µF, 16 V
C2—220 µF, 16 V
C3, C4—0.047 µF, 50 V ceramic
Cf—For overall circuit gains greater than 20, use 10 µF, 16 V
U1—LM386N (8-pin DIP)

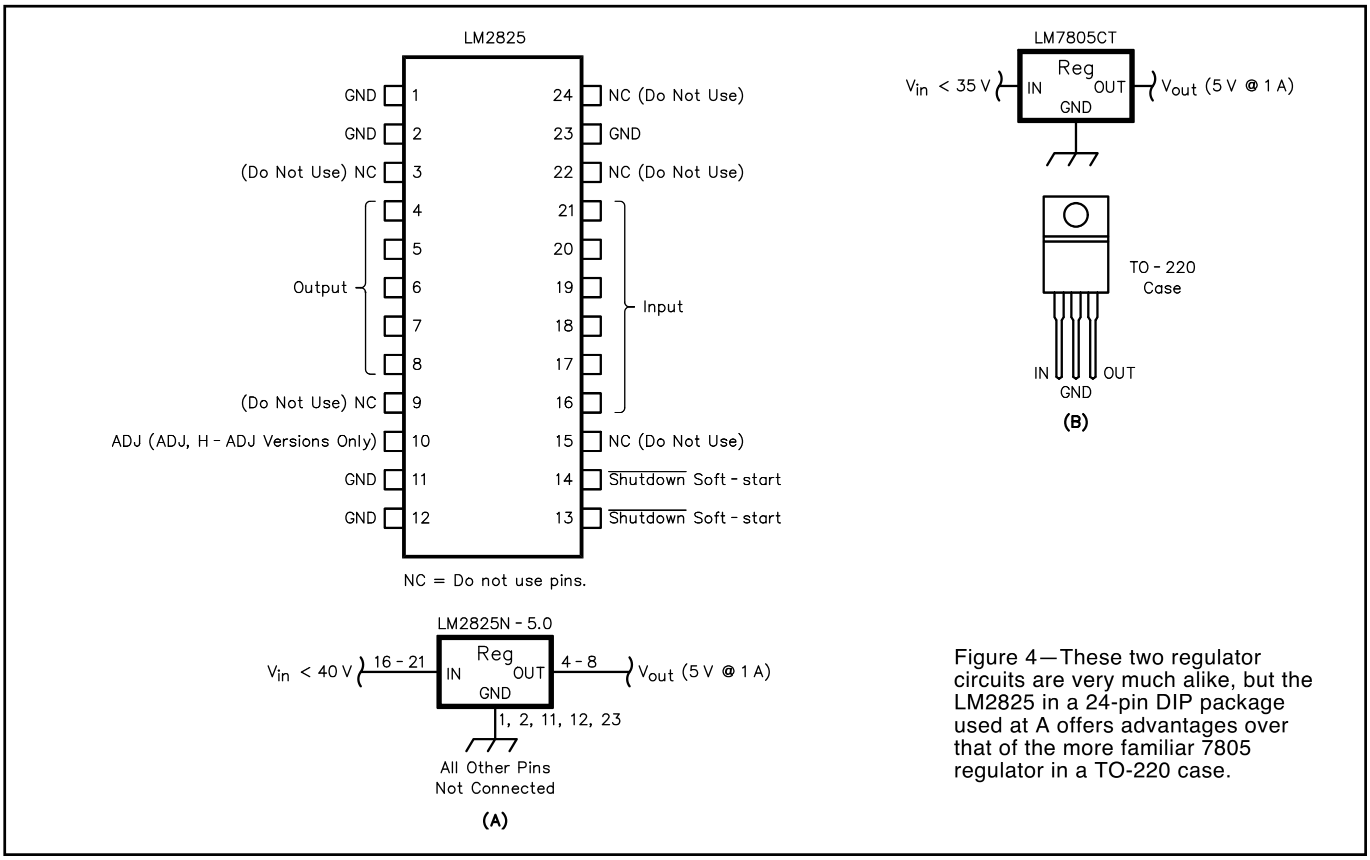


Figure 4—These two regulator circuits are very much alike, but the LM2825 in a 24-pin DIP package used at A offers advantages over that of the more familiar 7805 regulator in a TO-220 case.

hold the chip down while tacking two IC legs at diagonally opposite corners. After each tack, check that the part is still aligned. With a dry and *clean* soldering iron, heat the *PC board* near the leg.[8] If you do it right, you will see the solder flow to the IC.

The legs of the IC must lie flat on the board. The legs bend easily, so don't press down too hard. Check each connection with a continuity checker placing one tip on the board the other on the IC leg. Check all adjacent pins to ensure there's no bridging. It is easier to correct errors early on, so I recommend performing this check often. If you find that you did not have enough solder on the board for it to flow to the part, add a little solder. I find it best to put a drop on the trace near the part, then heat the trace and slide the iron and melted solder toward the part. This reduces the chance of creating a bridge. Soldering resistors and capacitors is similar to soldering an IC's leads, except the resistors and capacitors don't have exposed leads. My reflow method works well for these parts, too.

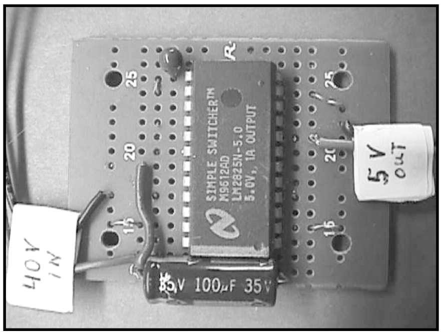


Figure 5—LM2825 circuit is constructed on readily available RadioShack perfboard (RS 276-150 or similar). Photo shows optional input filter capacitor and optional soft-start capacitor. Refer to the datasheet at the National Semiconductor Web page for suggested values for these optional components.

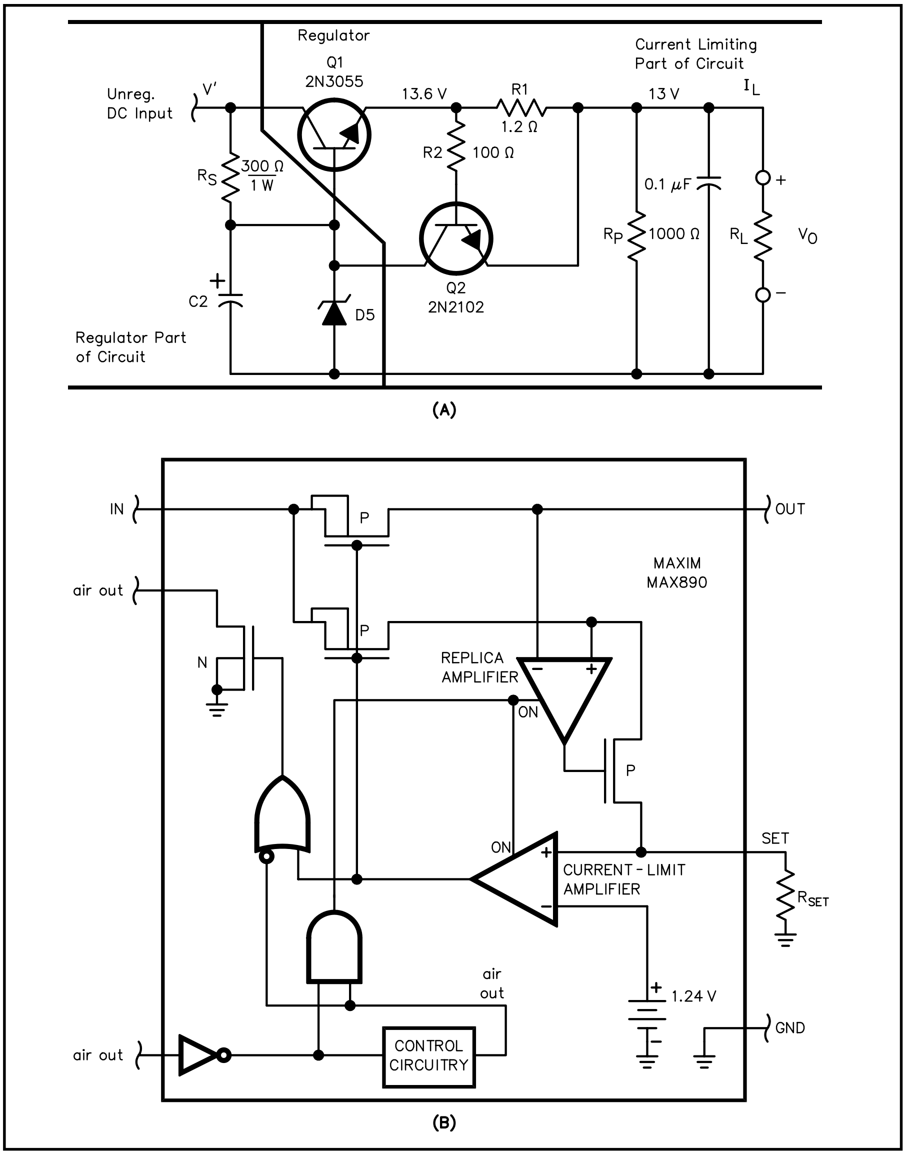


Figure 6—At A, a current-limiting circuit using older technology. Within the confines of the tiny MAX890, (B) newer technology offers a 1-A, P-channel MOSFET switch, a comparator, a voltage reference, a current-measuring circuit, control and fault-indicator circuits! For Project 1 they are:

Attaching wires that connect to points off the board can be a bit of a challenge because even #24 stranded wire is large in comparison to the SM parts. First, make sure all the wire strands are close together, then pre-tin the wire. Carefully place the wire on the pretinned pad and heat it with the soldering iron until the solder melts.[9]

Making a SM PC Board

It is possible to etch SM PC boards just like a conventional board, but I recall Doug (W1FB) DeMaw's comment on etching: "If you don't mind a few brown stains here and there on your garments, etching is one way to make the board." Evidently he, like I, *did* mind, and he proposed a strong-arm method of using a hacksaw to cut square pads in the board foil. Hacksaws are too large and wide for SM use; I use a Dremel Mototool and a thin cutoff wheel. With these, I can cut a line as narrow as 0.005 inch, which lets me build with most of the available SM ICs.

To make such a "PC" board, start by sketching a layout for it. Don't worry about drawing it to scale, but make the sketch large enough to see what is happening. Normally, we think in terms of *connections between parts* because schematics show lines from point to point, representing the interconnecting wires. I find it is more useful to think in terms of the *spaces between the lines* because I am removing copper material to separate traces, not adding material to make traces. Where wires attach to the board, leave a large surface because the wires are relatively large. When making cuts, it is easiest to do it using a large piece of material that you can hold securely. Cut the board to size after you have cut all the traces.

Once I have the layout drawn, I hold the IC to the copper and used a fine-pointed tool (a 0.5 mm pencil or my dividers) to mark the location of the cuts on the PC board. I then remove the IC and use my Dremel tool to cut the copper along the marks. For critical cuts between an IC's closely spaced leads, I make one cut, then reposition the IC on the board and verify that the remaining marks are still correctly aligned. I do all this freehand. Using an ultra-thin cutoff wheel, I find it is quite easy to cut in a straight line. At first, I used a fine cutting bit (#108), but that tip made a wider cut and it was difficult to cut a straight line. Dedeco (see the sidebar "Manufacturers and Distributors of SMT Equipment and Parts"), supplier of tools for jewelers and dentists, makes two ultra-thin wheels—0.009 and 0.005 inch. For the very small ICs—those in SOT23-5 and SuperSOT-8 packages—I use a 0.005-inch wheel, otherwise the 0.009-inch wheel is ideal. Be careful when handling these wheels as they break easily. Also, don't cut too deeply into the board material. At the intersection of the cuts, take care not to cut too far. Sometimes I cut close to an intersection, then use a razor blade to finish the job. A quick sanding deburrs the cuts. Run through the cuts with a small screwdriver or pen knife to ensure they are cleanly cut and without burrs. Finally, use your ohmmeter to verify that the islands/pads aren't connected.[10]

I recommend you make your own PC boards: They're easier to produce than through-hole boards, and you'll then be able to experiment with *your own* projects rather than waiting for others' projects to come along. You can use this method with SM or non-SM projects.

The Projects

All the projects I'll present are easy-to-build beginner projects, yet each offers significant advantages over similar projects based on the old (DIP) technology. As you build each project, you'll develop SM skills and wind up with some useful gadgets. I have tried to arrange the projects by degree of skill required. For those who want to make their own PC boards, I describe my layout. Ready-made PC boards are available for all of these projects except the first, Project 0A.[11]

Project 0A—Getting a Feel for SM Techniques

This audio amplifier is a good starter project for those who want to learn to work with SM devices because the technique is the same, but the parts are physically larger because no SM parts are used. I made the layout, cut the board and assembled this project in a little over an hour. Try doing that with etching and through-hole construction! I think you'll agree that the finished product looks as good as if it were assembled on a commercially made PC board (see Figure 2).

This project is shown in *The 1996 ARRL Handbook* (and subsequent editions) on page 25.8 using "dead-bug" construction.[12] All the parts are mounted on a groundplane with no component-mounting holes. It is easy to duplicate this project using SM techniques. Figures 3A and B show the schematic and board layout, respectively. I bent and trimmed the pins on the LM386 so that they look like a large SO-8 package. You can make the cuts with a 0.015-inch wheel. When cutting the ends of the traces to pins 1 and 3, be careful that you don't cut too far and run through the cut from pin 2.

An LM386 is *not* state of the art. If you want to see the difference between it and a state-of-the-art amplifier, build the SMALL.[13] It uses an LM4861, which is available only in an SO-8 package. In addition to its smaller size, the SMALL has more power output, far better fidelity and the ability to work with low-voltage power sources.

Project 0B—The World's Easiest Surface-Mount Project

You may be curious about comparing SMT with conventional technology, but not want to solder those small ICs. If so, this project is for you. It is based on the LM2825, a large DIP 5 V regulator used in the circuit shown in Figure 4A. Next, build a conventional 5 V regulator using an LM7805 in a TO-220 case, Figure 4B. Both can be built on a RadioShack universal PC board; the LM2825 project is shown in the photograph (Figure 5). Although the circuits look nearly identical, if you use a 12-V source to power both of them and put a load of 0.5A or more on each, you'll see that the LM7805 gets *very hot*, while the LM2825 stays cool. That's because the LM2825 is a sophisticated *switching regulator* with all of the tiny SM parts packed in a DIP case.

Out with the Old...

The (*1996 ARRL Handbook*) current-limiting circuit of Figure 6A uses a resistor (R1) and series pass transistor (Q1) in series with the load. R1 detects the current flow and Q1 limits it when necessary. This design has a voltage drop from input to output of 600 to 1200 millivolts depending on the load (before any overload). Its voltage regulation is poor and its efficiency is low.

...In with the New

By contrast, Maxim's MAX890 (Figure 6B) operates with voltage levels from 2.7 to 5.5 V (6 V maximum) with a current drain of only 15 μA. On this tiny chip are a 1-A, P-channel MOSFET switch, a comparator, a voltage reference, a current-measuring circuit and control and fault-indicator circuits! The maximum voltage drop across the switch is only 90 mV unless an overcurrent condition exists. Instead of using a series resistor to monitor current, the MAX890 uses a current replica circuit that controls the MOSFET limiting switch. For a short circuit—or for a large initial surge current—the circuit shuts off the switch in just five microseconds, then slowly turns it on while limiting the current to 1.5 times the maximum current. For prolonged overcurrent situations, there is a large amount of power dissipated in the MOSFET. To combat this, the chip has a thermal shutdown circuit that cycles the switch on and off, if necessary, to keep the temperature within a safe range.

Project 1—The SmartSwitch

This project is based on Maxim's MAX890, available in a common and fairly large SO-8 SM package that is relatively easy to work with. The switch is smart because it limits the current it passes to an amount you *preset*. This device not only protects your expensive electronic projects against a short circuit, but extends their life by limiting inrush current, a major cause of component

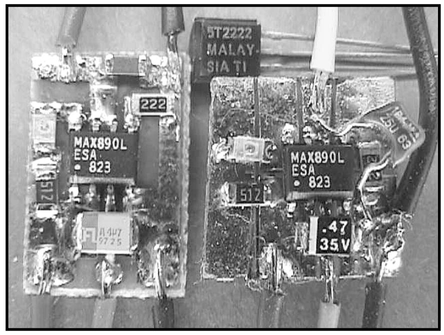


Figure 7—Here are two versions of the SmartSwitch compared in size to a TO-92 package transistor. The board on the right is homemade; the one to the left is available from N4UAU; see Note 11.

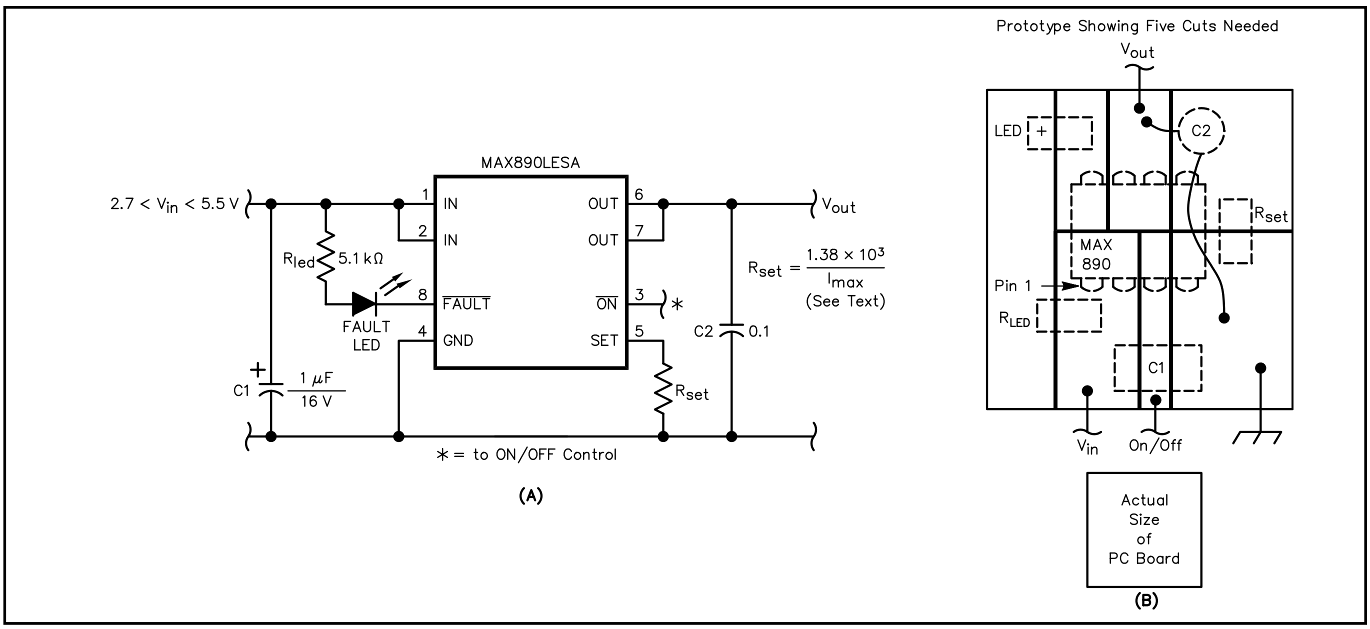


Figure 8— The SmartSwitch circuit (A) and board layout (B).

U1—MAX890LESA
Rset—See Eq 1 in text.
Rled—5.1 kΩ
C1—1 μF tantalum capacitor, 10 V or greater
C2—0.1 μF ceramic

failure. The IC has an output that can be set to trigger a fault indicator, such as an LED or bell. The **ON/OFF** pin exhibits a high impedance and can be controlled by a computer or low-output sensor such as a photoelectric cell. Building the SmartSwitch is straightforward and relatively easy. Figure 7 shows how physically small the switch is.

Figure 8A is the SmartSwitch schematic; the board layout is shown in Figure 8B. Circuit operation is simple: Power connections are made at pins 1 and 2, the high side of the switch and switched power are available at pins 6 and 7, respectively. **Rset** sets the trip current:

$$Ilimit = 1.38 \times 10^3 / \mathbf{Rset} \qquad \text{(Eq 1)}$$

where *Ilimit* is the trip current in amperes, and **Rset** is the controlling resistance value in ohms.

I used a 2.2 kΩ resistor at **Rset** to establish a current limit of 625 mA. (Current-trip levels can be set to values between 200 mA and 1 A.) C1, the input capacitor, prevents input-voltage drop with current surges; in many cases, C1 can probably be eliminated. Output capacitor C2 protects the circuit against inductive spikes. When a current or thermal overload trips the switch, **FAULT** pin 8 goes low. I put a SM LED on the board to indicate when a fault occurs. Pin 8 is not intended to sink a lot of current, so I used a 5.1 kΩ resistor to limit the LED current to about 1 mA. You could use a100 kΩ pull-up resistor instead and an external high-impedance indicator.

Construction Comments

To make this project's PC board, I used a Dremel tool and a 0.009-inch disc. For my prototype, I found it easiest to use a monolithic (non-SM) capacitor for C2, mounting it across the top of the IC. (There is no rule that prohibits you from mixing technologies, and this made construction easier.) Notice how large the capacitors are compared to the IC. As is true with most SM projects, circuit layout is important: Short leads offer low inductance to promote fast switching in the event of a current overload. In case of a short circuit, the board's ground plane helps dissipate heat.

Tune In Again...

Next month, we will look at two chips that turn a positive voltage into a negative voltage and are only available in SM cases. One of these is in the large SO-8 case (as in Project 1); the other is in a smaller SOT-23 case. I hope you build Projects 0A and 1 because the skills you develop working with them will be useful in completing the projects to come.

Manufacturers and Distributors of SMT Equipment and Parts

AAVID (manufacturer)—143 North Main St, Suite 206, Concord, NH 03301; tel 603-224-9988; fax 603-223-1738; **http://www.aavid.com**; heat sinks, information about them.

AVX (manufacturer)—**http://www.avxcorp.com/products/capacitors/smtc.htm**; low-ESR capacitors

Bourns (manufacturer)—**http://www.bourns.com/**; resistors and potentiometers.

Chemtronics (manufacturer)—8125 Cobb Ctr Dr, Kennesaw, GA 30152-4386; tel 800-645-5244, 770-424-4888; **http://www.chemtronics.com**; soldering paste, solder, solder wick.

Contact East (distributor)—tel 800-225-5370, 888-925-2960, fax 800-743-8141; **http://www.contacteast.com**; flux pens, soldering equipment, illuminated magnifying glasses.

Dedeco International (manufacturer)—Long Eddy, NY 12760; tel 800-964-6616; **http://www.dedeco.com**. Manufactures the cutoff wheels I use. My 0.005-inch wheel is #5190, the 0.009-inch wheel is #5187. I found an assortment of Dedeco wheels at Home Depot, but they did not include the 0.005-inch wheel.

Digi-Key (distributor)—701 Brooks Ave S, PO Box 677, Thief River Falls, MN 56701-0677; tel 800-344-4539, 218-681-6674, fax 218-681-3380; **http://www.digikey.com**. Carries a wide selection of National, Maxim and International Rectifier ICs, many SMT parts, lithium batteries, holders and soldering equipment. They have good links to manufacturers' Web pages. Digi-Key has a $5 handling charge on orders less than $25.

FAR Circuits (manufacturer)—18N640 Field Ct, Dundee, IL 60118; tel 847-836-9148 voice/fax; **http://www.cl.ais.net/farcir**; custom PC boards.

Gerber (distributor)—Gerber Electronics, 128 Carnegie Row, Norwood, MA 02062; tel 800-225-8290, 781-769-6000, fax 781-762-8931; **http://www.gerberelect.com**. National Semiconductor products, most of the new ICs; $25 minimum order.

Hosfelt Electronics Inc (distributor)—2700 Sunset Blvd, Steubenville, OH 43952; tel 800-524-6464, 888-264-6464, 740-264-6464, fax 800-524-5414; (no e-mail address, no Web site); tilt switches and some SMT parts, 3 V lithium batteries and battery holders.

International Rectifier (manufacturer)—233 Kansas St, El Segundo, CA 90245; tel 310-726-8000, fax 310-322-3332; **http://www.irf.com**; IRF7201, IRLML2402, IRFZ46 and other MOSFETs, diodes, etc.

Kemet (manufacturer)—PO Box 5928, Greenville, SC 29606; **http://www.kemet.com**; capacitors; lots of technical information at this site.

Keystone Electronic Corp—(manufacturer), 31-07 20th Rd, Astoria, NY 11105; tel 718-956-8900; **http://www.keyelco.com**. Manufactures a complete line of battery holders, components and hardware.

Maxim (manufacturer)—120 San Gabriel Dr, Sunnyvale, CA 94086; tel 800-998-8800, 408-737-7600; **http://www.maxim-ic.com**; MAX871, MAX890 and other ICs.

Micrel (manufacturer)—1849 Fortune Dr, San Jose, CA 95131; tel 408-944-0800; **http://www.micrel.com**; MIC1555 and other ICs.

N4UAU (distributor)—5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; supplies parts kits for most of *QST* projects.

National Semiconductor (manufacturer)—2900 Semiconductor Dr, PO Box 58090, Santa Clara, CA 95052-8090; tel 408-721-5000, 800-272-9959; **http://www.national.com/**; LM2662 and many other ICs.

Newark (distributor)—tel 800-463-9275; call this number to get the phone and fax information of the representative in your area; **http://www.newark.com**. Carries products from many manufacturers including National, Maxim, International Rectifier, Micrel, Motorola, Sprague, Bourns. Many SMT parts, batteries, holders. There is a $5 handling charge for orders less than $25.

Motorola (manufacturer)—**http://www.mot-sps.com/sps/General/chips-nav.html** MC14020 and almost every other IC in the world. Motorola has a large Web site. This is where I have found the most useful information. If the site does not have what you want, try the links to other of its sites.

Sprague (manufacturer)—PO Box 231, Sanford, ME 04073; tel 207-490-7257, fax 207-324-7223; **http://www.vishay.com/products/capacitors.html**; low-ESR capacitors.

Star Micronics (manufacturer)—**http://www.starmicronics.com**—information on buzzers.

Notes

[1]Doug DeMaw, W1FB, "Quick-and-Easy Circuit Boards for the Beginner," *QST*, Sep 1979 pp 30-32.

[2]Sam Ulbing, N4UAU, "Mega-Mini Micropower Timeout Switch," *73 Amateur Radio Today*, Jul 1998, pp 42-48.

[3]I was intrigued to come across an engineer's comment in an industry magazine: "RF circuits are readily available as easy to use building blocks, so you needn't fully understand their operation to employ them in an application." Perhaps he had Amateur Radio builders in mind!

[4]Flex-mounted illuminated magnifying lenses are available at office-supply stores and electronic-component suppliers such as Office Depot, Office Max, Digi-Key, Newark, etc. Dremel tools are available from discount stores, Home Depot and Lowe's. Thin 0.020-inch diameter solder can be found at RadioShack (#64-013). Digi-Key, Contact East and Newark sell rosin flux pens.

[5]I have found the best way to locate state-of-the-art parts is via the Internet. Virtually every manufacturer has their component datasheets, applications notes and other information posted. It's a design engineer's dream! No longer do you need lots of databooks. Distributors, too, have catalogs

on-line. If you want to know if a company stocks the Maxim 890 for instance, you need only go to the Maxim home page, check out who their distributors are, then go to those sites and see if they have the part. It's true that some distributors have large minimum quantities for orders, but others don't. If you want more information on the parts in this project, see the sidebar "Manufacturers and Distributors of SMT Equipment and Parts."

[6]You might wonder "How small can they go?" National Semiconductor has recently introduced a device (the LMC6035) in a Micro-SMD package that is one-quarter the size of an SOT-23 package! According to National, the package is only slightly larger than the die itself: "This time we may have reached the packaging limits with the smallest possible footprint." Paul McGoldrick, Senior Technology Editor for *EDTN* said he "...expects to see a lot of licenses being sought in the next months for other manufacturers seeking to take advantage of this huge jump in process 'packaging' and in the lower costs associated with it," *EDTN*, Sep 1998. This is available for viewing at **http://www.EDTN.com/analog/prod194.htm/**.

[7]Per Kemet Electronics Corp monograph F-2103A, *Repair Touch Up Hand Solder—Can These Be Controlled*, by Jim Bergenthal. This and other free literature can be obtained from Kemet Electronics at their Web site (**http://www.kemet.com**). In the upper-left-hand corner of the page, select **Literature Request** after clicking on **Tantalum Capacitors**, then fill in the information form. Finally, click on **Request Selected Literature**. Or, use the Kemet mailing address given in the sidebar "Manufacturers and Distributors of SMT Equipment and Parts."

[8]See Note 7. Kemet emphasizes that: "UNDER NO CONDITIONS SHOULD THE IRON TOUCH THE PART. This is a major cause of part damage." I have touched parts often while soldering them and they have not sustained damage. Perhaps I have been lucky!

[9]Another approach to SMT soldering was suggested to me by Fred, W3ITO. He uses solder paste and a hot plate. He believes it is the only reliable method for amateur SMT (but he was dealing with equipment that had to meet military standards). I have not tried this approach as it appears to need fairly accurate temperature control and the solder paste is difficult to locate, expensive and must be specially stored in a cool dry environment. I would be interested to hear from others who may have tried this method.

[10]Universal SM prototype boards are also available from FAR Circuits. See Paul Pagel, N1FB, "Breadboards from FAR Circuits," *QST*, Nov 1998, p 74.

[11]If you are interested in learning to make your own boards as described, I have a limited number of parts kits consisting of a 3¥6-inch double-sided, copper-clad board, eight cut-off wheels (two 0.005 inch, four 0.009 inch and two 0.025 inch) and the special mandrel recommended for use with the ultra-fine cut-off wheels. This kit allows you to make boards for all the projects in this series and many more. Price $13. (Florida residents must add sales tax. For orders outside the US, please add $3 for shipping.)

Project #0B, Gerber Electronics has agreed to sell this chip to readers of this article at a special price of $12.50 ($8 less than the normal unit price) and waive their normal $25 order minimum. Be sure to identify yourself as a *QST* reader to qualify for this price.

Project #1, A limited number of parts kits are available from me for $6, without a PC board. If you want a premade PC board add $1.50. (Florida residents must add sales tax. For orders outside the US, add $1 for shipping.)

Order from Sam Ulbing, N4UAU, 5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; **n4uau@afn.org**. Credit cards are *not* accepted.

[12]I omitted the feedback capacitor between pin 1 and pin 8 to reduce the gain but my layout allows for it to be added if desired.

[13]Sam Ulbing, N4UAU, "SMALL—The Surface Mount Amplifier that is Little and Loud," *QST*, Jun 1996, pp 41-42 and 68.

Sam Ulbing, N4UAU, studied electronics in the 1960s, but spent his work career in the financial area. Since he retired in 1986, Sam has enjoyed exploring the opportunities offered to the amateur builder by the new ICs. He feels that electronic design for amateurs has become much easier than it used to be. Sam recalls how in the '60s, he spent hours sweating over complex equations to design even simple circuits. Now, although he has forgotten almost all of his math, the circuits he has built with the new electronics do very sophisticated functions and best of all they work! Presently, Sam is playing with three projects, choosing to build all of them using his "surface-mount style" because "It's just more fun to do it that way." You can contact Sam at 5200 NW 43rd St, Suite 702-177, Gainesville, FL 32606; **n4uau@afn.org**.

By Sam Ulbing, N4UAU

Surface Mount Technology—You *Can* Work with It!

Part 2—Last month, we built a couple of simple projects with surface-mount devices. This month's inverter projects go a bit farther.

Projects 2A and 2B—Two 5 V Inverters

A low-current, negative 5 V supply is often a handy item to have on the workbench. Many amplifier circuits are simpler to design using positive and negative voltage sources. Perhaps you have an alphanumeric LCD and found it needs a negative voltage on the **CONTRAST** pin to work. A simple way to supply this negative voltage is to use an ICL7660 voltage inverter, which has been around for a long time. (I'll present another voltage-inverter application in Project 4.) Advances in technology have improved on the '7660. Two ICs I know of that offer significant improvements over their precedents, but both are available only in SM cases: The LM2662 by National is in an SO-8 package and Maxim's MAX871 is available only in SOT-23. Certainly it is possible for manufacturers to make these improved IC versions in a DIP, but neither National nor Maxim have chosen to do so. This appears to me as another signal that the industry is moving toward SM-only parts.

The Technology

Figure 9 shows how these voltage-inverter ICs operate internally. Each consists of four CMOS switches (S1 through S4) sequentially operated by an internal oscillator. During the first time interval, S1 and S3 are closed and S2 and S4 are open; the +5 V input charges C1 with its + terminal being positive and the opposite terminal at ground. At time interval two, S1 and S3 are open and S2 and S4 are closed. There is still 5 V across C1 with the pin 2 side being positive, but pin 4 is no longer at ground potential. The 5 V charge across C1 is transferred to C2—and since C2's positive side is connected to ground—the other side must be 5 V lower than ground, or –5 V. The reason the SM switches can handle more current and still be physically smaller is related to their lower resistance and, since both operate at higher frequencies, smaller-value capacitors can be used for a given current output. For best efficiency, low ESR (equivalent series resistance) capacitors should be used. An input bypass capacitor (the value of which depends on the IC and application) improves performance if the power source has a high impedance.

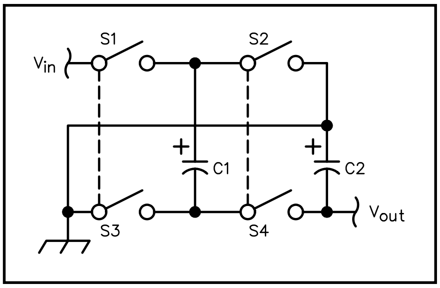


Figure 9—Diagram of the internal workings of the three voltage inverters. See the text for an operational explanation.

LODER BROOKS, KD4AKW

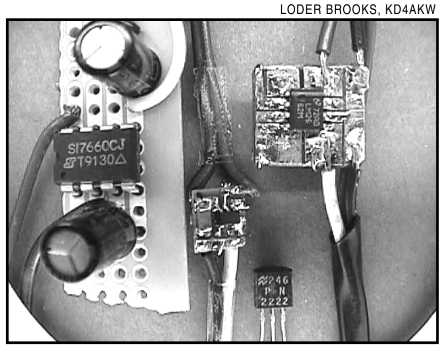


Figure 10—Here, the three voltage-inverter projects are compared in size to a PN2222 transistor.

With the trend toward smaller ICs and fewer IC pins, there are often families of nearly identical but specialized ICs. The LM2662 is one of two nearly identical inverters described in the same data sheet. The other, the LM2663, uses pin 1 as a shut-down control instead of a frequency control. This is a common feature with the new technology because of the ever-increasing use of battery power sources, and is especially useful when the inverter is computer controlled. During shut-down, the IC's current drain is reduced to only 10 µA. The MAX871, like the LM2662, has a brother described in the same data sheet. The MAX870 is identical to the MAX871n except that it runs at 125 kHz, and although it needs larger capacitors, it draws only 0.7 mA.

Because large-value capacitors increase a circuit's physical size, it's good to know the minimum capacitance value you can use. This depends on the frequency of operation and the ESR of the capacitor. Nonpolarized capacitor types commonly recommended are Sprague series 593D or 595D, AVX series TPS and the ceramic X7R series. Unfortunately, a capacitor's ESR is often not given in a parts catalog and you may have to consult a data sheet. If you want to try other capacitor values, use the following formulas to calculate output resistance and ripple. Note that C1's resistance is four times as important for reducing resistance as C2, but C1 has no effect on ripple.

$$R_{out} = 2R_{sw} + 1/f \times C1 + 4ESR1 + ESR2 \quad \text{(Eq 2)}$$

$$V_{ripple} = I_{load}/f \times C2 + 2I_{load} \times ESR2 \quad \text{(Eq 3)}$$

where

R_{out} = output resistance of the circuit
R_{sw} = sum of the *on* resistances of the internal switches
f = frequency of the oscillator driving the inverter
ESR1 = equivalent series resistance of C1
ESR2 = equivalent series resistance of C2

[16]Notes appear on page 50.

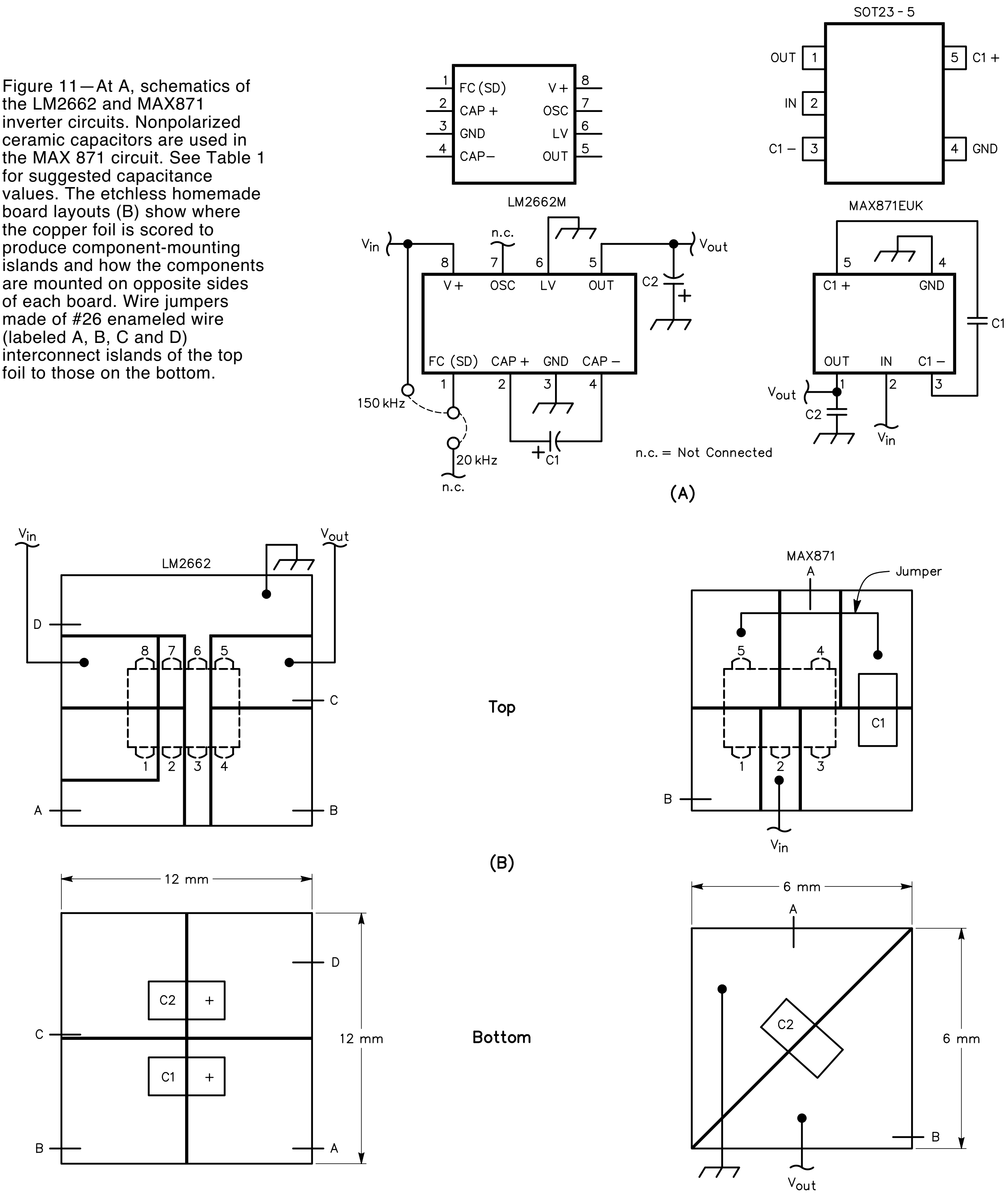


Figure 11—At A, schematics of the LM2662 and MAX871 inverter circuits. Nonpolarized ceramic capacitors are used in the MAX 871 circuit. See Table 1 for suggested capacitance values. The etchless homemade board layouts (B) show where the copper foil is scored to produce component-mounting islands and how the components are mounted on opposite sides of each board. Wire jumpers made of #26 enameled wire (labeled A, B, C and D) interconnect islands of the top foil to those on the bottom.

V_{ripple} = peak-to-peak ripple voltage at the output

I_{load} = load current delivered by the inverter

All three ICs can be used in other modes, such as voltage doublers, connected in cascade to increase output voltage, or connected in parallel to increase output current. For information on circuits to use and more information about design consi-derations, refer to the device data sheets.

The SOT-23 is a popular IC size and it is important to develop the skills to work with it if you want to make full use of the new technology. When you build the MAX871 project, set it aside because you may find it useful in Project 4.

Table 1 summarizes some features of the ICs mentioned, and Figure 10 shows you what the three completed circuits look like. You can see that the LM2662 circuit is somewhat smaller than the '7660, yet it provides *10 times* the current output! The MAX871 circuit is extremely small and outperforms the ICL7660.

Figures 11A and 11B show the schematics and board layouts, respectively. The circuits are simple, each requiring but two capacitors and one IC.[17]

Table 1

	ICL7660	LM2662	MAX871
Package	DIP, SO-8, Can	SO-8	SOT-23
Circuit Resistance (Ohms)	55	3.5	25
Osc Frequency (kHz)	10	20	500
Recommended Cap (μF)	10	100	0.2
V_{out}* @ I = 0	10.0	10.0	10.0
I = 14 mA	9.41	9.97	9.71
R = 100W	6.30	9.66	8.04
I_{supply} (mA)	0.17	0.30	2.7

*These figures are based on actual circuit measurements with the load connected between the positive and negative outputs.

Building the LM2662 Circuit

To save space, I put the IC on one side of a double-sided board, mounting the capacitors on the opposite side. Interconnections between the two board sides are made by short pieces of #26 enameled wire. The wires (labeled A, B, C, D in Figure 11B) bend around the edge of the board. If you have built Project 1, you will have no problem with this one. Be careful to observe capacitor polarity. Even though the LM2662 is smaller than the ICL7660, it offers more features. Pin 1 (which is not used in the ICL7660) controls the LM2662's internal oscillator. The inverter runs at 20 kHz when this pin is left unconnected, and at 150 kHz when connected to V_{CC}. If you want the circuit to operate at 150 kHz, add a jumper between pins 1 and 8 of the IC. This allows you to use smaller capacitors, but at the price of a higher supply current.

Building the MAX871 Circuit

The first time you see this project, you may think "It's too small to build by hand!" But I've built four different circuits this size and made a PC board for each one—so can you! Because the SOT-23 package is smaller than the SO-8, I used a 0.005-inch wheel to make the island-separating cuts on my PC board. Although the IC's pins are small and closely spaced, the SOT-23-5 board requires only two critical cuts: those between pins 1 and 2 and between pins 2 and 3. The spacing between pins 4 and 5 is as large as that of an SO-8 package. Mounting C2 beneath the board makes component layout much easier.

SOT-23 packaged devices are too small for manufacturers to imprint the part number on them—MAX890EUK just will not fit! Instead of MAX890EUK, Maxim uses the marking **ABZO**. If you get two SOTs mixed up, you will have to consult the data sheets to determine which is which.

Next Month

In Part 3, we'll look at a low-voltage battery protection switch that makes use of a few SM ICs: three SO-8s and one SOT-23.

Notes

[16]Part 1 of this four-part series appears in the April 1999 issue of *QST*, pp 33-39.

[17]Obtaining the parts—Project #2A: Gerber Electronics stocks the LM2662 and Newark Electronics stocks low-ESR tantalum SM capacitors. If you cannot find an LM2662, use the LM2660, Maxim MAX660 or the Linear Technology LTC660; all have similar characteristics and identical pin outs. Digi-Key carries some of these ICs, but does not stock the low ESR SM capacitors. Low-ESR SM capacitors are quite expensive, so you may want to use standard tantalum capacitors instead. These are available from most suppliers. I have a PC board for the layout described; price: $1.50. Contact Sam Ulbing, N4UAU, 5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; **n4uau@afn.org**. Credit cards are *not* accepted.

Project #2B: A limited number of parts kits, with hard-to-find 1 μF ceramic capacitors (to permit maximum current output with minimum ripple) are available from me for $6 *without* a PC board. If you want a pre-made PC board add, $1.25. (Florida residents add sales tax.)

If you are interested in making your own boards as described, I have a limited number of parts kits consisting of a 3×6-inch double-sided, copper-clad board, eight cutoff wheels (two 0.005 inch, four 0.009 inch and two 0.025 inch diameter) and the special mandrel recommended for use with the ultra-fine cutoff wheels. Price: $13. This kit allows you to make the boards for all the projects in this series and more. (Florida residents must add sales tax. For orders outside the US, please add $3 for shipping.)

You can contact Sam Ulbing, N4UAU, at 5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; **n4uau@afn.org**.

By Sam Ulbing, N4UAU

Surface Mount Technology—You *Can* Work with It!

Part 3—This more-complex SM project employs a total of four ICs and seven other parts on a PC board three-quarters of an inch square! Despite its small size, it can control current levels of up to 10 A—without using mechanical relays!

When Hurricane Georges came through in 1998, my friend, Dave, NØLSK, had left his boat at a marina in the Keys. Although he had the boat tied up well, he forgot that his refrigerator shifts to battery power if the ac-line power is lost. When he returned to the boat after the storm, his boat was okay, but its battery was dead. With the switch about to be described, Dave wouldn't have lost his expensive battery.

Project 3—A Low-Voltage Battery-Protection Switch

This switch, based on a MAX835 (available only in an SOT-23 case), is a *latching* voltage monitor—ideal for controlling a switch. A recent *QST* project used a MAX8211 (a DIP IC) in an undervoltage circuit,[18,19] but because it doesn't latch, that chip wouldn't work well for controlling a switch. Here's why: Every time the voltage dropped low enough to trip the monitor, it would disconnect the load, and the voltage would rise and turn the monitor back on. Such cycling could injure the equipment.

Figure 12 is a schematic of the switch. When V_{CC} drops below 12 V, and the voltage at U1 pin 4 goes below 1.2 V. That causes the output voltage on pin 5 to drop from about 5 V to 0 V. Pin 2 of U2, an MIC5014, accepts a logic-level signal and uses it to control an on-board charge pump. Q1 and Q2 are N-channel MOSFETs used as a 10 A high-side switch. (See the sidebar "Selecting a MOSFET for power control.") To turn the switch on, the gate must be at least 10 V above the source voltage (5 V for logic-level MOSFETs). That means you need 22 V to turn on the switch. This voltage is supplied by the charge pump in U2. U2 also acts as a buffer for U1, which cannot operate at 12 V. R1 through R4 provide a nominal 5 V power source for U1 and provide the voltage-level input signal to pin 4 of U1. Pull-down resistor R5 prevents unplanned resetting. Although the data sheet doesn't show that R5 is required, the very high impedance of this pin (the current drain is 1 nA) makes a pull-down resistor a wise investment, especially in an RF environment.

Depending on your circuit needs, you will use a variety of values for R1 to R4. Here's how I selected my values: U1 can operate with voltages of 2.5 to 11 V, so the R1-R4 divider must keep the voltage at pin 3 in this range as V_{CC} changes. The maximum V_{CC} I ever expected to encounter was 15 V, and the least, 11 V. U1 draws 2 µA,

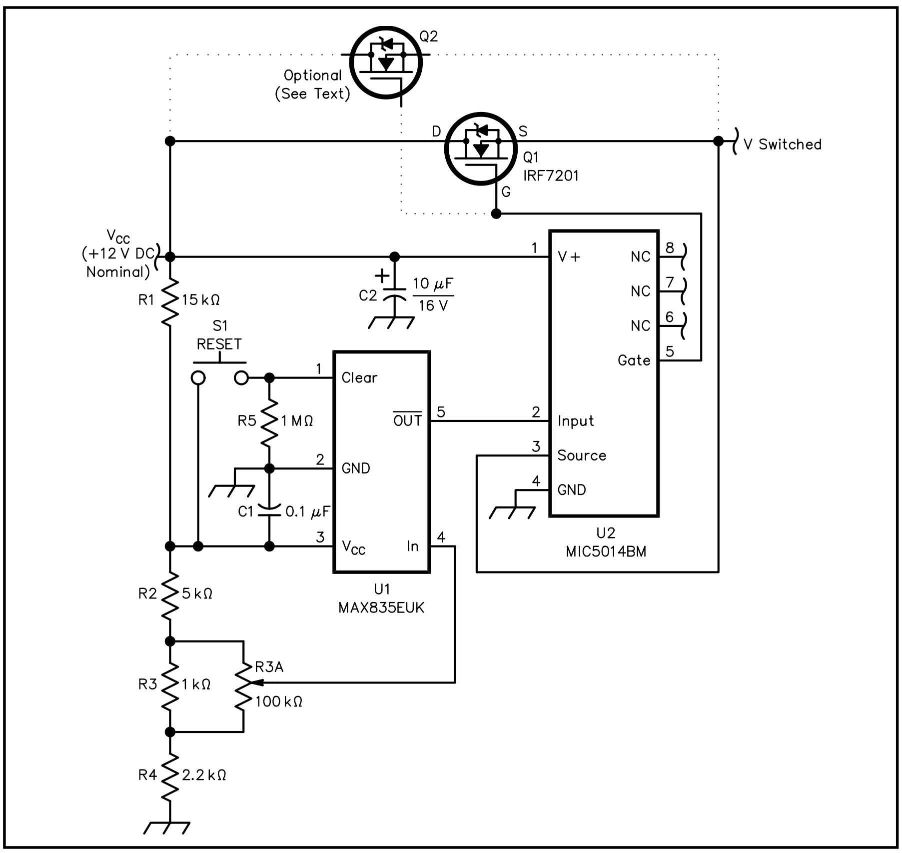


Figure 12—The low-voltage battery-protection circuit schematic. Unless otherwise specified, resistors are 1/8 W chip resistors in a 1206 package. Equivalent parts can be substituted. See Note 21 and notes and sidebar in Part 1 for parts availability.

C2—10 µF, 16 V tantalum
R3A—100 kΩ pot
S1—SPST pushbutton
Q1, Q2—IRF7201
U1—MAX835EUK
U2—MIC5014BM

[18]Notes appear on page 36.

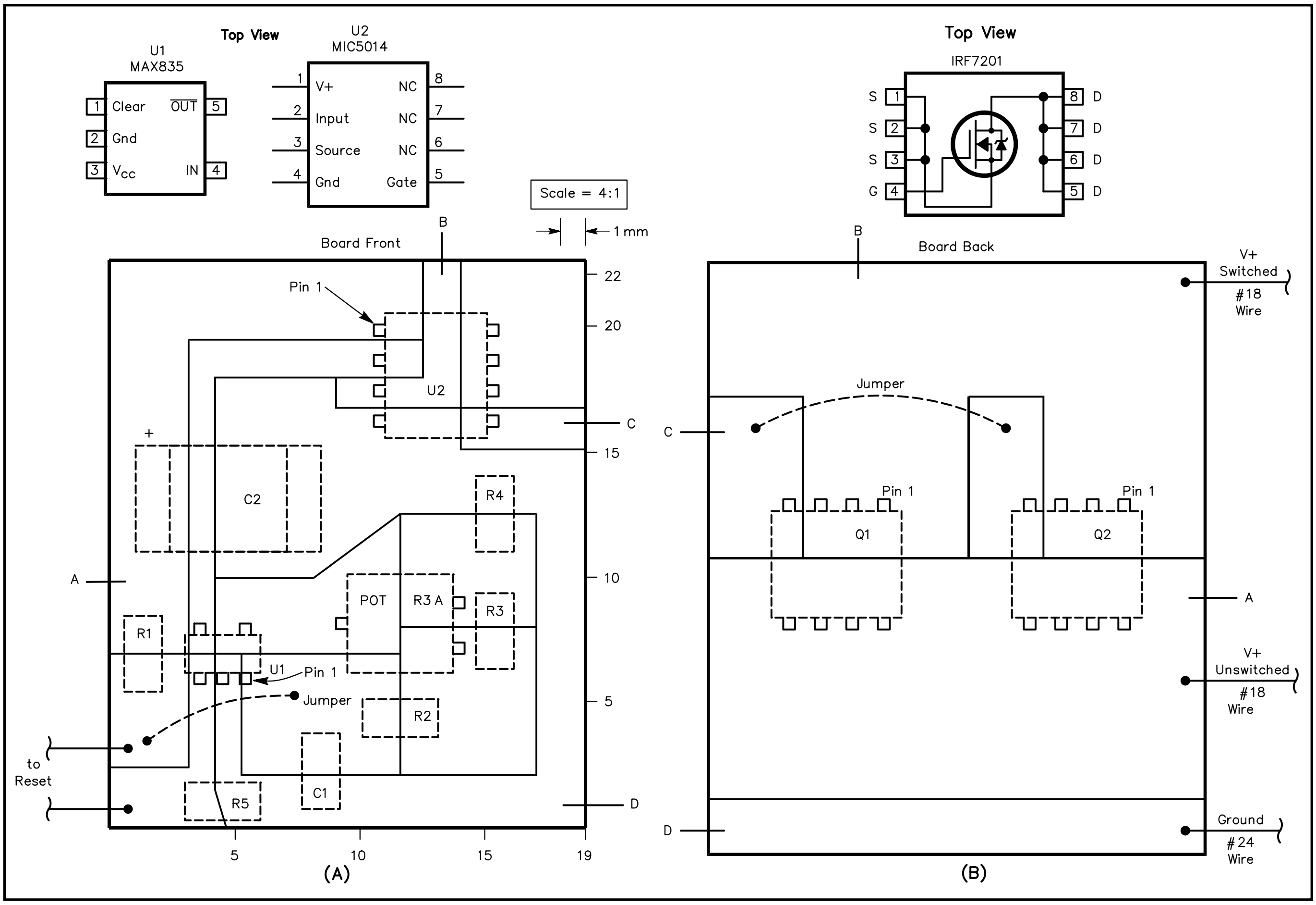


Figure 13—Part placement for the PC-board top (A) and back sides (B), respectively.

so to keep a stiff supply, I wanted the current through the voltage divider to be at least 200 μA. This calls for a total resistance of not more than 50 kΩ. As current drain is not important, I decided to use a total resistance of about 20 kΩ. Using an Excel spreadsheet, I calculated the values shown. The voltage on pin 3 is 5.3 V for V_{CC} of 15 V, and 3.8 V for V_{CC} of 11 V. I used four fixed-value resistors and a SM potentiometer (R3B) in parallel with R3 to allow better control when setting the trip level at 12.0 V. You can run U1 at a lower voltage, but keep in mind that U2 needs at least 2 V to trigger it.

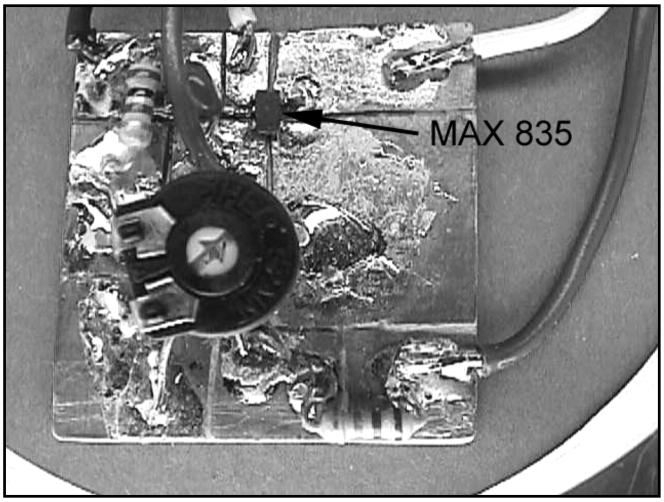
This basic circuit can be optimized for other uses. To handle more current, you need only replace the MOSFET with a more-robust one. I use an IRFZ46 and a heat sink with my ICOM IC-735. If you want to control a low-voltage NiCd-powered circuit, you could use U1 alone, connecting it directly to a logic level *low-side* N-channel MOSFET. In that case, increase the resistances of R1 to R4 for minimum current drain.

The Technology

There is a lot of new technology in this simple circuit. The entire project—including the 10 A MOSFET switch—is on a PC board smaller than the MAX8211 project. Because the quiescent current of U1 is only 2 μA—and it has a wide operating-voltage range (2.7 to 11 V)—it's possible to power it from a resistive divider rather than a 5 V regulator. U2 is a single-chip charge pump that requires no external parts. Like U1, it can operate over a wide voltage range (2.75 to 30 V) and draws only a few microamperes. U2 is designed to let low-level signals control high-voltage and high-current circuits through low resistance *N-channel* MOSFETs used on the *high side*. This arrangement is important for at least two reasons: High-side switches are usually needed with Amateur Radio applications because there is almost always more than one path to ground: the antenna, keyer, computer etc. A low-side switch would force the current to go through one of those connections rather than shutting off the rig. Second, N-channel MOSFETs have much lower resistance than equivalent P-channel MOSFETs (typically, 2.5 times less[20]). This permits the use of smaller MOSFETs for a given current.

MOSFET technology has advanced dramatically. The circuit shown uses two small SO-8 MOSFETs in parallel to control the power to my Kenwood TM-241, which draws a maximum of 11 A on high power. The two MOSFETs in parallel have an *on* resistance of only 15 mΩ (milliohms). The voltage drop across the plug connections and fuses is greater than the drop across the MOSFET! Unlike power transistors, no equalization resistors are needed when paralleling MOSFETs. That's because MOSFET *on* resistance increases with temperature, so they tend to be self-equalizing. MOSFETs make better circuit breakers than fuses or relays because they have no moving parts, are resettable, do not arc or bounce, emit less EMI and are much faster than relays or fuses. The latter can be important in an overcurrent situation. Typical MOSFET shut-down time is less than a microsecond (excluding circuit delays). The blow time on a fast-acting fuse is usually longer than 1000 μs.

U1 and U2 have families. U1 has a push-pull output; the output is internally held at either 0 or V_{CC}. The other version (MAX834) has an open-drain output that requires a pull-up resistor to provide the logic-high output. This is a common family in the SM world. One advantage of the open-drain output is that you can control a circuit with a voltage level different than the V_{CC} of the IC itself. U2's brother is the MIC5015, which operates exactly like the



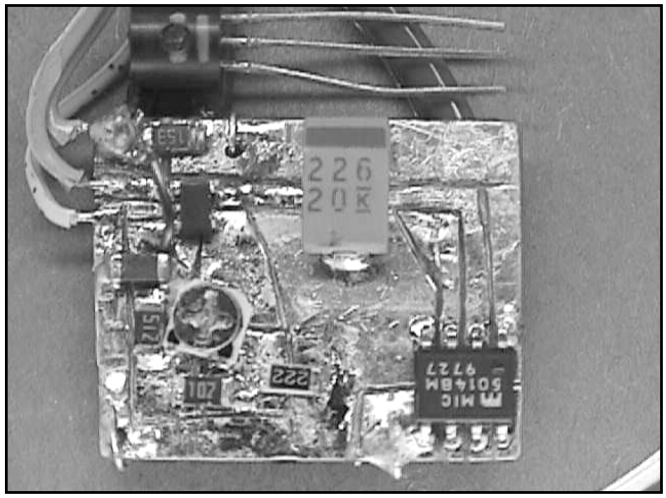


Figure 14—Close-up views of the low-voltage battery-protection switch using a Maxim MAX835 SM IC. At A (left) is a trial board made using the MAX835 and non-SM parts. The MAX835 can be seen above the large pot and to the right of the 1/4-W resistor. At B (right) is the top side of a PC board made using the hobby tool and all SM parts. The MAX835 is above the SM pot and to the left of the tantalum capacitor. The board size can be compared to the TO-92 case transistor above it. The MIC5014 is at the bottom right of the board. The SM MOSFETs are on the bottom side of the board and not shown.

'5014, but 0 V at the input turns it on, and a high level turns it off.

Making the PC Board

Figures 13A and B show the part placement for the top and back sides of the PC board for this project.[21] Before I made the all-SM version (shown in Figure 14B), I built a quick-and-dirty prototype using two PC boards. Except for the ICs, I used all standard-size leaded parts (one side of one of these boards is shown in Figure 14A), so the only critical cutting area was around the IC; all the rest was old-fashioned pad construction such as used in Project 0A. If you want to use an SM-only IC but don't need small size, this is an easy way to do it. You can also add solid wires as leads from the board and plug the entire circuit into a protoboard to use the subcircuit in a larger through-hole circuit. If you realize that SM projects needn't require *only* SM devices, experimentation becomes easier.

Once more, four jumpers (A, B, C and D) make connections between the top and bottom sides of the board where necessary. Because the board has parts on both sides, soldering is a little trickier than dealing with a board with parts mounted only on one side. Once you solder parts to one side of the board and turn the board over to solder the backside parts, it won't lie flat. Here's where a small vise can help by holding the board steady. I place parts on the more-congested side first. It isn't difficult to use the two SO-8 chips because they are so large; soldering SOT-23 packages would be more of a challenge.

Tune In Again

Before we wrap up this series next month, we'll look at a project with a large number of small parts mounted on both sides of the board. This project is one you can use as an appeasement gift to your loved ones for spending so much time at the workbench!

Selecting A MOSFET for Power Control

The MIC5014 can drive just about any N-channel MOSFET. Which MOSFET you use depends on your current load. My circuit uses two small SO-8 MOSFETs in parallel. Although the specs show the maximum current for each as 7 A, a check of the I^2R (power) loss and thermal resistance shows that 4 or 5 A is a more reasonable amount when the chip is mounted on a small PC board. Using two MOSFETs in parallel, the circuit has no problem passing 10 A continuously. When selecting a different MOSFET, *calculate its heat loss* and don't be fooled by the maximum-current figure which—for nonsurface-mounted MOSFETs—is achievable only with a perfect heat sink.

The data sheets give the thermal resistance as temperature rise per watt of heat dissipated in the MOSFET (°C/W). For surface-mount MOSFETs, the data sheet gives a single number: junction-to-ambient thermal resistance. For the IRF7201 used in the project, that is 50°C /W. At 10 A total, each MOSFET carries 5 A, and the I^2R loss is $25 \times 0.030 = 0.75$ W, giving a temperature rise of 37.5°C (100°F) above room temperature.

For nonSMT MOSFETs, the junction-to-ambient figure applies *only* if you are *not* using a heat sink. To gain the most from the MOSFET, you need to use a heat sink. In this case, you can determine the thermal resistance by adding the thermal resistances of the junction to case, the case to sink and the sink to ambient, which depends on the heat sink used. (This is just like electrical circuits: Resistances in series are added.) When in doubt, try a heat sink and see if things get hot. If so, use a larger heat sink, or add another MOSFET in parallel to reduce the current through each one.*—*Sam Ulbing, N4UAU*

*If you want to use a junk-box MOSFET, be sure to check its *on* resistance as it will probably be much higher than those of the MOSFETs I am using. The IRF510, a common MOSFET in a TO-220 package, has an *on* resistance of 0.54 Ω. Even with a rather large heat sink, the *maximum* current it can pass is about 4 A.

Notes

[18]Parts 1 and 2 of this series appear in the April and May 1999 issues of *QST*, pages 33-39 and 48-50, respectively.

[19]Donald G. Varner, WB3ECH, "A Battery-Voltage Indicator," *QST*, October 1998, pp 50-51.

[20]Micrel Applications Note 5 (Micrel, 1849 Fortune Dr, San Jose, CA 95131; tel 408-944-0800; **http:www.micrel.com**).

[21]If you are interested in learning to make your own boards as described in this series, I have a limited number of parts kits available. Each consists of a 3 × 6-inch double-sided, copper-clad board, eight cut-off wheels (two 0.005 inch, four 0.009 inch and two 0.025 inch) and the special mandrel recommended for use with the ultra-fine cut-off wheels. Order from Sam Ulbing, N4UAU, 5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; **n4uau@afn.org**. Price $13. (Florida residents must add sales tax). For orders outside the US, please add $3 for shipping.

A limited number of parts kits for Project 3 are available from me for $12 without a PC board. If you want a premade PC board, add $1.50. (Florida residents add sales tax). The kit includes only one IRF7201 MOSFET. If you want to parallel more MOSFETs or try an IRFZ46, Digi-Key, Newark and other suppliers carry those parts.

You can contact Sam Ulbing, N4UAU, at 5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; **n4uau@afn.org**.

By Sam Ulbing, N4UAU

Surface Mount Technology —You *Can* Work with It!

Part 4—This month, we wrap up the series. Before we do, though, here's that project I mentioned last month…

The first three parts of this article[22] have described rather easy-to-build projects. This one is a bit more complex. If you like to experiment, you have the opportunity to tailor this project to your specific needs and optimize its operation. Build it for a loved one and impress them with your skills! If you spend as much time working on electronic projects as I do, that loved one might appreciate a little project like this made just for them!

Project 4—The Hourglass 10-Minute Timer

This month's project is a modernized version of "A Simple 10-Minute ID Timer," that appears in *The ARRL Handbook*.[23] You can use the Hourglass as an egg timer, or to remind you to move the sprinkler, or put the laundry in the dryer, or as a two-hour timer to remind your teenager it's time to get off the telephone! You start the timer by *turning it upside down*, just like a sand hourglass! As you'll see, the operations of the old and new circuits are similar, but not exactly the same.

The Old-Technology Circuit

The *Handbook* circuit (Figure 15) is specified for use with a 12 V supply, which could limit its portability and application. LM555 timer U1 is set up for a short duty cycle: 1 second *on* and 59 seconds *off*. Pin 3 of the 4017 counter, U2, triggers after 10 cycles, increasing the time delay to 600 seconds. The alarm sounds, the circuit resets and starts counting again. Ten minutes is about the maximum practical time delay of this circuit.

The New-Technology Circuit

Surface-mount technology allows us to build this month's project (including its power supply) on a board that fits inside a 35-mm film canister (see Figure 16), so it's completely portable.[24] The low voltage and current demands of the ICs allow powering the circuit with a 3-V lithium battery.

Refer to Figure 17. An RC controlled timer, U1, is routed to a counter, U2, to extend the time base to 10 minutes. When

[22]Notes appear on page 41.

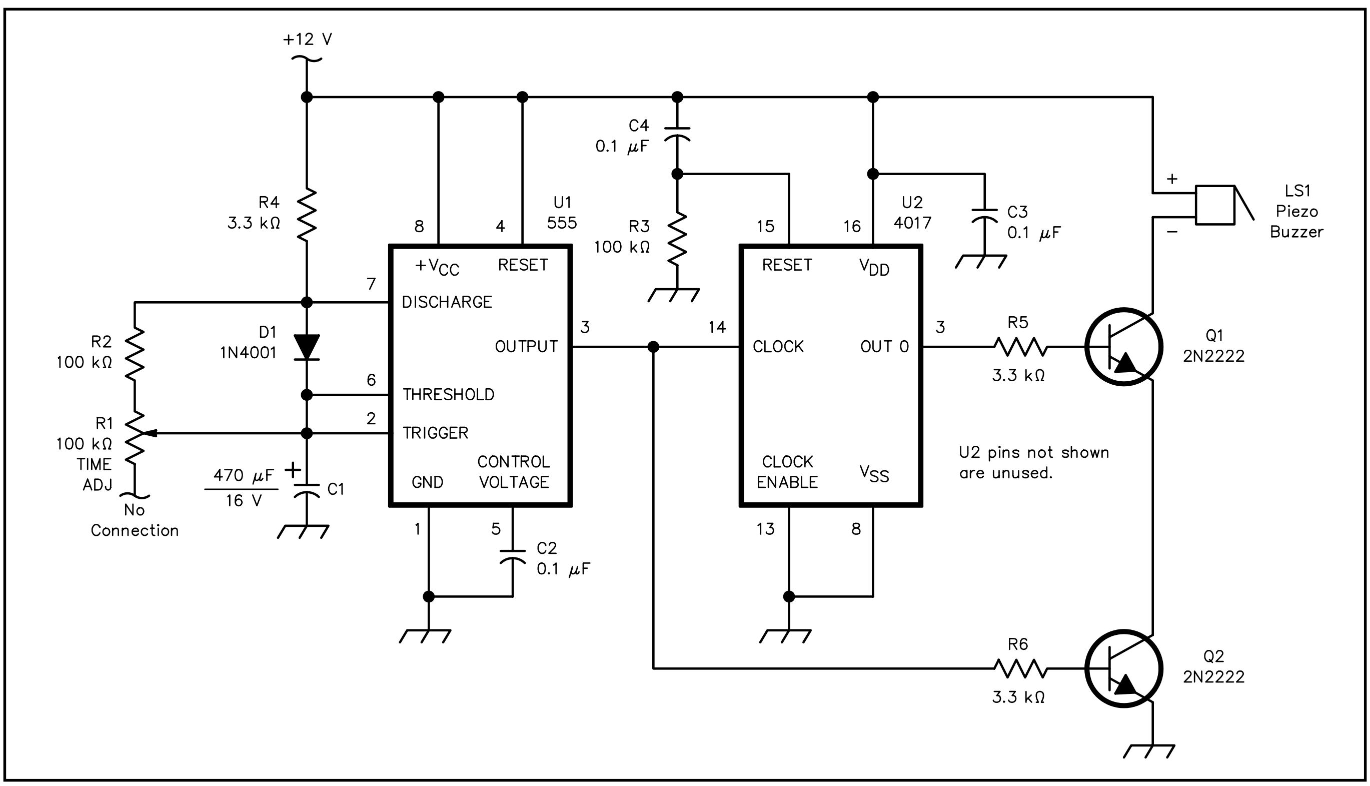


Figure 15—Schematic of the older 10-minute timer. Unless otherwise specified, resistors are $\frac{1}{4}$ W, 5% tolerance carbon-composition or film units. Equivalent parts can be substituted.

PHOTO BY LODER BROOKS, KD4AKW

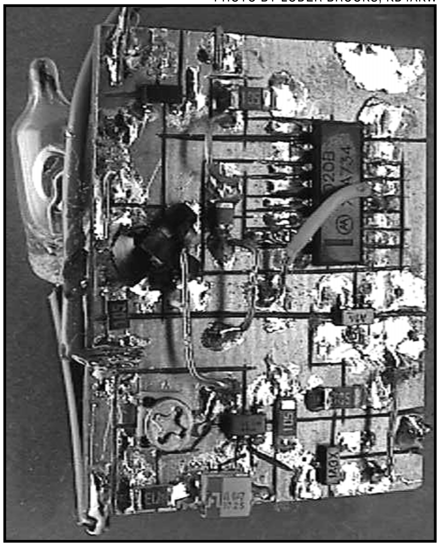


Figure 16—A top view of the SM version of the timer described in the text. The 3-V battery that powers the circuit is mounted on the bottom side of the board.

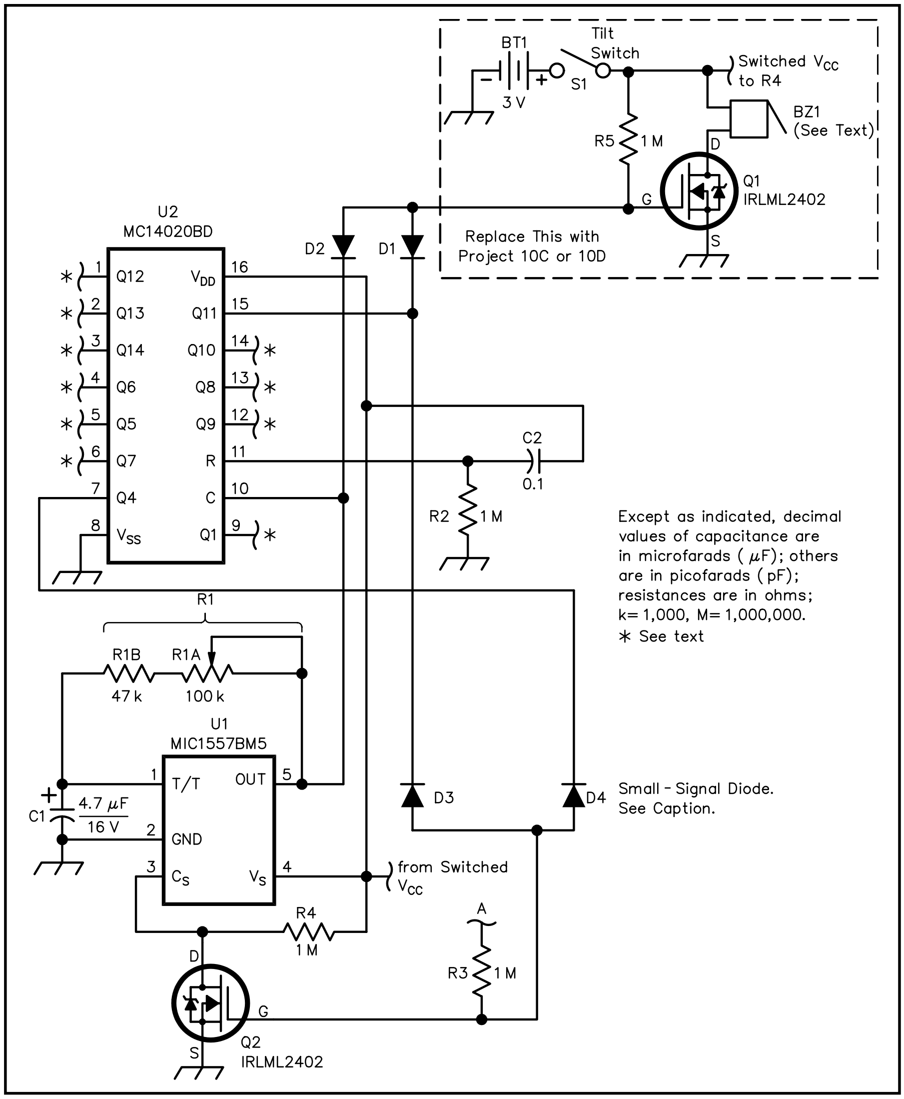


Figure 17—Schematic of the SM "hourglass" 10-minute timer. A 3-V lithium battery powers the circuit. The section of the circuit enclosed in dashed lines can be replaced by either of the circuits shown in Figures 18A and B. Unless otherwise specified, resistors are 5% tolerance SM units. The resistors I used are SM devices in 1206 cases. Equivalent parts can be substituted. See the sidebar "Manufacturers and Distributors of SMT Equipment and Parts," in Part 1, *QST*, May 1999, for a list of suppliers.

BT1—3-V, lithium CR2032, etc.
BZ1—Piezo buzzer (see text)
C1—4.7 µF, 16 V tantalum.
C2—0.1 µF ceramic (I used a SM device in a 0805 case).
D1/D2, D3/D4—BAW56LT1 (common-anode diode pairs in an SOT23 case); pairs of 1N914 or 1N4148 diodes can be substituted.
Q1, Q2—IRLM2402 MOSFET
R1A—100-kΩ pot (Bourns 3364W)
SW1—Encapsulated tilt switch (available from author)
U1—MIC1557BM5, Micrel IttyBitty RC timer/oscillator
U2—MC14020BD, 14-bit binary counter
Misc: Battery holder, Keystone #3002.

the 10 minute limit is reached, the appropriate pin on U2 goes high, turning on a switch, Q1, which sets off an alarm.

U1 of Figure 17 is an MIC1557. Dubbed the "IttyBitty RC Timer" by the manufacturer,[25] it's an SOT-23 version of the 555. R1 and C1 set the cycle time. (R1 is composed of a pot, R1A, and a fixed-value resistor, R1B.) I use a 50%-duty-cycle timer because it requires fewer parts than an asymmetrical-duty-cycle timer. I selected a cycle time of about one second because the data sheets for the LM555 and the MIC1557 indicate that capacitor leakage affects the accuracy of periods longer than 10 seconds. With just a one-second cycle time, it's necessary to use a longer delay in U2, so I added an MC14020, a 14-bit binary counter that can count up to 16,384. By using a count of 1024—and adjusting the values of R1 and C1—I achieved an accurate 10-minute delay.

This flexible circuit can be modified for longer or shorter delays, from as little as a few seconds to as long as 24 hours! (See **Experimenting with the Timer** later.) I had a difficult time finding counters in SM packages, and as you can see in the photo, the chip is "huge." (Perhaps this indicates there's a better way to handle delay circuits with SMT.)

The output at pin 15 of U2 triggers Q1 through D1. Q1 is not just any MOSFET—the IRLML2402 is a state-of-the-art device. Its gate turn-on voltage is only 1.5 V, and at 3 V, the MOSFET is fully on. (Not too many years ago, MOSFETs required 10 or 12 V to turn on. Most logic-level MOSFETs today still require 5 V, which makes them useless in a 3-V supply project.) Although the IRLML2402 is packaged in a Micro 3 package (which is smaller than an SOT-23 package), its *on* resistance is only 0.25 Ω and it can switch current levels up to 1 A.

You might ask, "Why not use a bipolar transistor instead of a MOSFET?" There are several reasons. Transistors require bias current, MOSFETs do not. A small transistor with a 30 mA load develops a 300 mV drop. The MOSFET has only a 4 mV drop, an important consideration when the supply voltage is only 3 V.[26] Also, a MOSFET can be used as a comparator. At levels less than 1 V (for this device), the MOSFET is *off*, and for levels above 1.5 V, it is *on*.

I wanted to use an **AND** condition to sound the buzzer, BZ1. D2 connects the gate of Q1 to the output of U1. This arrangement turns on the buzzer only when U2 pin 15 is positive *and* pin 5 of U1 is positive. Because the level at U1 pin 5 changes at about one cycle per second, the result is a pulsating buzzer that is more noticeable and uses less power than a continuously sounding buzzer. Another reason I could not use a transistor at Q1 is because the voltage drops of D1 and D2 result in a low voltage level of 0.6 V at Q1; that is too high to turn off a transistor.

With a battery-powered device, I didn't want the timer to cycle continually; that would deplete the battery if I forgot to shut

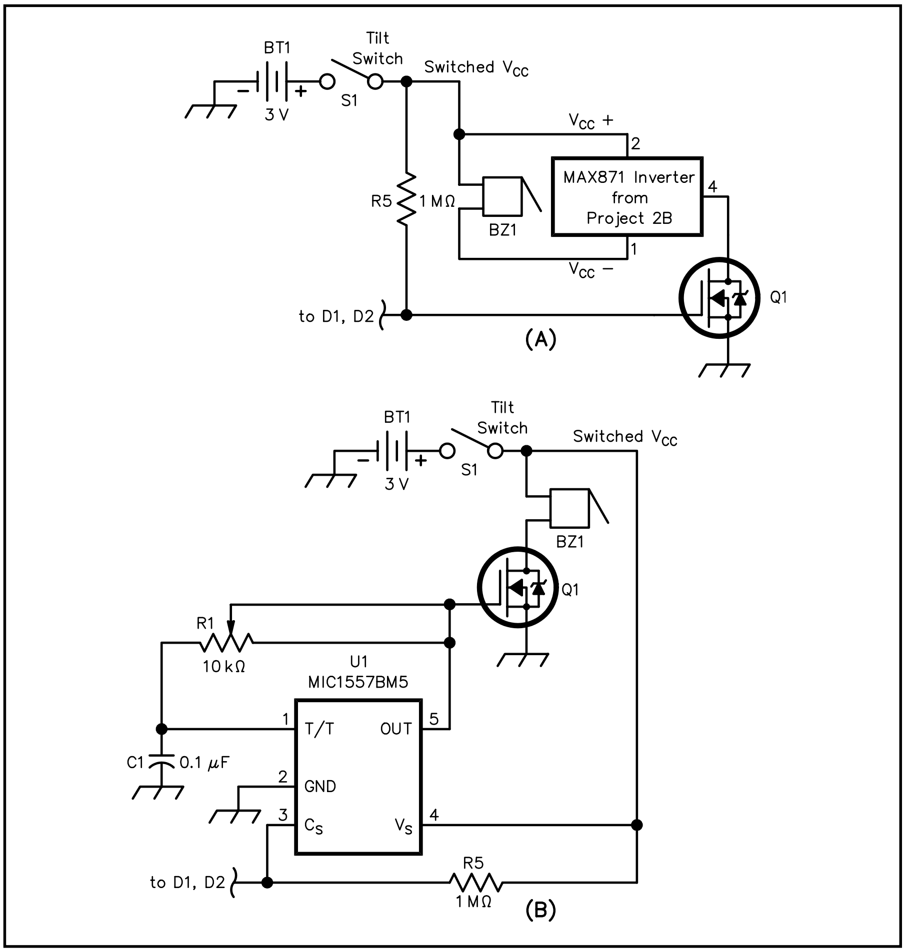


Figure 18—Two modifications you can make to the basic hourglass timer of Figure 17. At A, use of a piezo buzzer requiring a higher supply voltage (6 V) can take advantage of the MAX871 inverter circuit described in Project 2B. R5 is a 1 MΩ resistor in a 1206 SM case. An externally driven buzzer can employ the circuit shown at B using an MIC1557. Component identifications are those given in Figure 17. Note the value change of R1.

R1—10-kΩ pot (Bourns 3364W)

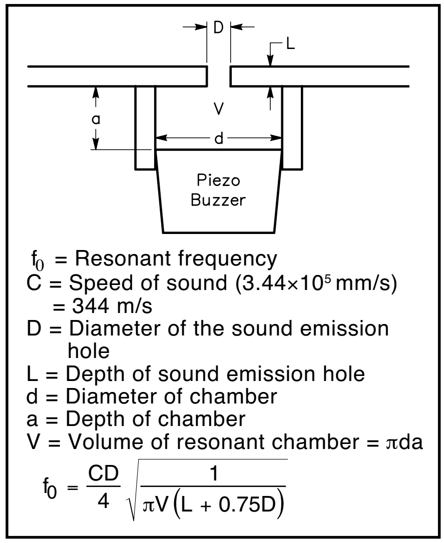


Figure 19—Here's the fundamental approach to constructing a resonator to improve the loudness and purity of the buzzer's tone. The resonant frequency (f_0) should equal twice the frequency of the buzzer to increase sound pressure. Do not make dimension D too small, or the acoustic resistance will increase. The equation is a starting point; experimentation will optimize your results (see text). Dimensions are in millimeters unless otherwise noted.

off the timer. This circuit shuts itself off. The buzzer sounds for about three seconds, and if it is not restarted, the circuit goes to sleep. It works like this: D3 and D4 form an **AND** gate controlling Q2. After pin 15 of U1 goes high and the buzzer sounds, the timer continues to count until U2 pin 7 also goes high. Then, Q2 turns on and pin 3 of U1 goes low. Pin 3 is U1's **CHIP SELECT** pin; when it goes low, U1 stops running and its current drain is reduced to 1 μA. With U1 sleeping, its output goes low. That shuts off the buzzer via D2. Total current drain while sleeping is about 5 μA. Under these conditions, a lithium 2032 battery should last several years.

To restart the timer (from sleep mode or when it is buzzing), just turn it upside down and then right side up. The tilt switch turns the power off, then on. C2 and R2 form a power-up reset that restarts U2 at 0 with a positive pulse to pin 11 through C2.

Experimenting with the Timer

Using the right audio transducer makes a major difference in audibility. Most transducers require more than 3 V to operate. I tried a RadioShack 273-074 transducer and it worked, but its output level was quite low. One way to raise the sound level is to raise the buzzer voltage. I did that with the circuit of Project 2B, as shown in Figure 18A. Some parts catalogs list piezo transducers that are externally driven and operate at 1.5 or 3 V. (The RadioShack buzzer mentioned earlier is internally driven: It has a square-wave generator built into it). I used a piezo transducer driven by an MIC1557, as shown in Figure 18B. It has a loud signal, but I found that setting the exact frequency needed for maximum sound was tricky. The best signal I could obtain came from a TMB-05[27] buzzer that I placed in a resonator and drove with the MAX871 circuit.

A Resonator

A neat way to improve the loudness and purity of the buzzer's tone (some piezo resonators have a harsh note) is to place the buzzer in a Helmholtz resonator.[28] This is a cylinder or tube designed to resonate at a certain frequency. Every resonator I used made the sound available from the transducer louder and clearer. The information in Figure 19 can help you design a resonator. If the math bothers you, try using a simple resonator made from a half-inch water pipe PVC end cap and drill a 3-mm diameter hole in the end; it worked well for me. I ground down the material surrounding the hole to make it thinner (smaller L) and adjusted the distance (A) for maximum sound. Best results are obtained when the tube's resonant frequency is about twice that of the piezo transducer's frequency.

Other Time Delays

In Figure 17, instead of connecting D1 to pin 15 of U2, you can attach it to another pin to obtain a different time delay. Table 2 shows the delays you can achieve when using a one-second cycle time at U1. By adjusting the values of R1 and C1, you can obtain nearly any time delay you want. For the arrangement to work correctly, the U2 pin you use to trigger Q1 must have a greater number of counts than the pin you use to shut down the circuit, which is why the data in Table 2 starts at 16 counts.

If you make your own PC board, you can customize it as needed. The premade PC board (see Note 24) is designed so you can add the circuits of Figure 18A or B on a separate board to drive the buzzer.

Construction Comments

I used a 0.005-inch wheel for the critical cuts at U1, Q1 and Q2 (see Figure 20). For the other cuts, I switched to a 0.009-inch wheel. The 0.005-inch cut is so narrow that

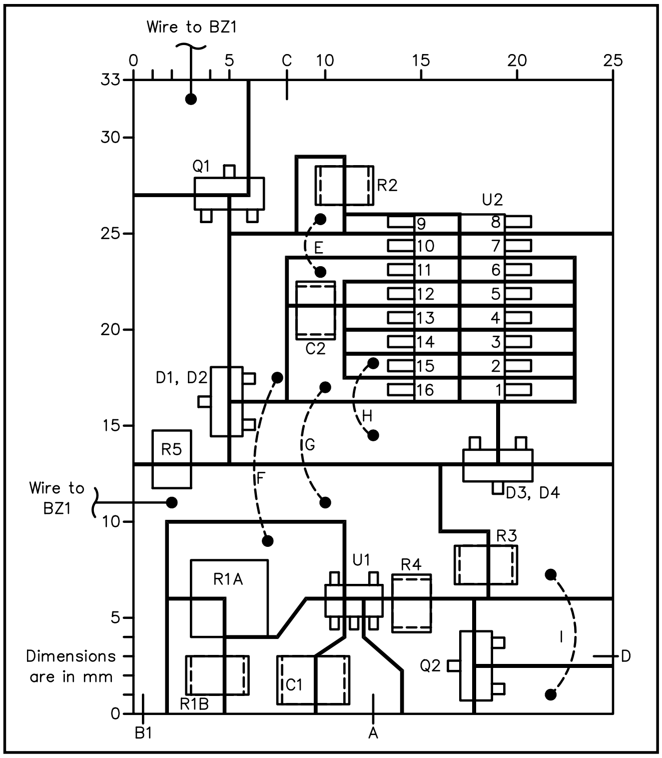


Figure 20—PC board layout of the 10-minute timer. Heavy lines designate cuts made in the foil to create component-mounting islands as described in Part 1 of this series.

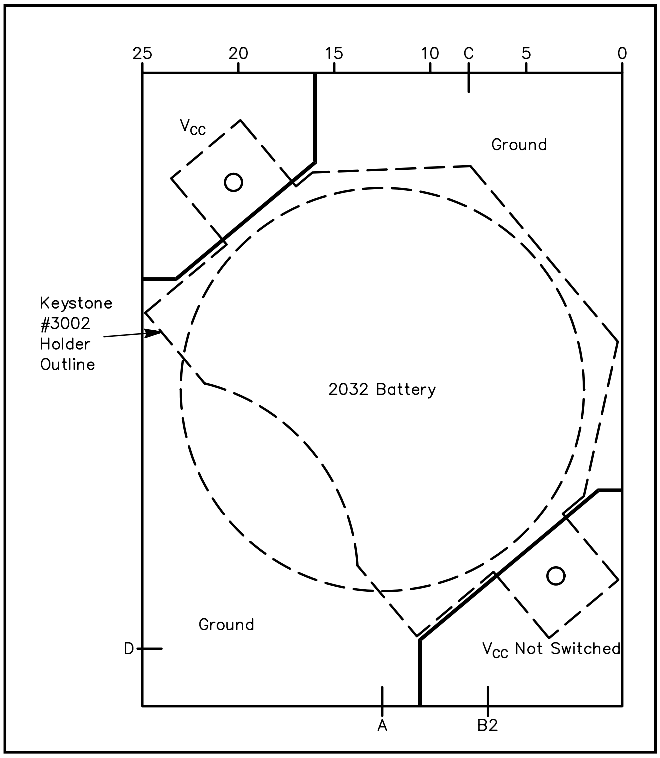


Figure 21—The 3-V battery occupies the bottom side of the 10-minute timer. Again, the heavy lines indicate where cuts are made in the PC-board foil.

solder tends to bridge the gaps. With a 0.009-inch cut, bridging is much less likely to occur. The circuit is on the board's top side; the battery and holder are on the bottom as shown in Figure 20. The tilt switch is connected between B1 of Figure 20 and B2 of Figure 21.

Summary

After completing these projects described over the past months, you should feel comfortable working with SMT devices. And, as I do, you'll probably be turning the pages of *QST* looking for a neat SMT radio project. A couple of the projects I would like to see include: a small, inexpensive VHF transceiver and a pocket-size HF receiver. In addition to the Maxim parts I mentioned earlier, Phillips Semiconductor sells a single-chip SMT AM receiver, MicroChip has an SMT microprocessor and Texas Instruments (TI) has a highly efficient SMT Class-D stereo amplifier. The parts are there. I hope we amateurs start to make use of them. [Let's see some of those projects! *QST* depends on readers and authors such as Sam and you for projects. Send your manuscripts to Steve Ford, WB8IMY, 225 Main St, Newington, CT 06111; **sford@arrl.org**.—*Ed.*]

Table 2
Timer Delay for a U1 Cycle Time of One Second

Pin	*Counts*	*Time*
5	16	16 s
4	32	32 s
6	64	1 m 4 s
13	128	2 m 8 s
12	265	4 m 16 s
14	512	8 m 32 s
15	1024	17 m 4 s
1	2048	34 m 8 s
2	4096	1 h 8 m 16 s
3	8192	2 h 16 m 32 s

Time shown in hours, minutes and seconds.

Notes

[22]Parts 1 through 3 of this series appear in the April, May and June 1999 issues of *QST*, pages 33-39, 48-50 and 34-36, respectively.

[23]R. Dean Straw, N6BV, Ed., *The 1999 ARRL Handbook* (Newington: ARRL, 1998), p 22.58

[24]A limited number of parts kits are available from me for $11, which includes all the parts (including the hard-to-find tilt switch) except for the buzzer and PC board. If you want a premade PC board, add $2 (Florida residents must add sales tax). Piezo buzzers are widely available at places like RadioShack or many of the parts sources listed in the article.

If you are interested in learning to make your own boards as I described in Part 1, I have a limited number of parts kits available. These consist of a 3 × 6-inch double-sided copper-clad board, eight cutoff wheels (two 0.005 inch, four 0.009 inch and two 0.025 inch) and the special mandrel recommended for use with the ultra-fine cutoff wheels; price: $13. (Florida residents must add sales tax.)

[25]The MIC1557 has a brother, the MIC1555, optimized for monostable operation. It is described in the same data sheet.

[26]These are the results of measurements I made.

[27]The TMB-05 internally driven buzzer is made by Star Micronics, 70-D Ethel Rd West, Piscataway, NJ 08854; tel 800-782-7636 (X512), fax 732-572-5095; **sales@starus.com; http://www.starmicronics.com/product/audio/index.cfm**. See the Star Micronics Buzzers and Transducers catalog, page 5.

[28]A resonator based on this principle is described by Wally Millard, K4JVT, "A Resonant Speaker for CW," Hints and Kinks, *QST*, Dec 1987, p 43–*Ed.*

You can contact Sam Ulbing, N4UAU, at 5200 NW 43rd St, Suite 102-177, Gainesville, FL 32606; **n4uau@afn.org**.

A DELUXE SOLDERING STATION

The simple tool shown in **Figs 8.4** through **8.6** can enhance the usefulness and life of a soldering iron as well as make electronic assembly more convenient. It includes a protective heat sink and a tip-cleaning sponge rigidly attached to a sturdy base for efficient one-handed operation.

Soldering-iron tips and heating elements last longer if operated at a reduced temperature when not being used. Temperature reduction is accomplished by half-wave rectification of the applied ac. D1 conducts during only one-half of the ac cycle. With current flowing only in one direction, only one electrode of the neon bulb glows. Closing S1 short-circuits the diode and applies full power to the soldering iron, igniting both bulb electrodes brightly.

The base for the unit is a 2 × 6 × 4-inch (HWD) aluminum chassis (Bud AC-431 or equivalent). A 30- or 40-W soldering iron fits neatly on the chassis top. The holder has two mounting holes in each foot. A sponge tray nests between the feet and the case. In this model, a sardine tin is used for the sponge tray.

The tray and iron holder are secured to the chassis by 6-32 × $\frac{1}{2}$-inch pan-head machine screws and nuts, with flat washers under the screw heads (sponge tray) and lock washers under the nuts (chassis underside). One of these nuts fastens a six-lug tie point strip to the chassis bottom. Use the soldering-iron holder base as a template for drilling the chassis and sponge tray. The floor of the sponge tray must be sealed around the screw heads to prevent moisture from leaking into the electrical components below the chassis. RTV compound was used for this purpose in the unit pictured.

Notice that the soldering iron and the soldering station use separate ac line cords. This ensures that the cord of the soldering iron will be long enough to do useful work. Bushings are used to anchor both cords. If these aren't available, grommets and cable clamps work well. Knotting the cords inside the chassis is a simple technique that normally provides adequate strain relief.

The underchassis assembly is shown in Fig 8.5. The neon bulb is installed in a $\frac{3}{16}$-inch-ID grommet. The leads are insulated with spaghetti insulation or heat-shrink tubing to prevent short circuits. If you mount the bulb in a fixture or socket, use a clear lens to ensure that the electrodes are distinctly visible. Install a cover on the bottom of the chassis to prevent accidental contact with the live ac wiring. Stick-on rubber feet prevent the bottom of the unit from scratching your work surface.

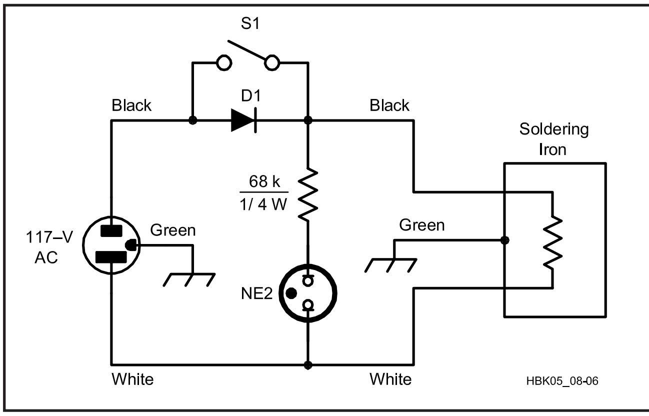


Fig 8.6—Schematic diagram of the soldering station. D1 is a silicon diode, 1-A, 400-PIV. S1 is a miniature SPST toggle switch rated 3 A at 125 V. This circuit is satisfactory for use with irons having power ratings up to 100 W.

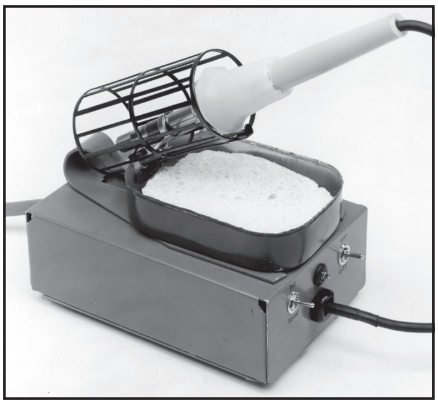


Fig 8.4—A compact assembly of commonly available items, this soldering station makes soldering easier. Miniature toggle switches are used because they are easy to operate.

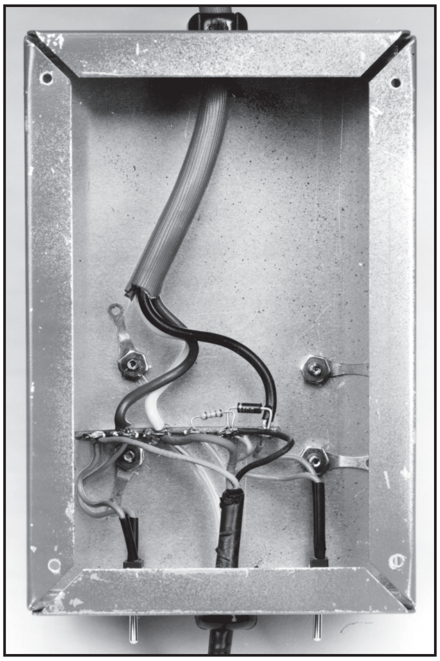


Fig 8.5—View of the soldering-station chassis underside with the bottom plate removed. #24 hookup wire is adequate for all connections. Make sure no possibility of a short circuit exists.